G-W PUBLISHER

Machining Fundamentals

10TH EDITION

JOHN R. WALKER | BOB DIXON

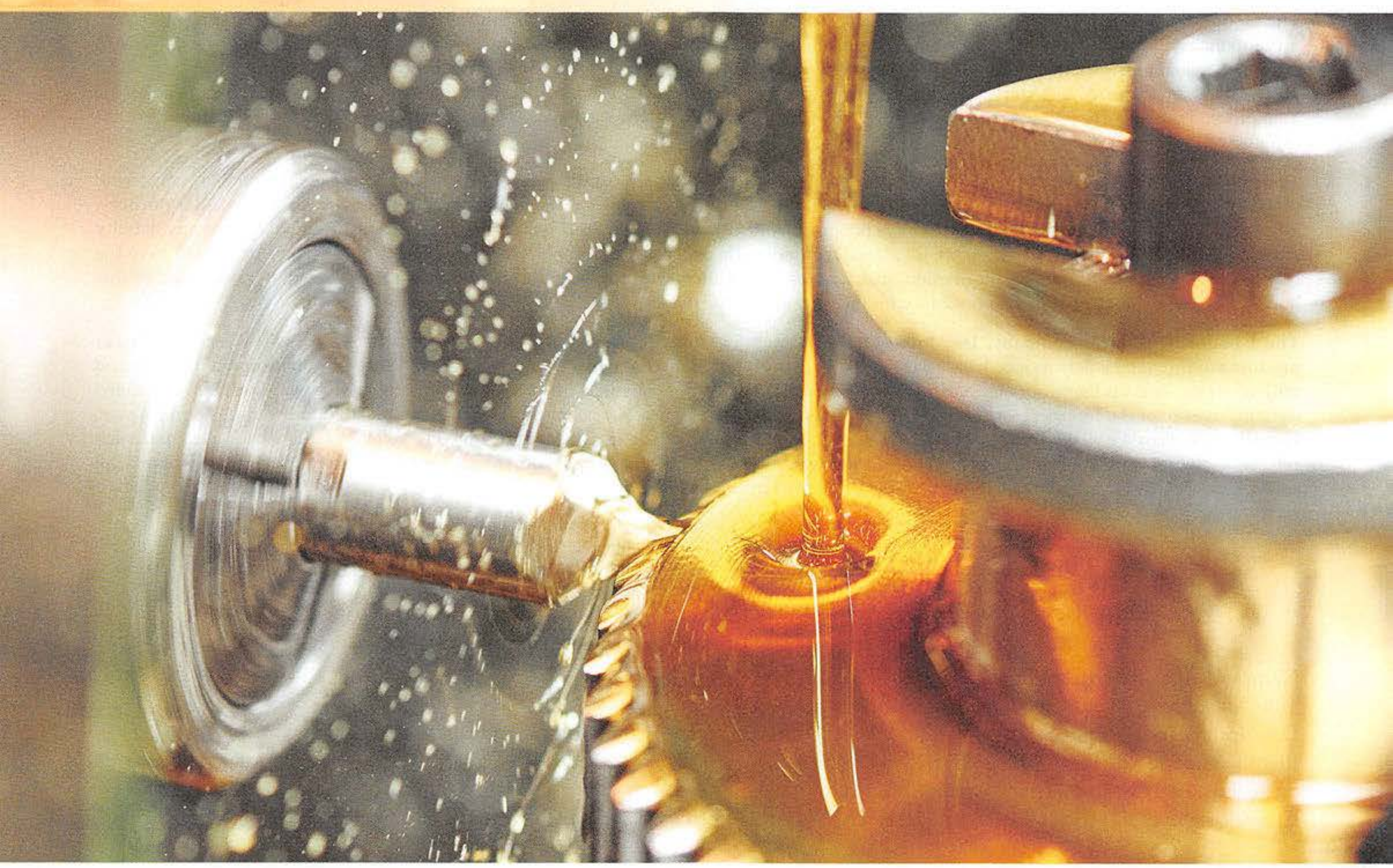
Machining Fundamentals

10TH EDITION

by

JOHN R. WALKER
Bel Air, Maryland

BOB DIXON
Walters State Community College
Morristown, Tennessee



Publisher
The Goodheart-Willcox Company, Inc.
Tinley Park, IL
www.g-w.com

Manufactured in the United States of America.

Library of Congress Catalog Card Number 2017046452

ISBN 978-1-63563-208-8

1 2 3 4 5 6 7 8 9 – 19 – 23 22 21 20 19 18

Cover image: Dmitry Kalinovsky/Shutterstock.com
Cover background: tomertu/Shutterstock.com
Back cover image: Pixel B/Shutterstock.com

Library of Congress Cataloging-in-Publication Data

Names: Walker, John R., author. | Dixon, Bob (Bob A.)
Title: Machining fundamentals / by John R. Walker, Bob Dixon.
Description: 10th edition. | Tinley Park, IL : The Goodheart-Willcox Company, Inc., [2019] | Includes index.
Identifiers: LCCN 2017046452 | ISBN 9781635632088
Subjects: LCSH: Machine-shop practice. | Machining.
Classification: LCC TJ1160 .W25 2019 | DDC 671.3/5--dc23 LC record available at https://lccn.loc.gov/2017046452

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 (such as math, science, English, print reading, and metallurgy) needed to make precision layouts and machine setups.

Machining Fundamentals provides an introduction to these important areas of manufacturing technology. The text explains the "how, why, and when" of numerous machining operations, setups, 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 more advanced processes, such as laser machining, waterjet 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. A new chapter expands coverage of geometric dimensioning and tolerancing (GD&T).

Machining Fundamentals has many features that make it easy to read and understand. The heads in each chapter are numbered to quickly locate specific information within a chapter. A chapter outline lists all chapter heads and subheads at the beginning of each chapter. Learning objectives are also presented in the chapter opener, along with a list of selected technical terms important to understanding the material in that chapter.

Throughout the text, technical terms are highlighted in bold italic type as they are introduced and defined. These terms are also listed and defined in the *Glossary* at the end of the text.

The extensive illustrations, photographs, and other visuals throughout ***Machining Fundamentals*** clarify and reinforce machining operations, procedures, and applications. A color key is used to indicate different materials and types of equipment. Features visually highlight and expand textual content by giving it practical value. *Workplace Skills* and *Career Connection* features introduce students to machining-related careers and the qualities employers are seeking. *Green Machining* features expose students to recent trends in environmentally friendly manufacturing.

Each chapter closes with a chapter review containing a summary and review questions. The summary reiterates and expands on the learning objectives given in the chapter opener. Review questions reinforce key learning objectives and offer students the opportunity to check their understanding.

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. Students will gain a strong foundation in machining to support practical skills.

About the Authors

John R. Walker is the author of thirteen textbooks and has written numerous magazine articles. Mr. Walker completed his undergraduate studies at Millersville University and has a master's degree in Industrial Education from the University of Maryland. He taught industrial arts and vocational education for more than 32 years, including 5 years as Supervisor of Industrial Education. He also worked as a machinist for the US Air Force and as a draftsman at the US Army Aberdeen Proving Grounds.

Bob Dixon is a Professor and Head of the Engineering Technology Department at Walters State Community College in Morristown, Tennessee. Dr. Dixon holds bachelor's and master's degrees in Engineering Technology from East Tennessee State University, a master's degree in Industrial Engineering from the University of Tennessee, and a doctorate in Educational Leadership from East Tennessee State University. Prior to entering the education field, Dr. Dixon spent over 20 years in industry working in a variety of machining, manufacturing, and engineering positions. He is an ATMAE Certified Senior Technology Manager and recipient of the 2005 ATMAE Outstanding Faculty of Industrial Technology Award for Region 3.

Reviewers

The authors and publisher wish to thank the following industry and teaching professionals for their valuable input into the development of ***Machining Fundamentals***.

Terry Anselmo
Sun Area Technical Institute
Northumberland, Pennsylvania

Marte Arreola
British Columbia Institute of Technology
Burnaby, British Columbia

George Beauchemin
Bay Path Regional Vocational High School
Worcester, Maryland

Paul Bois
Milford High School and Applied Technology Center
Somersworth, New Hampshire

Mark Bosworth
Southwestern Illinois College
Belleville, Illinois

Craig Brazil
Sheridan College
Oakville, Ontario

Michael Buechler
Santa Ana College
Santa Ana, California

Jason S. Carpenter
Tennessee College of Applied Technology
Morristown, Tennessee

Dr. Chesley Chambers
Floyd County College & Career Academy
Cedartown, Georgia

Donald Dean
C-TEC
Newark, Ohio

Rose M. DeNunzio
Eastfield College, Dallas County Community College District
Mesquite, Texas

Bruce Dirks
Owatonna High School
Owatonna, Minnesota

David Eledge
Metropolitan Community College
Council Bluffs, Iowa

Gottfried Georgi
William Horlick High School
Mount Pleasant, Wisconsin

John Glenn
Clinton Technical School
Clinton, Missouri

Howard Gray
Sault College
Sault Ste. Marie, Ontario

Richard Grossen
Blackhawk Technical College
Cottage Grove, Wisconsin

Garth Hansen
Dixie Applied Technology College
Saint George, Utah

Henry Hatem
Renton Technical College
Carnation, Washington

Troy Hayes
TCAT Crossville
Crossville, Tennessee

Michael Hilber
Anoka-Hennepin Secondary Technical Education Program
East Bethel, Minnesota

Jeremy Hodkiewicz
Shawano High School
Clintonville, Wisconsin

Roger Holland
Minnesota State College Southeast
La Crescent, Minnesota

Bruce Jacobs
Mt. Bauer Middle School
Puyallup, Washington

Bart Jenkins
Georgia Northwestern Technical College
Rome, Georgia

Stephen B. Jenkins
Parkside High School and Wor-Wic Community College
Salisbury, Maryland

Mike Koppy
Lake Superior College
Hermantown, Minnesota

David Laton
Alabama Department of Postsecondary Education
Montgomery, Alabama

Danny Lester
Somerset Community College
Somerset, Kentucky

Tom Livingstone
The Pennsylvania College of Technology
Williamsport, Pennsylvania

William J. McCracken, Jr.
Colorado Mesa University/Western Colorado Community College
Grand Junction, Colorado

Dale Miller
Lebanon County Career and Technology Center
Lebanon, Pennsylvania

Daniel J. Miller
SUNY Canton
Canton, New York

Daniel Morales
South Texas College
McAllen, Texas

Brian Nelson
Dunwoody College of Technology
West Saint Paul, Minnesota

James Niehaus
Southern Indiana Career & Technical Center
Evansville, Indiana

Shan Packer
Central Mountain High School
Howard, Pennsylvania

Samuel Ray
The Williamson Free School of Mechanical Trades
Saint Michaels, Maryland

Edward Schmidt
Trenton Central High School
Philadelphia, Pennsylvania

Preet Singh
Donna High School
Pharr, Texas

Bruce Smith
Tulsa Technology Center
Claremore, Oklahoma

Marty Snagg
Heritage High School
Conyers, Georgia

Paul Sorenson
Minnesota State College Southeast
Winona, Minnesota

James Standing
Algonquin College
Ottawa, Ontario

Steve Stayton
Colbert County Schools
Tuscumbia, Alabama

Daniel Wagner
HACC, Central Pennsylvania's Community College
Hegins, Pennsylvania

Roger Wensink
Lakeshore Technical College
Sheboygan Falls, Wisconsin

Olaf Wick
Wisconsin Indianhead Technical College
New Richmond, Wisconsin

Dr. John Wren
Fanshawe College, St. Thomas Campus
St. Thomas, Ontario

Dr. John Wyatt
Mississippi State University
Starkville, Mississippi

Nick Yates
Patterson High School
Baltimore, Maryland

Jim Zeleny
Confederation College
Thunder Bay, Ontario

Andy Zwanch
Johnson College
Scranton, Pennsylvania

Acknowledgments

The authors and publisher would like to thank the following companies, organizations, and individuals for their contribution of resource material, images, or other support in the development of ***Machining Fundamentals***.

3D Systems
3M Company
Alden Corp.
American Foundrymen's Society
American Society of Mechanical Engineers
AMT—The Association for Manufacturing Technology
Autocon Technologies, Inc.
Baldor
Bethlehem Steel Co.
BIG Kaiser Precision Tooling Inc.
Bill Hannan
Bird-Johnson Company
Bob Walker
Bridgeport Machines, Inc.
Buehler, a division of ITW Company
C. & E. FEIN GmbH
Carboloy
Carl Zeiss, Inc.
Carpenter Steel Co.
Chicago-Latrobe
Chick Machine Tool, Inc.
Cincinnati Milacron
CITCO Div., Western Atlas, Inc.
Clausing Industrial, Inc.
CNC Software, Inc.
Coated Abrasive Manufacturers Institute
Cogsdill Tool Products, Inc.
Compositek Corporation
Dake Corporation
Dapra Corporation
Delcam International
DoALL Co.
duMont Corp.
DuPont Co.
Engis Corporation
EZFeatureMILL—Engineering Geometry Systems
Fanuc Robotics North America, Inc.
Federal Products Co.
FEIN Power Tools, Inc.
Flow International Corp.
Ford Motor Co.
Fowler High Precision
General Dynamics Corp.
General Motors Corp.
Giddings & Lewis, Inc.
Greenfield Industries
Grizzly Industrial, Inc.
Haas Automation, Inc.
Hardinge, Inc.
Heli-Coil Corp.
Helisys, Inc.
Hertel Cutting Technologies, Inc.
Heule Precision Tools
Hirata Corporation of America
Honeywell, Inc., Industrial Automation and Control
Hougen Manufacturing, Inc.
Hydromat, Inc.
IBAG North America
IBM
Iscar Metals, Inc.
Jenoptik
Jergens, Inc.
Jet Equipment & Tools
John Winter
Jones & Lamson Machine Co.
Justrite Manufacturing Company
K.O. Lee Co.
Kennametal
KEO Cutters
Klein Tools, Inc.
Komet of America, Inc.
L. S. Starrett Co.
Landis Div. of Western Atlas
LeBlond Makino Machine Tool Co.
Library of Congress, Prints & Photograph Division
Light Beam Technology, Inc.
Lockwood Products, Inc.
Lovejoy Tool Company, Inc.
Lufkin Rule Co.
Lumonics Corp.

Mack Molding Co.
MAG IAS, LLC
Magnaflux Corp.
Mahr Federal Inc.
Makita USA, Inc.
Manufacturing Technology, Inc.
Marposs Corp.
Master Lock Co.
Maswerks, Inc.
Mazak
Metal Powder Industries Federation
MHI Machine Tool U.S.A., Inc.
Microphoto, Inc.
MIFCO McEnglevan Industrial Furnace Co., Inc.
Millersville University
Mitsubishi Materials USA Corporation
Mitutoyo/MTI Corp.
Morse Tool Co.
NASA
National Broach & Machine Co.
NEYTECH
Northrop-Grumman Corp.
Norton Co.
NSK America
O.S. Walker Co.
OSG Tap & Die, Inc.
PA&E Bonded Metals Division
Parker-Kalon
Parlec, Inc.
Precision Castparts Corp.
Quality Vision International, Inc.
Radyne
Rem Sales, Inc.
Renishaw, Inc.
Republic-Lagun Machine Tool Co.
Revolution Tool Company
Rofin-Sinar, Inc.
Rohm and Haas
Royal Products, Division of Curran Manufacturing Corporation
Rush Machinery, Inc.
Sandvik Coromant Co.
Sharnoa Corp.
Sharp Industries, Inc.
SpeedFam
Standard Tool Co.
Stratasys, Inc.
Sulzer
Sunnen Products Company
Tibor Machine Products
Tornos-Bechler S.A.
Tri-Tool, Inc.
Union Carbide Corp.
US Air Force
US Air Force Thunderbirds
US Amada, Ltd.
US Army
US Navy
US Navy Mass Communication Specialist 2nd Class Andrea Perez
Valenite, Inc.
Vermont Machine Tool and the Bryant Grinder Division
W.J. Savage Co.
Waldes Kohinoor Inc.
Webber Gage Div., L. S. Starrett Co.
Webster Instrument, Inc.
Weiler Corporation
Westech Products Group/Gantrex Machine Tool Loaders
William Schotta, Millersville University
Willis Machinery and Tools Corp.
Wilson Instruments/Instron Corporation
WMW Machinery Company, Inc.
www.mini-lathe.com
Yukiwa Seiko USA, Inc.

G-W Integrated Learning Solution

Together, We Build Careers

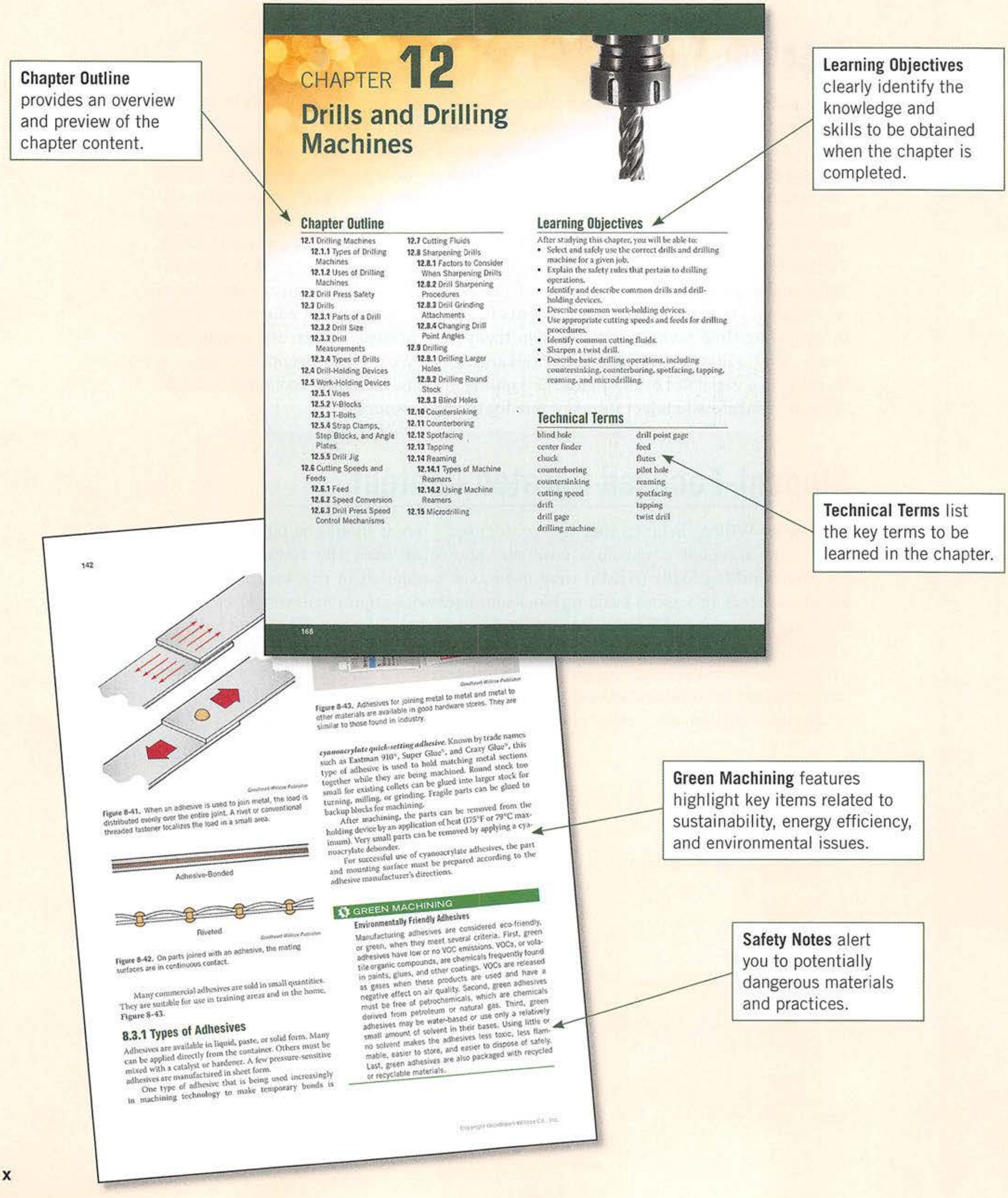
At Goodheart-Willcox, we take our mission seriously. Since 1921, G-W has been serving the career and technical education (CTE) community. Our employee-owners are driven to deliver exceptional learning solutions to CTE students to help prepare them for careers. Our authors and subject matter experts have years of experience in the classroom and industry. We combine their wisdom with our expertise to create content and tools to help students achieve success. Our products start with theory and applied content based on a strong foundation of accepted standards and curriculum. To that base, we add student-focused learning features and tools designed to help students make connections between knowledge and skills. G-W recognizes the crucial role instructors play in preparing students for careers. We support educators' efforts by providing time-saving tools that help them plan, present, assess, and engage students with traditional and digital activities and assets. We provide an entire program of learning in a variety of print, digital, and online formats, including economic bundles, allowing educators to select the right mix for their classroom.

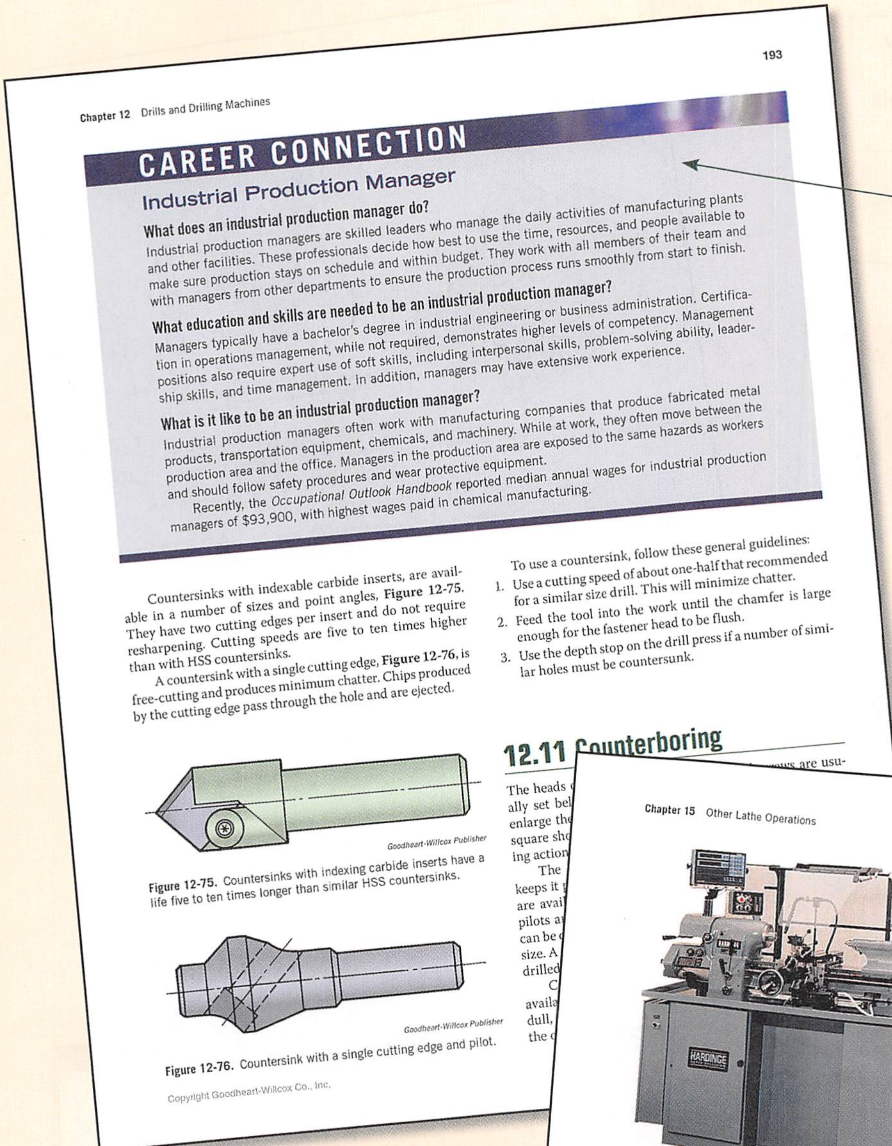
Student-Focused Curated Content

Goodheart-Willcox believes that student-focused content should be built from standards and accepted curriculum coverage. Standards from the National Institute for Metalworking Skills (NIMS) were used as a foundation in this text. ***Machining Fundamentals*** also uses a building block approach with attention devoted to a logical teaching progression that helps students build upon their learning. We call on industry experts and teachers from across the country to review and comment on our content, presentation, and pedagogy. Finally, in our refinement of curated content, our editors are immersed in content checking, securing and sometimes creating figures that convey key information, and revising language and pedagogy.

Features of the Textbook

Features are student-focused learning tools designed to help you get the most out of your studies. This visual guide highlights the features designed for the textbook.





193

Chapter 12 Drills and Drilling Machines

CAREER CONNECTION

Industrial Production Manager

What does an industrial production manager do?

Industrial production managers are skilled leaders who manage the daily activities of manufacturing plants and other facilities. These professionals decide how best to use the time, resources, and people available to make sure production stays on schedule and within budget. They work with all members of their team and with managers from other departments to ensure the production process runs smoothly from start to finish.

What education and skills are needed to be an industrial production manager?

Managers typically have a bachelor's degree in industrial engineering or business administration. Certification in operations management, while not required, demonstrates higher levels of competency. Management positions also require expert use of soft skills, including interpersonal skills, problem-solving ability, leadership skills, and time management. In addition, managers may have extensive work experience.

What is it like to be an industrial production manager?

Industrial production managers often work with manufacturing companies that produce fabricated metal products, transportation equipment, chemicals, and machinery. While at work, they often move between the production area and the office. Managers in the production area are exposed to the same hazards as workers and should follow safety procedures and wear protective equipment.

Recently, the *Occupational Outlook Handbook* reported median annual wages for industrial production managers of $93,900, with highest wages paid in chemical manufacturing.

Countersinks with indexable carbide inserts, are available in a number of sizes and point angles, **Figure 12-75.** 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 countersink with a single cutting edge, **Figure 12-76,** is free-cutting and produces minimum chatter. Chips produced by the cutting edge pass through the hole and are ejected.

Goodheart-Willcox Publisher

Figure 12-75. Countersinks with indexing carbide inserts have a life five to ten times longer than similar HSS countersinks.

Goodheart-Willcox Publisher

Figure 12-76. Countersink with a single cutting edge and pilot.

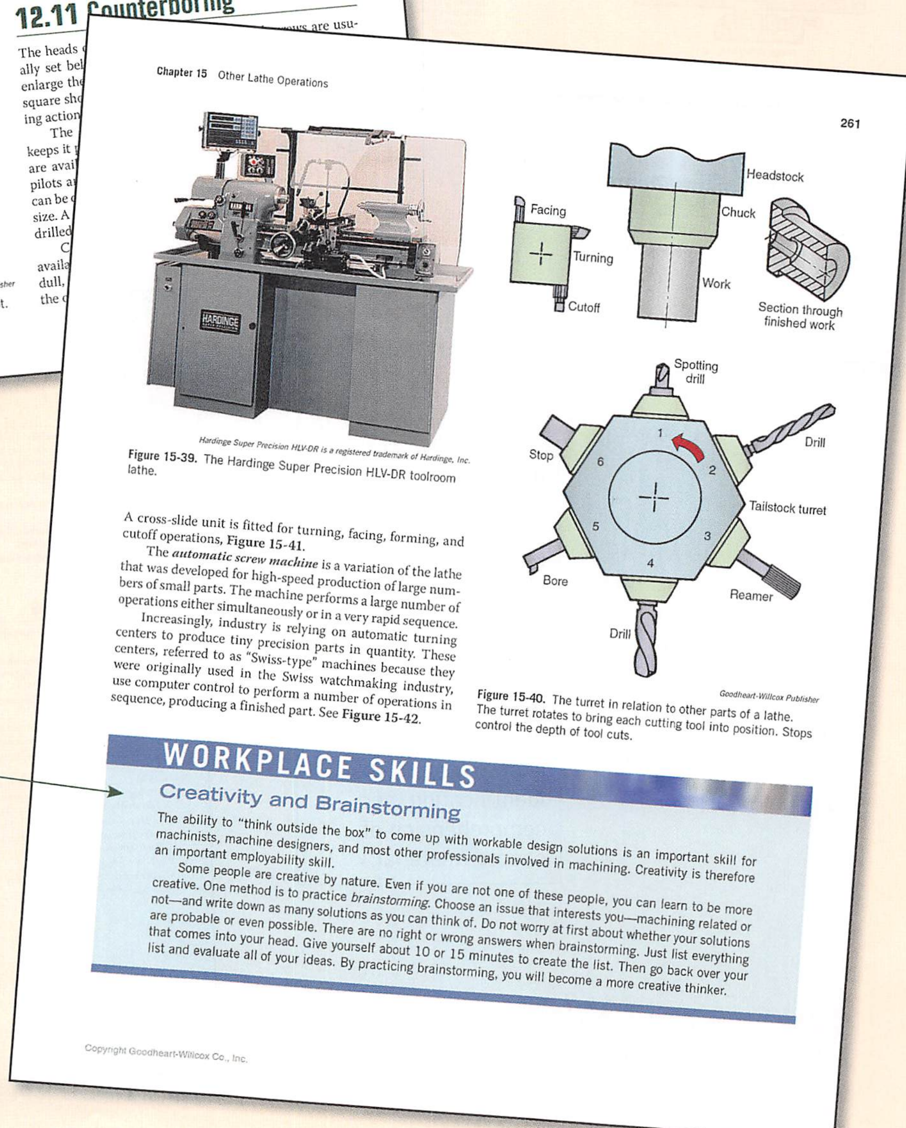
To use a countersink, follow these general guidelines:

1. Use a cutting speed of about one-half that recommended for a similar size drill. This will minimize 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.

12.11 Counterboring

Copyright Goodheart-Willcox Co., Inc.

Career Connection features and profiles can provide a path for career success.



Chapter 15 Other Lathe Operations

261

Hardinge Super Precision HLV-DR is a registered trademark of Hardinge, Inc.

Figure 15-39. The Hardinge Super Precision HLV-DR toolroom lathe.

Goodheart-Willcox Publisher

Figure 15-40. The turret in relation to other parts of a lathe. The turret rotates to bring each cutting tool into position. Stops control the depth of tool cuts.

A cross-slide unit is fitted for turning, facing, forming, and cutoff operations, **Figure 15-41.**

The ***automatic screw machine*** is a variation of the lathe that was developed for high-speed production of large numbers of small parts. The machine performs a large number of operations either simultaneously or in a very rapid sequence.

Increasingly, industry is relying on automatic turning centers to produce tiny precision parts in quantity. These centers, referred to as "Swiss-type" machines because they were originally used in the Swiss watchmaking industry, use computer control to perform a number of operations in sequence, producing a finished part. See **Figure 15-42.**

WORKPLACE SKILLS

Creativity and Brainstorming

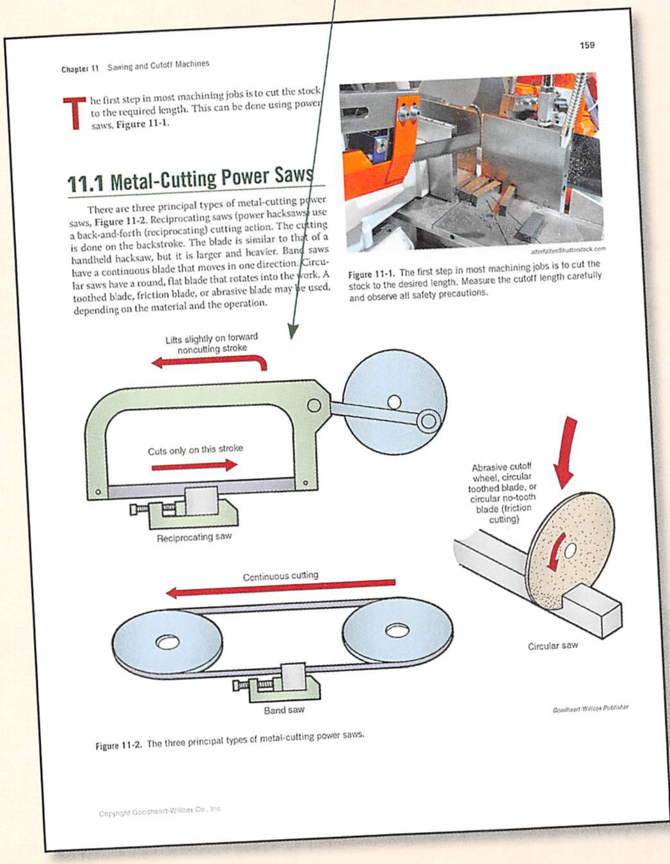
The ability to "think outside the box" to come up with workable design solutions is an important skill for machinists, machine designers, and most other professionals involved in machining. Creativity is therefore an important employability skill.

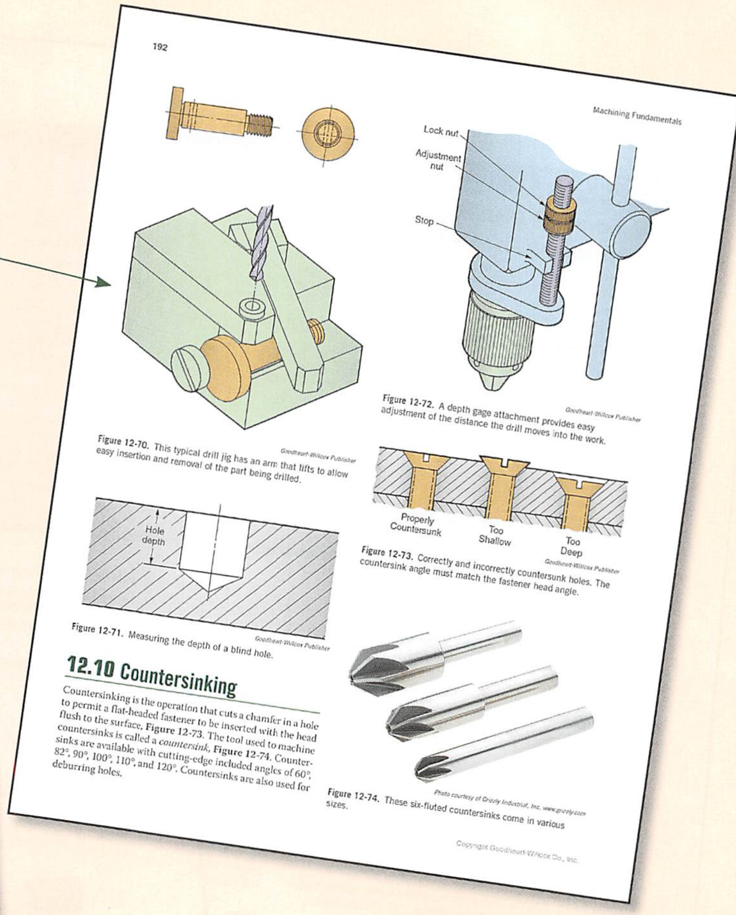
Some people are creative by nature. Even if you are not one of these people, you can learn to be more creative. One method is to practice *brainstorming.* Choose an issue that interests you—machining related or not—and write down as many solutions as you can think of. Do not worry at first about whether your solutions are probable or even possible. There are no right or wrong answers when brainstorming. Just list everything that comes into your head. Give yourself about 10 or 15 minutes to create the list. Then go back over your list and evaluate all of your ideas. By practicing brainstorming, you will become a more creative thinker.

Copyright Goodheart-Willcox Co., Inc.

Workplace Skills highlight the professional behaviors and traits that employers want.

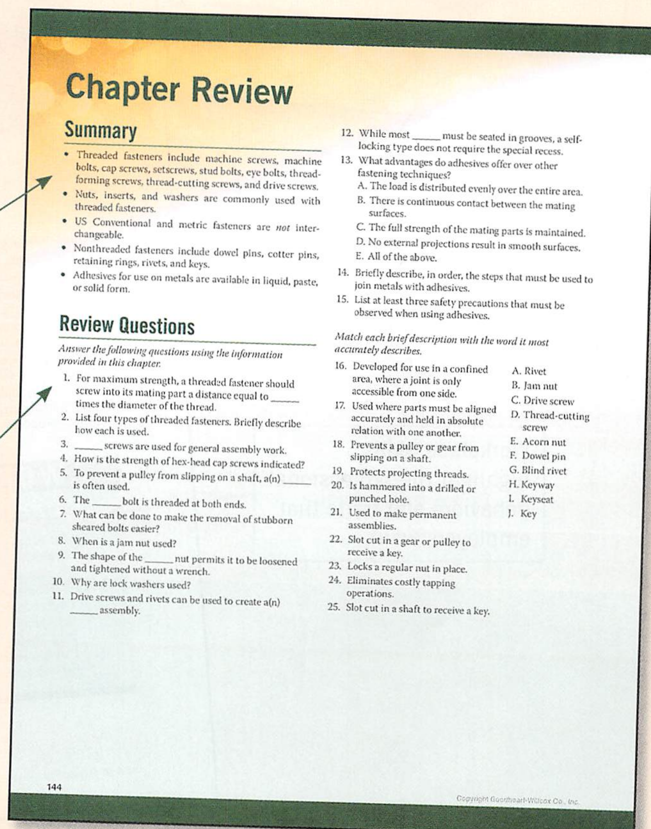
Illustrations have been designed to clearly illuminate key concepts, equipment, and methods and enhance the text with visuals for different learning styles.





Summary feature provides an additional review tool for you and reinforces key learning objectives.

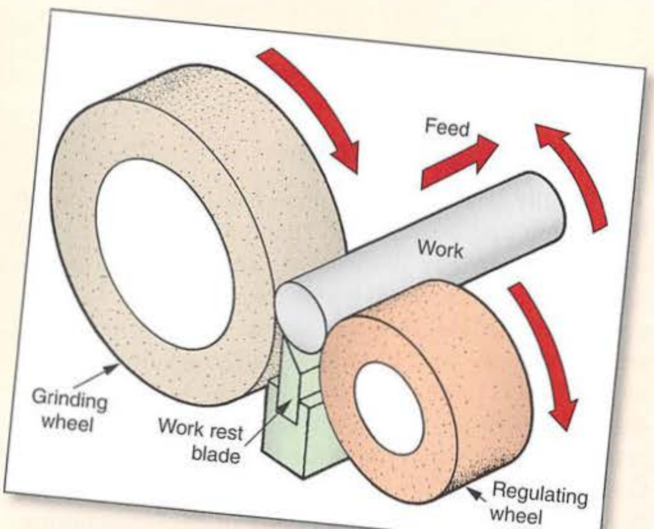
Review Questions allow you to demonstrate knowledge, identification, and comprehension of chapter material.

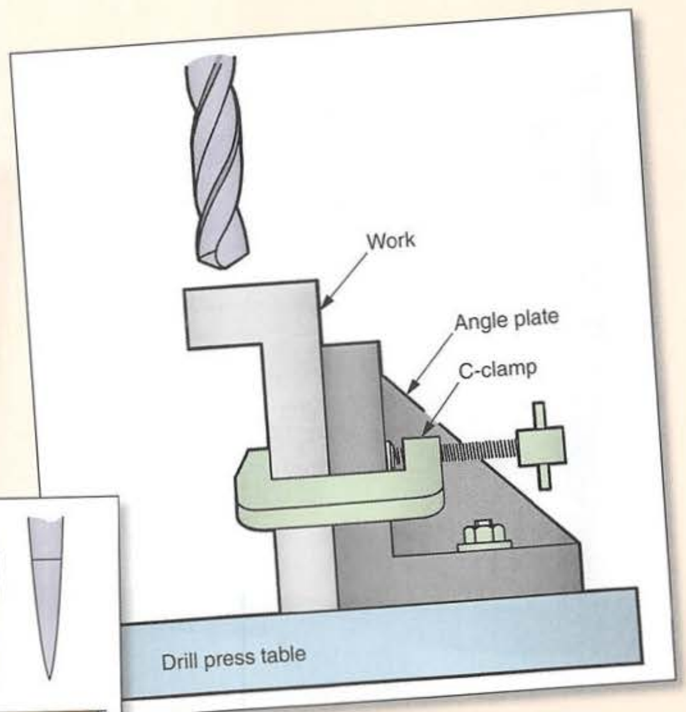


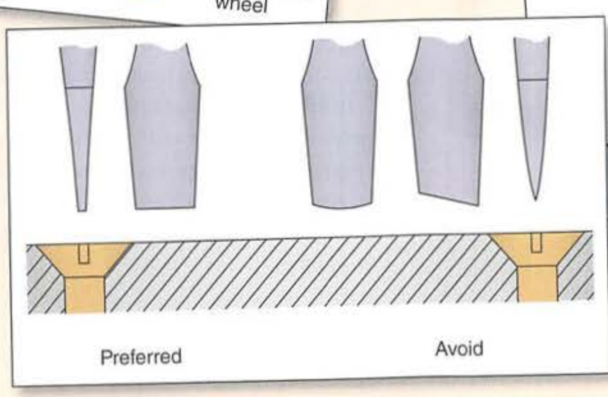
Machining Fundamentals Color Code

A consistent color code is used in the line illustrations throughout ***Machining Fundamentals*** to help you better visualize the machining operations and procedures. Specific colors are used to indicate different materials and equipment features. The following key shows what each color represents:

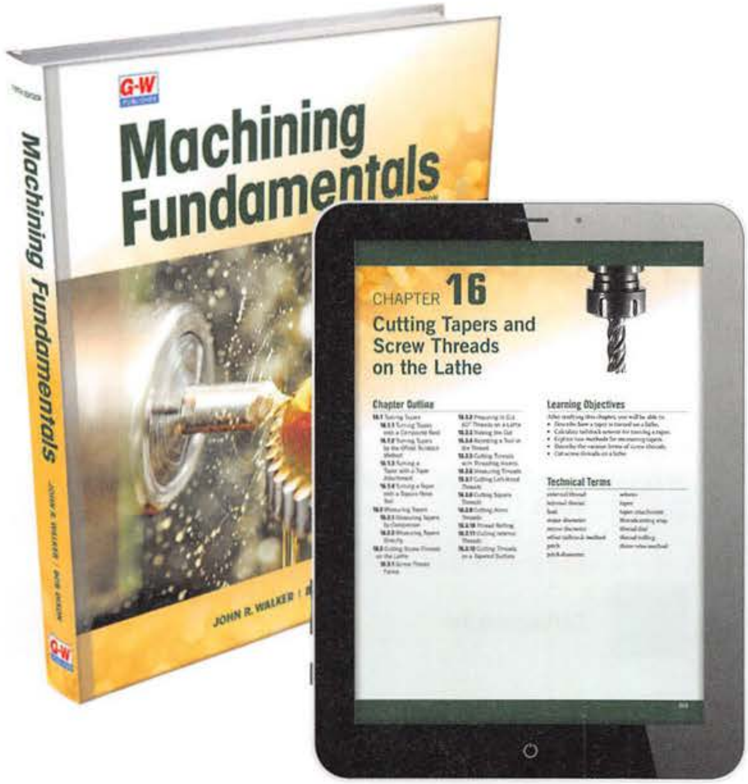
- **Metals**
- **Alternate metal**
- **Machines/machine parts**
- **Tools**
- **Cutting edges**
- **Work-holding and tool-holding devices**
- **Fasteners**
- **Abrasives**
- **Fluids**
- **Miscellaneous**
- **Direction or force arrows, dimensional information**







Student Resources



Textbook

The *Machining Fundamentals* textbook provides an exciting, full-color, and highly illustrated learning resource. The textbook is available in print and online versions.

Workbook

The student Workbook provides questions that reinforce and review textbook content. Organized to follow the textbook on a chapter-by-chapter basis, the Workbook assignments help you engage with the textbook content and aid in effective retention of key facts, ideas, and concepts.

Online Learning Suite

The Online Learning Suite provides the foundation of instruction and learning for digital and blended classrooms through any device with an Internet browser. All student instructional materials are found on a convenient online bookshelf and are accessible at home, at school, or on the go. The Online Learning Suite includes an interactive online textbook, student workbook with digital form fields, vocabulary activities, drag-and-drop activities, and a variety of other learning activities. The Online Learning Suite also contains 30 video clips that provide dynamic visual instruction of basic machining practices. Scripted by an expert machinist, shot in an actual manufacturing facility and loaded with practical hands-on demonstrations, these live-action videos help provide students with the essential knowledge and skills required for entry-level employment in today's manufacturing industry. The Online Learning Suite effectively brings digital learning to students and is easy for instructors to use.

Online Learning Suite/Student Textbook Bundle

Looking for a blended solution? Goodheart-Willcox offers the Online Learning Suite bundled with the printed text in one easy-to-access package. Students have the flexibility to use the print version, the Online Learning Suite, or a combination of both components to meet their individual learning style. The convenient packaging makes managing and accessing content easy and efficient.

Video Clip Library

Live-action videos provide students with solid information on 30 machining topics. High-quality video footage and crisp narrations enable visual learners to clearly understand the content.

Instructor Resources

Instructor resources provide information and tools to support teaching, grading, and planning; class presentations; and assessment.

Instructor's Presentations for PowerPoint®

Presentations for PowerPoint® are designed to support instructors and visually reinforce the textbook content. These time-saving and customizable presentations include objectives, key concepts, terms, and images from each chapter. Instructors can customize each presentation by modifying and adding slides and images to better meet classroom needs.

ExamView® Assessment Suite

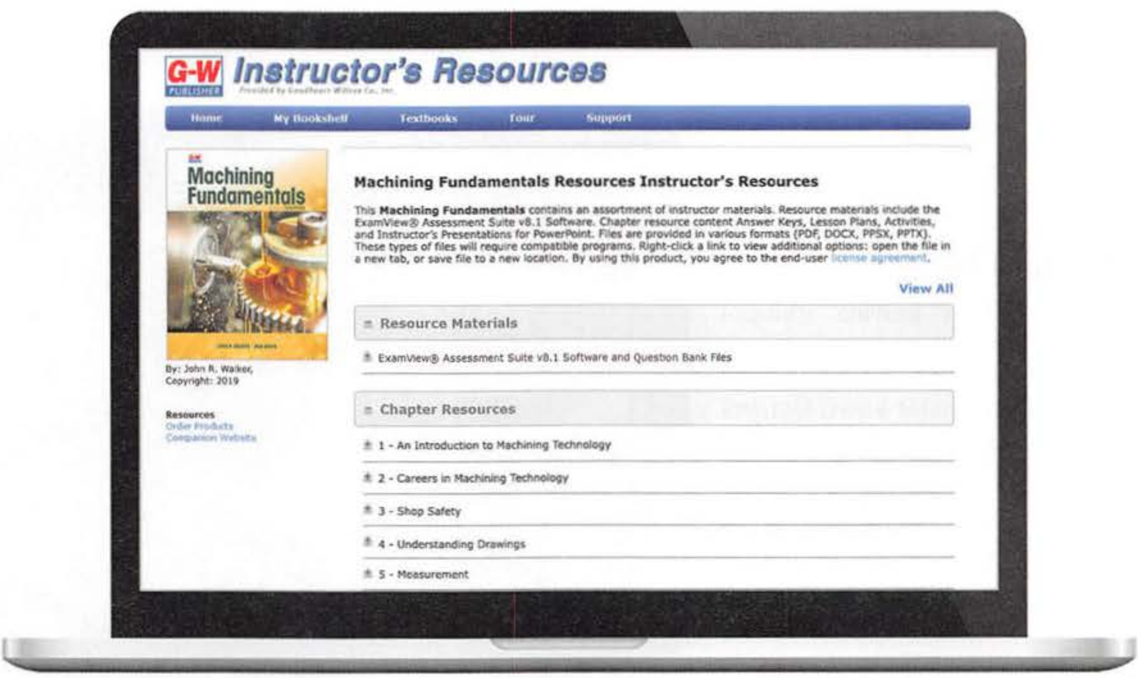
The ExamView® Assessment Suite allows instructors to quickly and easily create, administer, and score tests, both in print and online. Create many versions of formative and summative exams, multiple versions of a single test, and automatically generate answer keys. For additional versatility, instructors can easily modify existing questions and add new questions to fit course needs.

Instructor's Resource CD

A variety of time-saving teaching support tools are provided in the Instructor's Resource for ***Machining Fundamentals***. Answer keys are included for both the textbook and student Workbook. Customizable lesson plans provide chapter-specific instructional resources, tools for practice and assessment, and other resources available for teaching the chapter content. An overview of the products in the teaching package is provided, as well as correlation to NIMS Duties and Standards for Machining Skills Level I.

Online Instructor Resources

Online Instructor Resources are comprehensive, time-saving teaching tools organized in a convenient, easy-to-use online bookshelf. Lesson plans, answer keys, a correlation to NIMS Duties and Standards for Machining Skills Level I, Presentations for PowerPoint®, ExamView® Assessment Suite software, and other resources are available on demand, 24/7 from home or school.



Brief Contents

Contents

Feature Contents

WORKPLACE SKILLS

CAREER CONNECTION

GREEN MACHINING

CHAPTER 1

An Introduction to Machining Technology

Chapter Outline

1.1 The Evolution of Machine Tools
- **1.1.1** Early Machine Tools
- **1.1.2** Power Sources

1.2 Basic Machine Tool Operation
- **1.2.1** Sawing Machines
- **1.2.2** Drill Press
- **1.2.3** Grinding Machines
- **1.2.4** Milling Machine
- **1.2.5** Broaching Machines

1.3 Nontraditional Machining Processes

1.4 Automating the Machining Process
- **1.4.1** The Development of Numerical Control
- **1.4.2** Computer Numerical Control

1.5 The Evolving Role of the Machinist

1.6 Acquiring Machining Skills and Knowledge

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 machine tool operations.
- List nontraditional machining processes.
- Explain how CNC machining equipment operates.
- Describe the role of the machinist.
- Explain how machinists are trained and certified.

Technical Terms

broaching machine
computer numerical control (CNC)
drill press
grinding machine
lathe
machine tools
machinist
milling machine
numerical control (NC)
sawing machine
skill standards
turning

A study of technology will show that industry has progressed from a time when everything was made by hand to recent advances resulting in the fully automated manufacturing processes used today. 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 using the most advanced machine tools available. No industry or country can hope to take advantage of the most advanced machine tools without the aid of a ***machinist***—a person highly skilled in the use of machine tools and capable of creating the complex machine setups required for modern manufacturing.

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.

These 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 machines that can be used to manufacture other tools, including 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 evokes 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 handmade and human-powered. They have been dated back to about 1200 BC. Until the end of the seventeenth century, the lathe could be used only to turn softer materials, such as wood, ivory, or at most, soft metals such as lead or copper. 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.

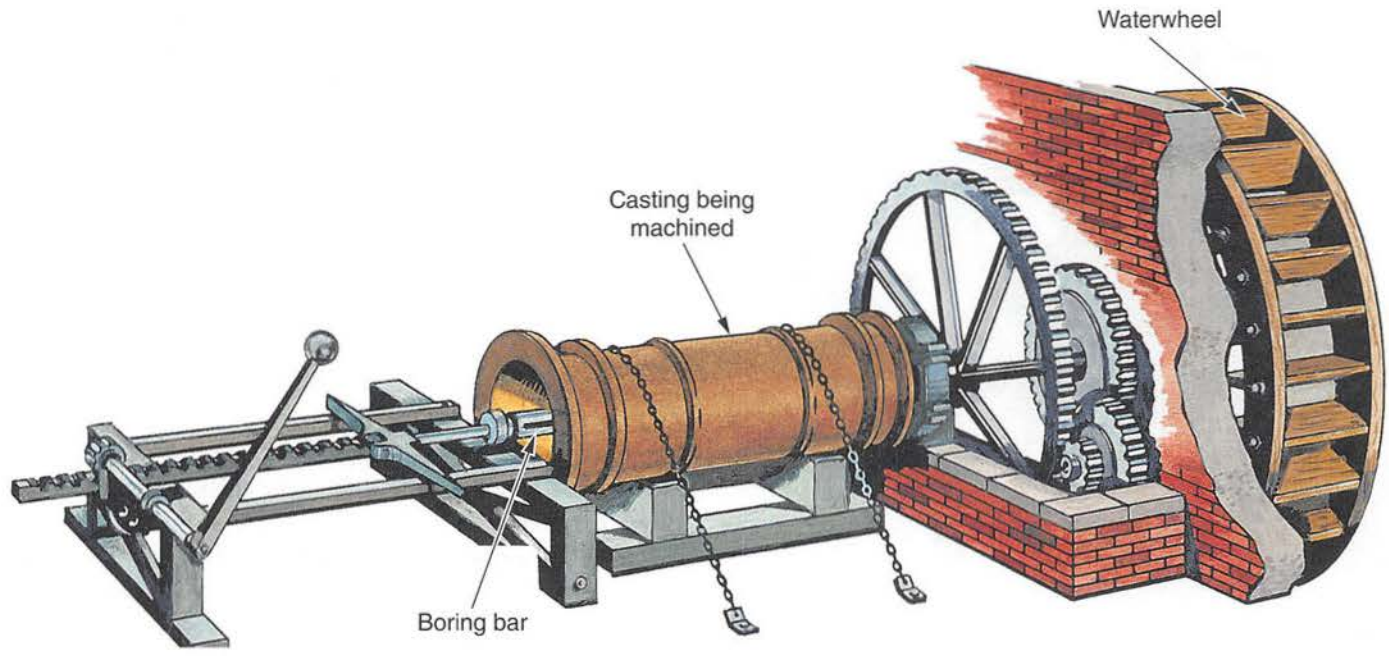
Boring Mill

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 type of 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 about the size of a modern US quarter). However, operation of the boring mill, like that of 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.

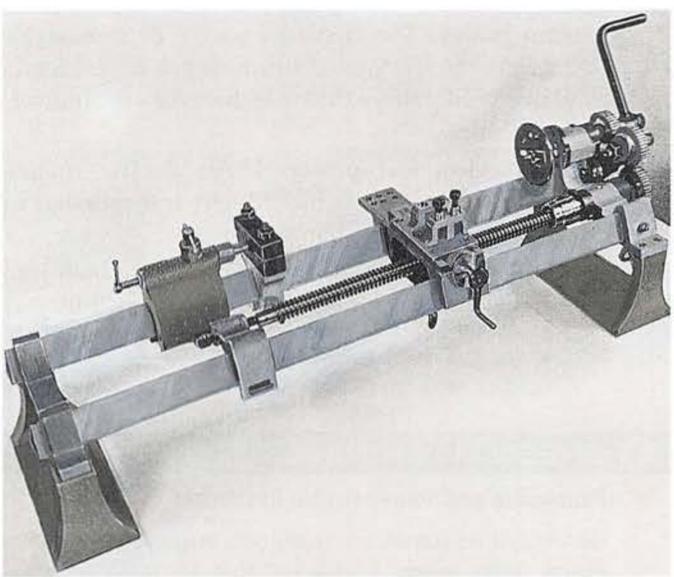
Lathe

The first lathe capable of cutting accurate screw threads was designed and constructed by Henry Maudslay, an English master mechanic and machine toolmaker, in about 1800. As shown in **Figure 1-3**, a handmade screw thread was geared



DoALL Co.

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.

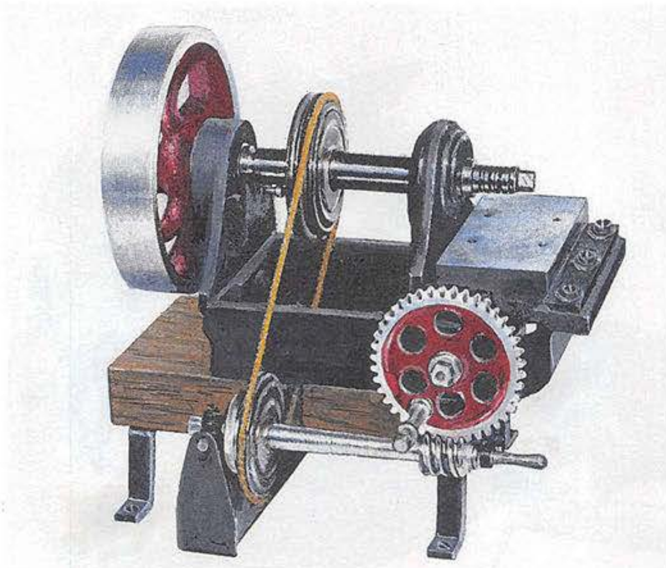
to the spindle and moved a cutting tool along the work. Maudslay also devised a slide rest and fitted it to his lathe. This allowed the cutting tool to be repositioned accurately after each cut. Maudslay's lathe is considered the "granddaddy" of all modern chip-making machine tools.

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 be located near a source of water power. This was often some distance from raw materials and workers. With cheap power, industry could be located 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 that metal could be machined with some degree of accuracy, there could be no steam engine.

Milling Machine

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, and it was soon 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



DoALL Co.

Figure 1-4. One of the first practical milling machines manufactured in America. Eli Whitney used this and similar machines to mass-produce interchangeable musket parts.

armory were not interchangeable with parts made in another armory. It was not until the mid-1860s that the United States adopted a standard measuring system.

Shaper

Another early machine tool that was important in the early machine shop was the shaper. The operation of early shapers was relatively simple. A single-point tool, much like those used in lathes, was held in a toolholder. The workpiece was clamped onto a table that was moved back and forth under the cutter. As the workpiece moved under the tool, the tool shaved material away from the workpiece. The toolholder was hinged so that, on the return stroke of the table, the tool was free to ride on top of the workpiece. When the tool cleared the workpiece, the toolholder dropped back into position. After the tool cleared the workpiece, the table moved sideways in preparation for another cutting stroke.

Later models used a hydraulic ram to move the tool in a linear motion for the cutting stroke. Another important innovation was the addition of electric limit switches that controlled the length of the stroke and the side-to-side indexing distance.

The primary use of the original shaper was to machine keyway slots and dovetails for linear slide control. It was also used to cut splines and teeth into gears. The addition of limit switches allowed the shaper to be used to clean up and reduce thicknesses of flat stock without direct supervision. This freed the worker to do other tasks while the shaper machined the workpiece.

By 1875, basic machine tools, such as the lathe, upright drill, and milling machine, **Figure 1-5**, were capable of attaining accuracies of one one-thousandth of an inch. This proficiency in machining and manufacturing would help America greatly during World War II. Factories were rapidly converted to produce military hardware instead of consumer goods. Of special importance to the war effort was the opening up of heavy industry professions to women. This supplied the labor needed to produce the large quantities of guns, ammunition, tanks, planes, and ships necessary to win the war, **Figure 1-6**.

1.1.2 Power Sources

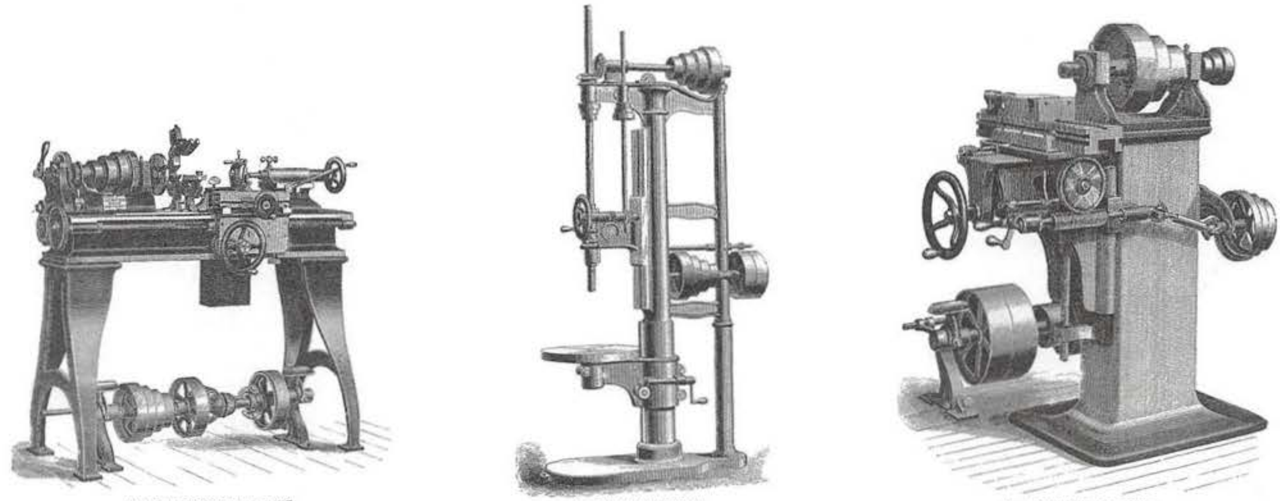
As machine tools were improved, so was the way they were powered. At first, the changes were very slow, occurring over hundreds of years. The greatest changes have come only in the last 150 years or so. The following are the various power sources used by machine tools throughout history in the order they evolved:

- **Hand power.** The bow lathe and bow drill are examples. The 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 strong enough for this work.
- **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 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.

GREEN MACHINING

Renewable and Nonrenewable Resources

Renewable resources are resources, such as water, solar energy, wind energy, and wood, that are replaced naturally and fairly quickly. Humans can use these resources repeatedly because nature provides a steady supply. Nonrenewable resources take much longer to produce. Their rate of consumption (how quickly we use them) outpaces their rate of production (how quickly they are made). When possible, industrial applications now favor environmentally friendly renewables over nonrenewable resources like coal, natural gas, petroleum, and rare earth minerals. Renewable energy sources, such as wind or solar power, offer long-term environmental and economic benefits to manufacturing over nonrenewable fossil fuels.



Goodheart-Willcox Publisher

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.

Library of Congress, Prints & Photograph Division, FSA-OWI Collection, LC-DIG-fsac-1a34951

Figure 1-6. Lathe operator machining parts for transport planes at the Consolidated Aircraft Corporation plant in Fort Worth, Texas.

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

Figure 1-7. A modern lathe featuring chuck safety guard, foot brake, coolant system, inch/metric dials, and a universal gearbox capable of cutting inch, metric, and diametral threads. Except those tools that perform nontraditional machining operations, all machine tools have evolved from the lathe.

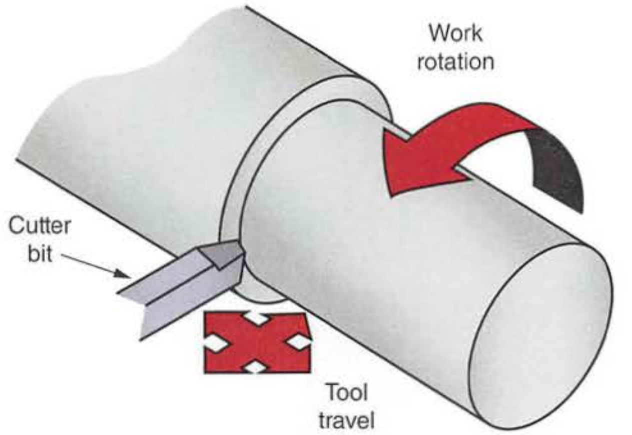
1.2 Basic Machine Tool Operation

Almost all machine tools have evolved from the ***lathe***, **Figure 1-7**. This machine tool performs one of the most important machining operations, ***turning***. It operates on the principle of rotating work against the edge of a cutting tool, as shown in **Figure 1-8**. 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.

1.2.1 Sawing Machines

A ***sawing machine***, **Figure 1-9**, or saw, makes use of a multi-toothed saw blade to cut away material. Sawing machines come in a variety of forms. All sawing machines perform one of two basic operations:

- **Cutoff sawing.** Sawing and cutoff machines cut stock material into more manageable lengths in preparation for other machining operations.
- **Band machining.** A vertical band saw uses a continuous saw blade. Chip removal is rapid and accuracy can be held to close tolerances, eliminating or minimizing many secondary machining operations.



Goodheart-Willcox Publisher

Figure 1-8. The lathe operates by rotating the work against the edge of a cutting tool.

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

Figure 1-9. Sawing machines, like this horizontal band saw, make use of a continuous saw blade, with each tooth functioning as a precision cutting tool.

1.2.2 Drill Press

A ***drill press*, Figure 1-10**, rotates a cutting tool (drill) against the material with sufficient pressure to cause the tool to penetrate the material. It is primarily used for cutting round holes. See **Figure 1-11**. Drill presses are available in many versions. Some are designed to machine holes as small as 0.0016″ (0.04 mm) in diameter.

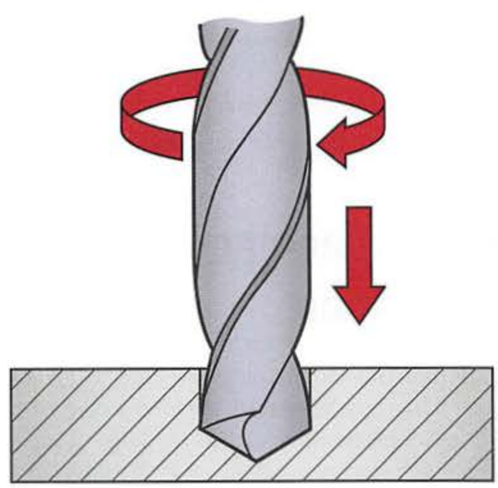
1.2.3 Grinding Machines

A ***grinding machine*, Figure 1-12**, or grinder, 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 handheld and manipulated until ground to the desired shape.

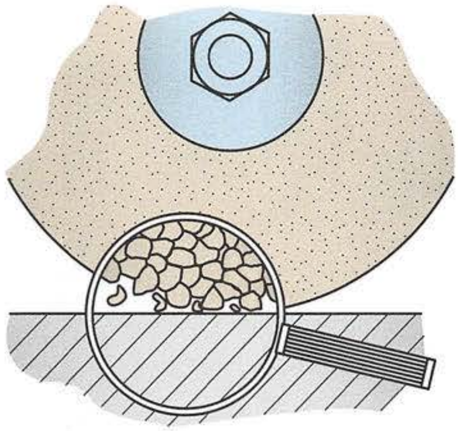
Willis Machinery and Tools Corp.

Figure 1-10. A typical 20″ variable-speed gearhead drill press with power feed. It can drill holes up to 1 1/2″ in diameter in cast iron.



Goodheart-Willcox Publisher

Figure 1-11. A drill press operates by rotating a cutting tool (drill) against the material with sufficient pressure to cause the tool to penetrate the material.



Goodheart-Willcox Publisher

Figure 1-12. 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 uses 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.

1.2.4 Milling Machine

A ***milling machine*** rotates a multitoothed cutter into the work, **Figure 1-13**. A variety of cutting operations can be performed on milling machines, including machining flat or contoured surfaces, slots, grooves, recesses, threads, gears, and spirals. Milling machines are available in more variations than any other family of machine tools, **Figure 1-14**, and are well suited to computer-controlled operation. The most advanced version of a milling machine is the CNC milling center.

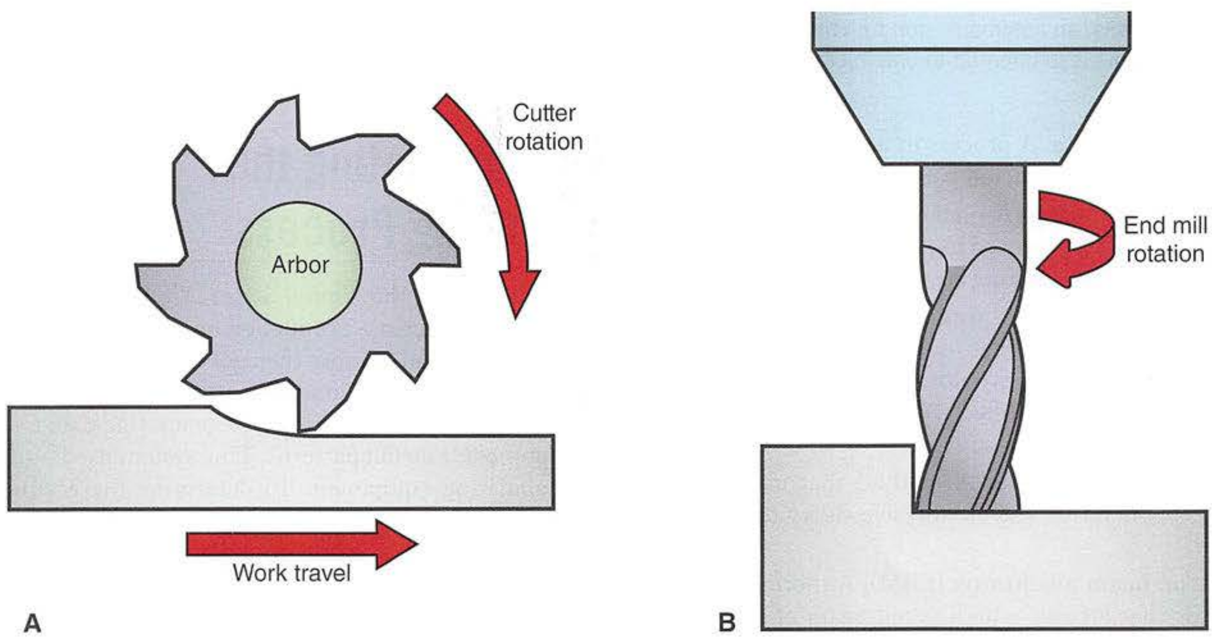
1.2.5 Broaching Machines

A ***broaching machine*** is designed to push or pull a multitoothed cutter across the work, **Figure 1-15**. Each tooth of the broach (cutting tool) removes only a small amount of the material being machined.

1.3 Nontraditional Machining Processes

A number of machining operations have not evolved from the lathe. They are classified as nontraditional machining processes. These processes include the following:

- **Electrical discharge machining (EDM).** An advanced machining process that uses a fine, accurately controlled electric 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.

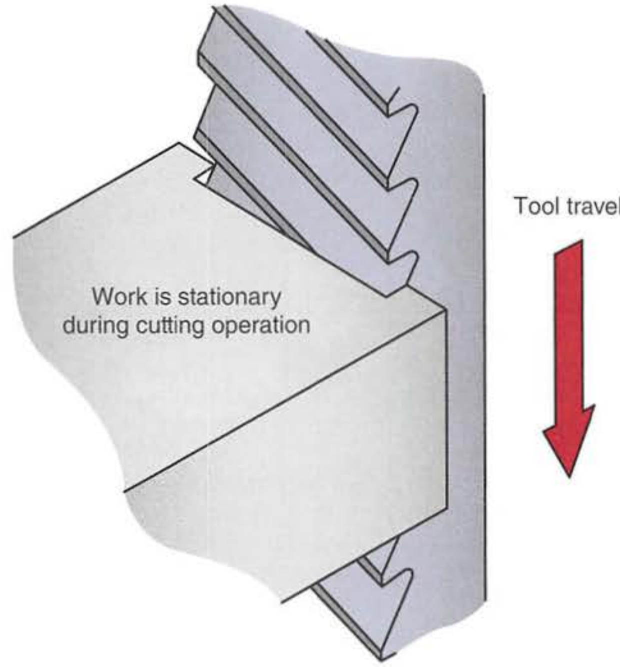


Goodheart-Willcox Publisher

Figure 1-13. Milling removes material by rotating a multitoothed cutter into the work. A—In 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 of the cutter.

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

Figure 1-14. A modern milling machine featuring power feed, variable speed controls, an automatic stop function, coordinate display, and selectable resolution up to one micrometer.



Goodheart-Willcox Publisher

Figure 1-15. 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.

- **Chemical milling.** A process in which chemicals are used to etch away selected portions of metal.
- **Chemical blanking.** A material removal method in which chemicals are used 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.

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 (MIT) to design a three-axis machining system. MIT eventually took over the entire 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 on 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 in 1955.

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 downloaded from an external computer. Programs on punched tapes are no longer used. A modern CNC machining center is shown in **Figure 1-17**.

WORKPLACE SKILLS

Joining Organizations

Belonging to a student or professional organization can help you reach your career goals. You will find student and professional organizations focused on almost every career topic. They will help you learn more about career options and meet other students and professionals who can help you establish a career. They will also help you learn teamwork skills that are needed in the workplace. Organizations that may help you prepare for a career in machining include the following:

- *SkillsUSA* is an organization for students preparing for technical, trade, and skilled-service occupations. Its goal is to create a strong American workforce. SkillsUSA programs include the Professional Development Program, which builds employability skills, such as communication and teamwork. The Work Force Ready System offers assessments for career and technical education, including areas such as technical drafting, residential wiring, and plumbing.
- *National Institute for Metalworking Skills (NIMS)* is an organization for students seeking training for a career in the machining and metalworking industries. NIMS accredits machining and metalworking programs and offers students credentials to prove their skills to future employers. NIMS offers credentials in manual milling, turning operations, and CNC machining among numerous other metalworking skills.
- *Society of Manufacturing Engineers (SME)* is a professional organization that supports manufacturing education. SME offers membership to students who have an interest in a STEM (science, technology, engineering, and mathematics) field or a career in the manufacturing industry. SME also recognizes and financially supports outstanding high school manufacturing programs.

Becoming involved in student and professional organizations now can help you land a job in your chosen field after completing your training. Your membership shows employers that you are already involved in the organization and serious about a career in a specific area.

Other professional organizations with student members include the National Society of Professional Engineers (NSPE), American Society for Quality (ASQ), American Welding Society (AWS), and ASM International. To be a student member of these organizations, you must be enrolled in a certain number of courses that will lead to a career in the given area. There is also a fee, which is usually less than a professional membership fee.

AMT—The Association for Manufacturing Technology

Figure 1-16. CNC machine tools are equipped with computers that permit computer-aided or manual programming. All controls needed for complete machine operation are in one location.

A CNC machine tool offers several benefits, including:

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

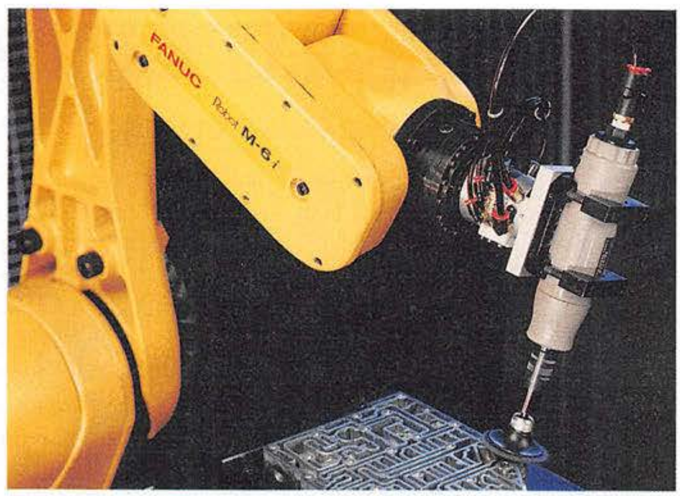
The use of robotic systems for loading and unloading permits some machine tools to operate unattended during the entire machining cycle. Robots, **Figure 1-18**, are also useful in many industrial applications:

- Operating in hazardous and harsh environments.
- Performing operations that would be tedious for a human operator.
- Handling heavy materials.
- Positioning parts with great repetitive precision.

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

Mack Molding Co., Arlington, Vermont

Figure 1-17. A modern CNC turning and machining center with multiaxis capabilities. Its advanced multitasking technology allows turning and milling operations to be performed with a single setup.



Fanuc Robotics North America, Inc.

Figure 1-18. A robot is a programmable, multifunctional manipulator designed to move material, tools, or other devices through programmed motions to perform a variety of tasks. This robot is deburring a complex part following machining operations.

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, 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.

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

Whether planning a CNC program or preparing to produce work on a conventional machine tool, a machinist must

Rainer Plendl/Shutterstock.com

Figure 1-19. The automotive industry makes extensive use of robots for positioning parts, welding, painting, and performing quality control tasks. Many production operations include computer-controlled robotic assembly lines like this one.

make many decisions on how to manufacture a part in the most economical way. A machinist must be able to perform the following activities:

- 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 these tasks require skill, knowledge, and experience. Essentially, a machinist is able to visualize the machining program.

1.6 Acquiring Machining Skills and Knowledge

The skills and knowledge needed by the machinist cannot be acquired in a short time. Machinists usually take part in a multiyear salaried apprentice program. In addition to machine tool training under an experienced machinist, the program includes related subjects such as English, algebra, geometry, trigonometry, print reading, safety, production techniques, and CNC principles and programming.

The National Institute for Metalworking Skills (NIMS), with the aid of the metalworking industry, developed a set of ***skill standards***, industry requirements for skilled workers. NIMS uses these standards to certify individuals through performance testing and accredited training programs that meet their standards. The standards provide skilled workers with certification that affords them industry recognition.

Chapter Review

Summary

- Machine tools and machinists are vital to our modern industrialized world.
- Modern machine technologies require highly educated machinists.
- Evolving power sources have changed the way machine tools operate and what materials they can machine.
- Basic machine tool processes include turning, sawing, drilling, grinding, milling, and broaching.
- There are a growing number of nontraditional machining processes.
- Computer numerical control (CNC) machine tools offer better accuracy, repeatability, and flexibility than manually controlled machine tools.
- Automation and computers have dramatically affected machine tool operations.
- Through skill, knowledge, and experience, a machinist determines the best possible way to manufacture a part, whether using manual or CNC machining techniques.
- Machinists follow skill standards established by NIMS.

Review Questions

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

1. Jobs such as tool and die making 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.
2. 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. All of the above.
   E. None of the above.
3. The Industrial Revolution could not have taken place without the cheap, convenient power of the ______.
4. 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.
5. What occurred in the mid-1860s that was very important to the development of machining technology in the United States?
6. List seven power sources in the order they have evolved.
7. Almost all machine tools have evolved from the ______.
8. List three types of basic machine tools and briefly describe their operation.
9. List four types of nontraditional machining processes and briefly describe their operation.
10. The introduction of the microchip in the mid-1970s led to the introduction of ______ machine tools.
11. CNC machine tools operate according to a(n) ______ made up of sentence-like commands.
12. List four industrial applications of robots.
13. When planning the manufacture of a part, a machinist must ______.
    A. ascertain tolerance requirements
    B. select the machine tool, cutter(s), and other tools and equipment that will be needed
    C. select a proper cutting fluid for the material being machined
    D. All of the above.
    E. None of the above.
14. Machinists can become certified through performance testing and accredited training programs that meet ______ developed by the National Institute for Metalworking Skills.

CHAPTER 2

Careers in Machining Technology

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Describe the types of jobs included in the four basic machining job categories and the qualifications needed for each.
- Explain where to obtain information on occupations in machining technology.
- State what industry expects of an employee.
- Describe what an employee should expect from industry.
- Summarize the information given on a résumé.
- List ways to keep your skills current throughout your career.

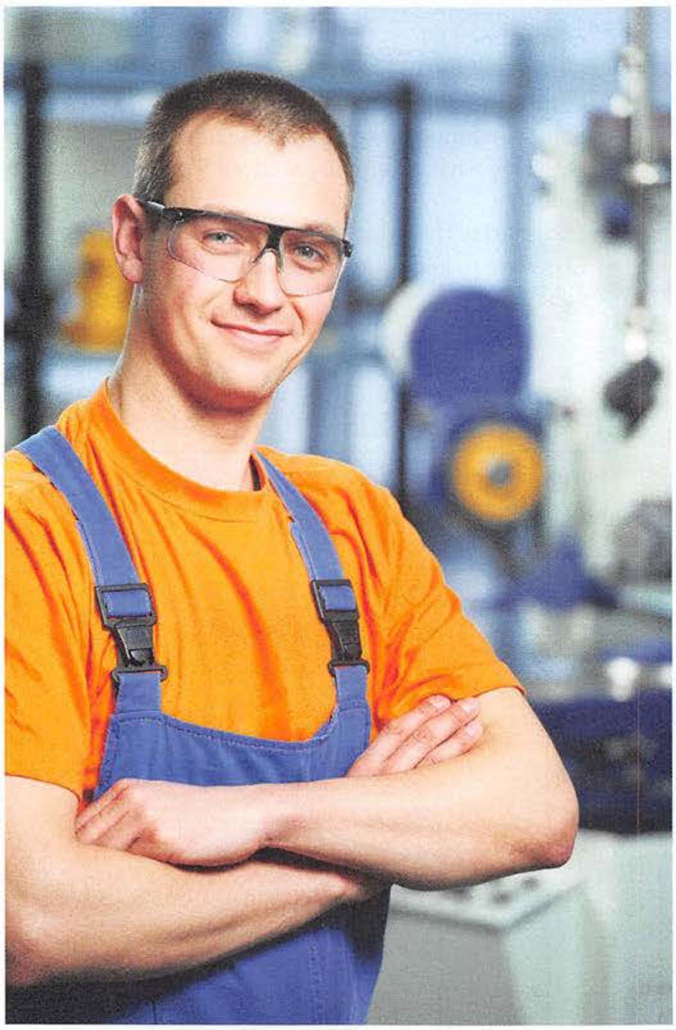
Technical Terms

apprentice	part programmer
associate's degree	résumé
bachelor's degree	semiskilled worker
career	skilled worker
engineer	technician
job shop	toolmaker

A *career* is a job that requires specialized training and commitment to the profession. Many people want a career that is both challenging and interesting. Their philosophy is that if you enjoy your job, you will never "work" a day in your life. Others are satisfied with whatever job comes along. With which viewpoint do you agree?

If you are looking for a career that is challenging, interesting, and rewarding, the field of machining offers many opportunities. See **Figure 2-1**. Whether you choose one of the machine shop areas or select a career in a related field, you will find that the study of *Machining Fundamentals* is basic to all of them.

No matter which occupational choice you make, you should realize that you will have to keep up with technical progress. To be successful and advance in your career, a continuing program of education is usually necessary.



Dmitry Kalinovsky/Shutterstock.com

Figure 2-1. The field of metal machining offers many opportunities for semiskilled and skilled workers, technicians, and professional personnel.

2.1 Machining Job Categories

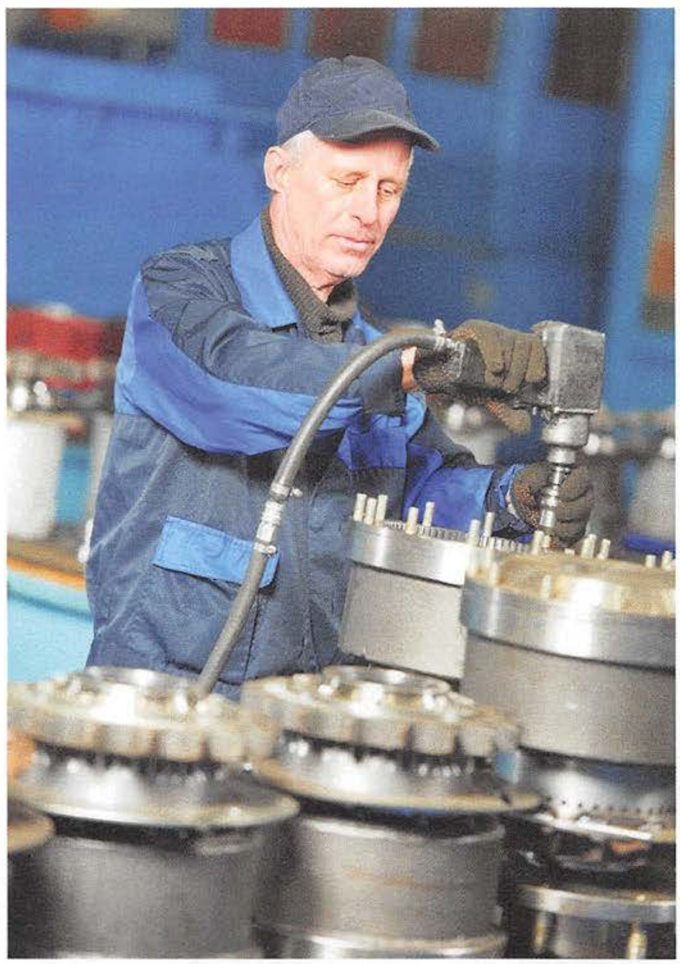
Jobs in material machining fall into four general categories:

- Semiskilled workers.
- Skilled workers.
- Technicians.
- Professionals.

2.1.1 Semiskilled Workers

A ***semiskilled worker*** performs basic, routine operations that do not require a high degree of skill or training. Semiskilled workers may be classified into the following general groups:

- Those who are helpers for skilled workers.
- Those who operate machines and equipment used in making things. The machines are set up by skilled workers.
- Those who assemble the various manufactured parts into final products, **Figure 2-2**.



Dmitry Kalinovsky/Shutterstock.com

Figure 2-2. Many semiskilled workers are employed in assembly industries, where they assemble manufactured parts into complete units. Training periods to learn these job skills are relatively short.

Most semiskilled work is found in production shops that have many repetitive operations. In general, semiskilled workers are told what to do and how the work is to be done. There is little chance for advancement from semiskilled jobs without additional education and training. Semiskilled workers are often the first to lose their jobs when there is a downturn in the economy and may have difficulty finding another job due to their lack of education. It is therefore a good idea for semiskilled workers to consider furthering their education and training.

2.1.2 Skilled Workers

A ***skilled worker*** has been trained to do more complex tasks. Skilled workers are found in all areas of material machining. Many skilled workers obtain their training as an ***apprentice*** in an apprenticeship program (on-the-job training while working with skilled machinists), **Figure 2-3**. Four or more years of instruction under an experienced machinist is generally required. In addition to working in the shop, an apprentice usually studies related subjects, such as math, science, English, print reading, metallurgy, safety, and production techniques. On completion of an apprenticeship program, the worker is capable of performing the precise work essential to the trade.

In recent years, the demand for skilled workers has grown tremendously. Workers entering the field can receive their training through the armed forces, **Figure 2-4**, or in career and technical education programs offered in high schools and community colleges. Many community college programs are offered in conjunction with local industry.

US Army

Figure 2-4. The Army and other branches of the armed forces offer excellent opportunities for learning a trade. These servicewomen are learning to machine parts on a manual lathe. The coursework they complete can be used toward obtaining National Institute of Metalworking Skills (NIMS) credentials.
(Use of military imagery does not imply or constitute endorsement of Goodheart-Willcox Publisher, its products, or services by the US Department of Defense.)

There are several areas in which a machinist may concentrate. Some of these areas of specialization are discussed in the following sections.

Monkey Business Images/Shutterstock.com

Figure 2-3. The apprentice studies under an experienced machinist for a period of four or more years. The training program also includes the study of related subjects, such as math, English, and science.

All-Around Machinist

An all-around machinist is skilled in the setup and operation of most types of machine tools. He or she must be familiar with both manual and computer-controlled machine tools and how they are programmed, **Figure 2-5**. An all-around machinist is expected to plan and carry out all of the operations needed to machine a job.

Many all-around machinists work in ***job shops***. Some job shops specialize in creating custom or experimental machining projects for various clients. Other job shops specialize in manufacturing products with very small production runs.

Tool and Die Maker

A ***toolmaker*** is a highly skilled machinist who specializes in producing the tools and tooling needed for machining operations. These include the following:

- **Dies.** Special tools for shaping, forming, stamping, or cutting metal or other materials.
- **Jigs.** Devices that position work and guide cutting tools.
- **Fixtures.** Devices that hold work while it is machined.
- **Gauges.** Devices that hold work while it is being measured.

These tools are necessary for modern mass production techniques. Toolmakers must have a broader background in machining operations and mathematics than most other skilled workers in the trade. See **Figure 2-6**.

A die maker specializes in making the punches and dies needed to stamp out parts, such as auto body panels, electrical components, and similar products. He or she also produces the dies for making extrusions (metal shaped by being pushed through an opening in a metal disc of proper configuration).

John-james Gerber/Shutterstock.com

Figure 2-5. The all-around machinist can set up and operate most types of machine tools, both manual or CNC.

Precision Castparts Corp.

Figure 2-6. A tool and die maker checking the dies used for molding a plastic pattern used to cast a jet engine component. Master tooling ensures that other sections of the engine, made elsewhere in the United States, Israel, and Europe, will fit together perfectly.

Tools used in die castings (parts made by forcing molten metal into a mold) are often called *dies*, which is incorrect. Tools that have molten material forced into them are called *molds*, and the correct term for the professional that builds molds is *mold maker*. Like the toolmaker, die makers and mold makers are highly skilled machinists.

Layout and Setup Specialists

The layout specialist is a machinist who interprets the drawings and uses precision measuring tools to mark off where metal must be removed by machining from castings, forgings, and metal stock. This person must be very familiar with the operation and capabilities of machine tools. He or she is well-trained in mathematics and print reading.

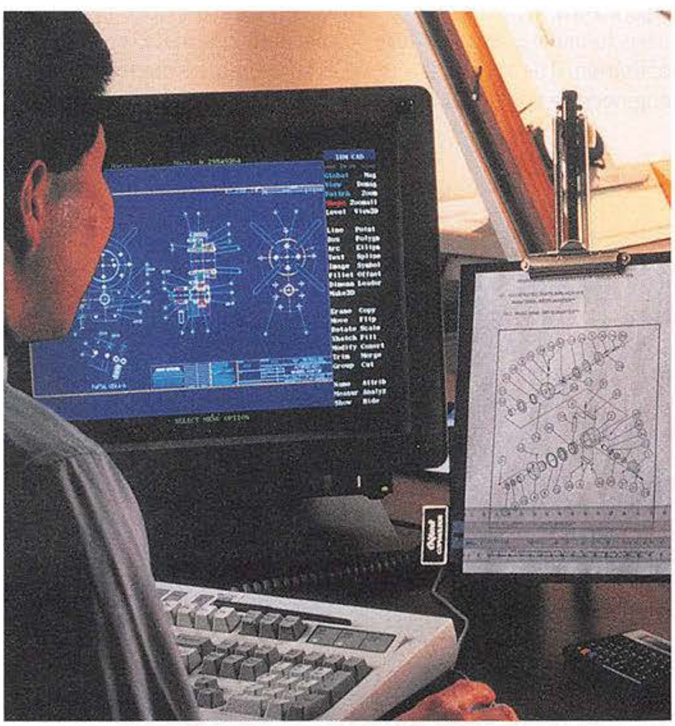
A setup specialist is a person who locates and positions ("sets up") tooling and work-holding devices on a machine tool for use by a machine tool operator. With CNC equipment, the setup specialist may also be required to load part programs into the machine controller and adjust various machine offsets. This worker may also show the machine tool operator how to do the job, and often checks the accuracy of the machined part. See **Figure 2-7**.

Part Programmer

A ***part programmer*** inputs data into a computer-controlled (CNC) machine tool for machining a product. CNC machine tools have revolutionized the fields of machining and manufacturing. However, computers must be programmed by a highly skilled part programmer who studies the drawings and determines the sequences, tools, and motions the machine tool must carry out to machine the part, **Figure 2-8**.

Hydromat, Inc.

Figure 2-7. A setup specialist is a master machinist who prepares machine tools for operation by less highly trained personnel. After thoroughly checking the machined part to be sure it meets specifications, the setup specialist turns the machine tool over to a machine operator.



Tri-Tool, Inc.

Figure 2-8. The programmer prepares the machining information, such as tools, tool paths, and machining sequence, for a CNC program that will direct the entire machining process of a specific part. The programmer must thoroughly understand machining technology.

To perform this task, a part programmer must have the following qualifications:

- Formal training in computer hardware as it relates to machine tool operation.
- Formal training in computer-aided design (CAD) and computer-aided manufacturing (CAM) software.
- Experience in reading and interpreting drawings.
- A thorough grounding in machining technology and procedures.
- A working knowledge of cutting speeds and feeds for various tools and materials.
- Extensive training in mathematics.

In high-volume production settings, the part programmer is also responsible for program adjustments and modifications that require knowledge above that of an operator or setup specialist. Many community colleges and career and technical education centers offer programs in CNC programming, CAD, and CAM.

Supervisor or Manager

A supervisor or manager is usually a skilled machinist who has been promoted to a position of greater responsibility. This person directs other workers in the shop and is responsible for meeting production deadlines and keeping work quality high, **Figure 2-9**. In many shops, the manager may also be responsible for training and other tasks.

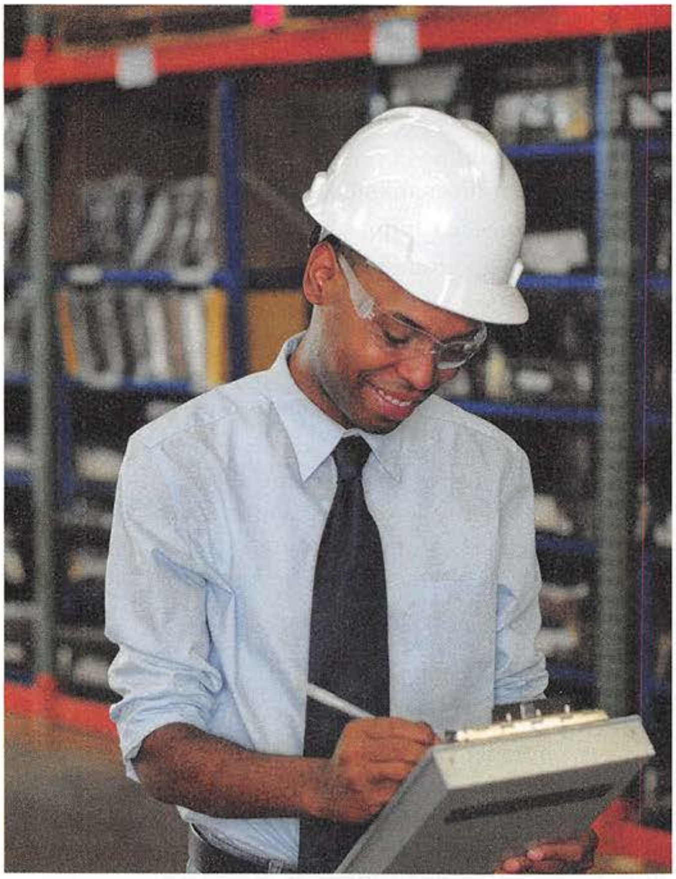
GREEN MACHINING

Green Manufacturing

Green manufacturing describes a two-fold trend in manufacturing: manufacturing green products and making the process of manufacturing green. Green products are environmentally friendly in their use. Green products are durable, free of toxic compounds, and often biodegradable. As part of green manufacturing, green products are produced with less waste and less pollution than traditional manufacturing practices and are often made from renewable or recycled materials. International attention on environmentally friendly manufacturing using renewable resources has driven a drastic increase in green manufacturing worldwide. "Greening" the manufacturing process by reducing waste and pollution improves efficiency and can also lead to innovation and job growth.

2.1.3 Technicians

A ***technician*** is a member of the production team who operates in the realm between the shop and engineering departments. The position is an outgrowth of today's highly technological and scientific world. The job usually requires at least two years of college, with a program of study centered



Steve Good/Shutterstock.com

Figure 2-9. The supervisor or manager of the production department works very closely with machinists, engineers, metallurgists, and other staff. Supervisors may also be responsible for planning projects, ordering stock, meeting production deadlines, and ensuring high-quality work.

Chuck Rausin/Shutterstock.com

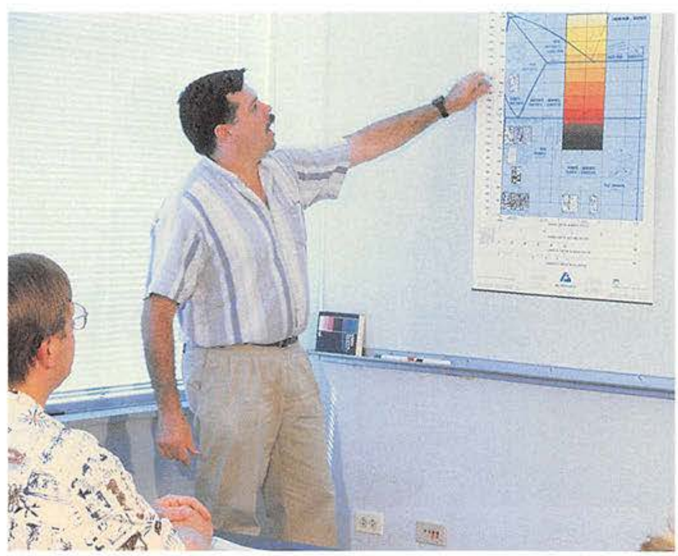
Figure 2-10. This quality control technician is inspecting a die used to make plastic patterns to be used in a metal casting operation. The technician must ensure that the die meets engineering specifications before being shipped to customers.

on math, science, English, computer science, quality control, manufacturing, and production processes. Many state and community colleges offer two-year ***associate's degree*** programs devoted to preparing students for such technical positions.

The technician assists the engineer by testing various experimental devices and machines, compiling statistics, making cost estimates, and preparing technical reports. Many inspection and quality control programs are managed by technicians, **Figure 2-10**. Technicians also repair and maintain computer controlled machine tools and robots.

2.1.4 Professionals

Several professions offer many excellent opportunities in the fields of machining, metalworking, and manufacturing. Teaching is a challenging profession that offers a freedom not found in most other professions. Teaching can be a very satisfying profession on a personal level, although it is a field that students often overlook, **Figure 2-11**. Teachers of industrial arts, industrial technology, vocational education, and career and technical education are in a fortunate



Goodheart-Willcox Publisher

Figure 2-11. The teaching profession is a challenging one. Many skilled educators will be needed in machining technology if the United States is to maintain its position as a world leader.

position. It is not an overcrowded profession, and there will be a demand for teachers for many years to come. To teach machining, four years of college training are usually needed, **Figure 2-12**. While industrial experience is ordinarily not required, it will prove very helpful.

Engineering is another fast-growing and challenging profession. An ***engineer*** uses mathematics, science, and knowledge of manufacturing principles to develop new products and processes for industry, **Figure 2-13**. A four-year ***bachelor's degree*** is usually the minimum requirement for entering the engineering profession. Some men and women have been able to enter the profession without a degree after a number of years of experience as machinists, drafters, or engineering technicians. However, they are usually required to take additional college-level training.

Industrial engineers are primarily concerned with the safest and most efficient use of machines, materials, and personnel, **Figure 2-14**. In some instances, an industrial engineer may be responsible for the design of special machinery and equipment to be utilized in manufacturing operations.

Mechanical engineers are normally responsible for the design and development of new machines, devices, and ideas. This engineering specialty is also involved with the redesign of or improvements to existing equipment, **Figure 2-15**. Some mechanical engineers specialize in different types of mechanical systems, such as heating and cooling systems, automotive vehicles, and robotics.

Tool and manufacturing engineers often work with the other engineers. A principal concern of the mechanical engineer is the design and development of the original

William Schotta, Millersville University

Figure 2-12. This college student may someday teach machine technology. During four or more years of training, she will learn all phases of machine tool operation and programming.

WORKPLACE SKILLS

Investigating Education and Training

You will need to consider what educational level is necessary for entering each career you investigate. How much training or experience is needed? Can people enter the field with less training and acquire expertise while working on the job? Are special certificates, licenses, or credentials needed? You can become a machinist through an apprenticeship program, earn a degree from a community college, or earn credentials through the National Institute of Metalworking Skills (NIMS).

Before deciding on a specific career, you may wish to shadow someone who holds the type of job you desire. Job shadowing is the process of observing a person in the workplace to learn more about his or her job and its requirements. In addition, you may seek to have a mentor's assistance. A mentor is someone with greater experience and knowledge who guides you in your career.

You might also consider assisting someone who knows how to do the job tasks well, such as working in an internship or apprenticeship. An internship is an arrangement with an educational institution whereby a student is supervised while working with a more experienced jobholder. An apprenticeship involves learning a trade under the direction and guidance of an expert worker. You also gain valuable experience through working at a part-time job or volunteering at community or charitable organizations.

Perhaps your plan of study leads you to acquire a license or become certified. Other programs may require a college degree. An associate's degree is a two-year college degree. A bachelor's degree is a college degree usually requiring four years of study. A master's degree requires another year or two of study beyond a bachelor's degree. A master's degree is also called a graduate degree.

Corepics VOF/Shutterstock.com

Figure 2-13. An engineer inspects the rotors of a huge industrial wind turbine that will be used to test the aerodynamic qualities of everything from aircraft to automobiles.

Minerva Studio/Shutterstock.com

Figure 2-14. Industrial engineers have many duties. Industrial engineers may be responsible for maintaining a safe work environment, managing workflow, and/or making the most efficient use of the machines and materials.

Chuck Rausin/Shutterstock.com

Figure 2-15. Mechanical engineers are responsible for the design and development of new machines, devices, and ideas.

or prototype model. When this model has been thoroughly tested and has met design requirements, the product is turned over to the tool and manufacturing engineer to devise methods and means required to manufacture and assemble the item, **Figure 2-16**.

Metallurgical engineers are involved in the development and testing of metals that are used in products and manufacturing processes. These engineers work closely with manufacturing and mechanical engineers to ensure that the proper materials are specified for new designs.

2.2 Preparing to Find a Job in Machining Technology

Machining technology is a technical area with constantly developing new ideas, materials, processes, and manufacturing techniques. This continual development creates new occupational opportunities. A recent study reported that the average graduate will be employed in at least ten different jobs in his or her lifetime, and many of these jobs do not even exist yet!

OSG Tap & Die, Inc.

Figure 2-16. Tool and manufacturing engineers devise new methods to manufacture complex products. This engineer is setting up a CNC machine for a new cycle.

2.2.1 Obtaining Information on Machining Occupations

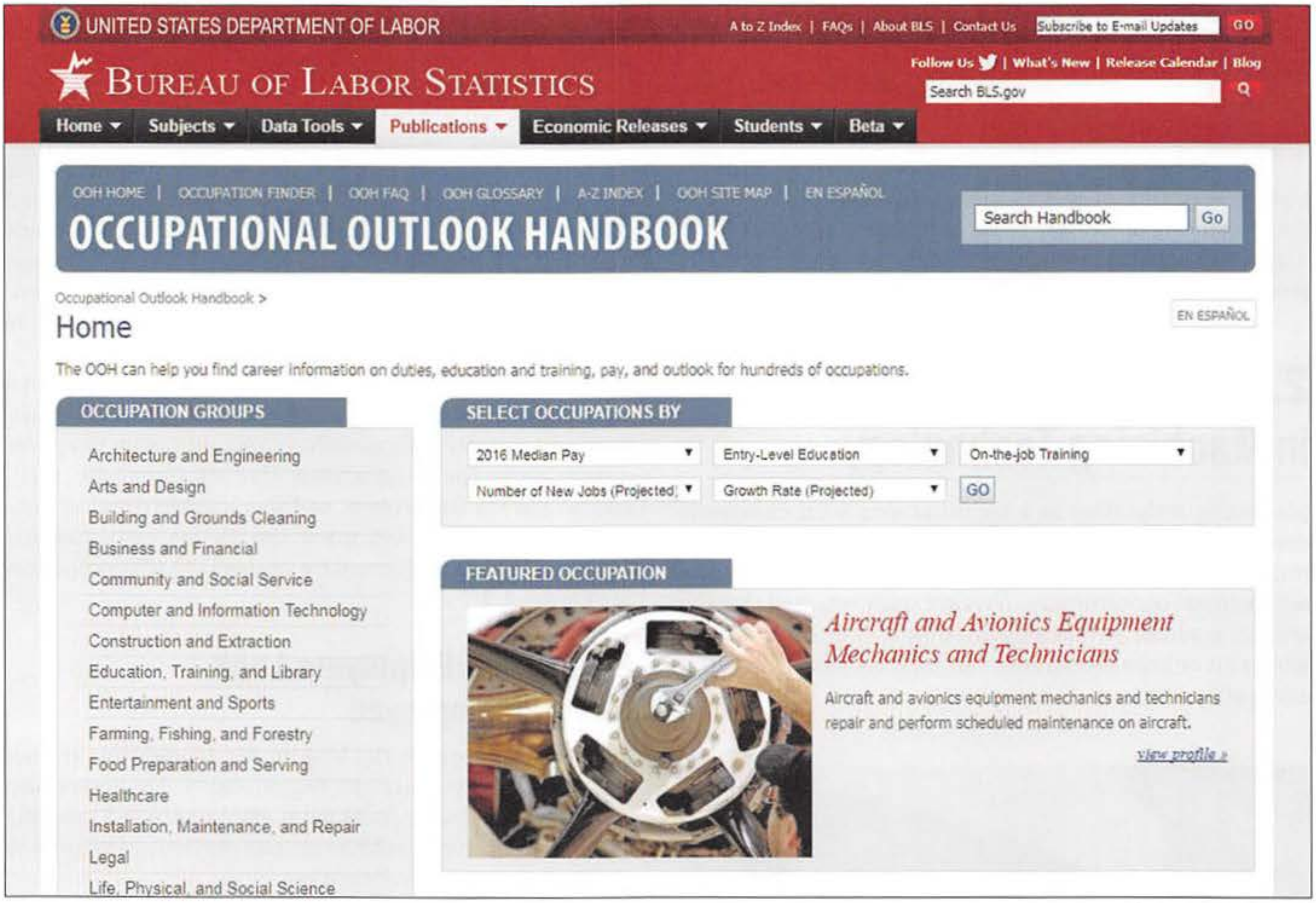
There are many sources of occupational information. The most accessible sources include the school's career center, technical education instructors, and the Internet. State employment services are also excellent sources for local and state employment opportunities, as are the various trade unions concerned with the metalworking trades. Information on technical occupations is also available from community colleges. Many of them offer associate's degrees in technical areas.

You may also wish to contact the field and regional offices of the Office of Apprenticeship of the US Department of Labor for information on apprenticeship programs in your area. The *Occupational Outlook Handbook*, available on the US Department of Labor's website and in published formats, describes many specific job categories and estimates the future demand for workers in each occupation. See **Figure 2-17**.

2.2.2 Traits Employers Look for in an Employee

Industry is always on the lookout for people who are not afraid to work and assume responsibility. Employers also look for the following traits in an employee, often referring to scholastic records, references, and previous employers to obtain the necessary information:

- **Skills and knowledge.** Has the technical skills and knowledge necessary for an entry position. Does work neatly and accurately. Pays attention to details.
- **Integrity and honesty.** Adheres to company policies, including those related to drugs and alcohol in the workplace. These traits are just as important as technical skills and knowledge.
- **Comprehension.** Is able to understand oral and written instructions and to read and interpret prints.
- **Dependability.** Has a good attendance and punctuality record in class and at former jobs.
- **Teamwork.** Shows the ability to work well with peers and supervisors.
- **Communication.** Has the reading, writing, and speaking skills to effectively communicate ideas.
- **Self-confidence.** Takes pride in work and does not knowingly turn out inferior or substandard material.
- **Accountability.** Is able to assume responsibility and be accountable for his or her actions.
- **Initiative.** Volunteers ideas and demonstrates leadership.
- **Grooming and dress.** Presents a positive personal appearance.



Goodheart-Willcox Publisher

Figure 2-17. One excellent source of information about occupations in the metalworking industry is the *Occupational Outlook Handbook*, provided online by the US Department of Labor.

2.2.3 Factors for Rejection for Employment

There are many factors and traits that can cause a person to be rejected for employment. They include the following:

- Poor personal appearance.
- Poor scholastic performance.
- Poor attendance record.
- Lack of maturity.
- Lack of interest or enthusiasm for the job being sought.
- Knowing little or nothing about the company where employment is sought.
- Unrealistic salary demands.
- Lack of ability to communicate.

Many employers perform background checks on job applicants. Prospective employees will likely have to pass a drug screening before an offer of employment is finalized. Once hired, the employee may be subjected to random drug screenings as defined in a company policy. Most drug policies also include mandatory drug screens following industrial accidents to determine if the influence of drugs or alcohol could have been a factor in the accident.

2.3 How to Get and Keep a Job

To be successful in getting a job, you may need to spend a significant amount of time and energy. However, there are several things that you can do to make this task easier. First, decide what type of work you would like to do. Most schools and state employment services administer tests that may help you determine the areas of employment in which you will have a good chance of succeeding.

Answering the following questions will give you additional help:

- What can I do with some degree of success?
- What have I done that others have commended me for doing well?
- What are the things I really like to do?
- What are the things I do *not* like to do?

- What jobs have I held? Why did I leave them?
- What skills have I acquired while in school?

You will probably have two or more areas of interest. After listing them, start gathering information on these areas of interest. Use as many different sources as time permits. This may include searching the Internet, reading publications about the profession, talking with people doing this type of work, and visiting industry. Plan your educational program to prepare for entry into a specific job or for advanced education or training.

The next task is to figure out how to go about getting the job you want. Jobs are always available. Workers get promoted or they retire. Some quit, die, or get fired. Technological progress also creates new jobs. However, you must track down these jobs. There is no easy way to get a challenging job.

Last, but not least, know where to look for the job you want. There are many websites dedicated to listing job opportunities. Schools often host job fairs where potential job seekers can meet with employers and companies. Check the classified advertising section of local newspapers. Talk with friends and relatives who are employed. They may be aware of job openings at their places of employment before the jobs are advertised.

A new office or factory building may indicate potential job openings. It would also be to your advantage to prepare a list of desirable employers in your community and visit their personnel offices. Plan these visits on a routine basis when jobs are not readily available. The office staff will then know you are interested in working for their firm and may give you preference.

Concentrate on getting the job. If possible, always make your initial request for a job in person. Always be specific about the type of job you are seeking. Make sure you are qualified for that job. Never ask for "just any job" or inquire, "What openings do you have?"

Dress appropriately. Job hunting is not the time to wear old clothes or torn and beat-up shoes. Be clean and well-groomed.

When filling out a job application, avoid leaving any spaces blank. The employer may think there is something you do not want to answer. If the question is not applicable to you, write in "Not applicable," "Does not apply," or "NA." For example, there might be a question on the application that asks, "What was your highest rank in the armed forces?" If you were not in the armed forces, you would write in, "Does not apply," or "NA," rather than leaving the answer space blank.

One thing you must remember: The job will *not* come to you. You must search for it!

2.3.1 Preparing a Résumé

To speed the tedious task of filling out job applications, prepare a ***résumé*** in advance. A résumé is a summary of your educational and employment background, **Figure 2-18**. It helps ensure that you are providing correct, uniform information with little chance for confusing responses. Submit your résumé when you apply for a job. Your résumé should include the following items:

- Your full name.
- Your telephone number, e-mail address, and full address. Do not forget area and zip codes. (A helpful hint: Make sure your e-mail address is professional. E-mails with funny or suggestive wording may have been humorous at one time, but a prospective employer will not be amused. The same goes for greetings on your telephone voicemail. Do not disqualify yourself from an interview by displaying unprofessional behavior.)
- Education and any special training. Include dates of all educational attendance.
- The types of equipment that you can safely operate.
- Names of previous employers. List the places you have worked, starting with the most recent. Include the items that follow (for each previous employer):
  - Company name and address.
  - Dates employed.
  - Immediate supervisor's name.
  - Salary or pay rate.
  - Reason for leaving.
- Names and addresses of references. Do not include relatives unless you have worked for them. Make sure you secure permission before using a person for a reference. Today, many job seekers specify "References available on request," instead of listing the names and addresses on the résumé.

2.3.2 Social Media

With the rise in the use of the Internet and social media websites, many people are documenting their lives in a public forum for everyone to see. This includes prospective employers, who regularly look for social media sites of prospective employees.

Social media content can reflect both positively and negatively on any job applicant. First impressions are important, so do not disqualify yourself from consideration for a job by posting things on the Internet that will make an employer think you lack some of the important traits that employers desire in employees. Never post anything that could be considered offensive or in poor taste. This includes making unkind comments about coworkers or former employers.

Keep your professional and personal social media use separate. Some social media sites, such as LinkedIn, can actually help you with your job search. Be sure to provide accurate information, and avoid exaggeration.

2.3.3 What an Employee Should Expect from Industry

From the preceding sections, you have some idea of what industry expects from an employee. However, are you aware of what an employee should expect from industry? In

Michael J. Garcia

117 Sierra Avenue
Meskwaki, WI 53583

(608) 633-1234
mjgarcia22@e-mail.com

Career Objective

To obtain an entry-level machinist position in the manufacturing industry.

Professional Experience

Minton Manufacturing, Holton, WI — August 2016–present
Machine Shop Helper

- Work with basic hand tools and machinery, including buffing and grinding parts.
- Inspect products for quality using measuring tools and machines.
- Perform general shop labor, including material loading and cleanup.

Putnam Supply Co., Meskwaki, WI — May 2015–August 2016
Parts Clerk

- Worked with customers at parts counter, checked inventory system, and obtained parts.
- Conducted daily and monthly inventory checks.
- General stocking and cleaning throughout store.
- Delivered and picked up parts and equipment.

Education

Associate Degree in Precision Machine Technology — May 2016
Wisaka Community College

- GPA: 3.43/4.0
- Coursework included precision machining, die making, CNC/CAM training, CAD drafting and solid modeling, print reading for industry, GD&T, physical metallurgy, and metrology.
- Obtained seven NIMS Machining Level I credentials: Measurement, Materials & Safety; Job Planning, Benchwork & Layout; Drill Press Skills I; Grinding Skills I; Manual Milling Skills I; Turning Operations: Turning Between Centers; and Turning Operations: Turning Chucking Skills.
- Participated in SkillsUSA chapter.

Community Service

Habitat for Humanity, volunteer, summers of 2014, 2015, 2016
Meskwaki Food Bank, volunteer, 2013–present

References

Available upon request.

Goodheart-Willcox Publisher

Figure 2-18. Your résumé is an important tool that summarizes your education and training, skills, and work experience for potential employers. It should be complete, well-organized, and easy to read.

addition to salary and benefits, what should you expect from an employer?

The following are a few questions you can ask when selecting a place of employment:

- Is a relatively safe and clean work area provided? Obviously, some areas can never be made as safe as others. For example, tapping a blast furnace is inherently more dangerous than working on a small lathe or drill press.
- Are work areas adequately lighted, heated, and ventilated? Are noxious fumes and dust particles filtered from the air?
- Is proper safety clothing and equipment available for all dangerous work? Safety items, such as goggles, hearing protectors, and steel-tipped safety shoes, may be provided free or at minimum cost.
- Are all necessary precautions observed when hazardous materials are involved?
- Is there a preventive safety program, and are safety regulations and precautions rigorously enforced?

2.3.4 Factors That Can Lead to Job Termination

The following factors can lead to failure to get a promotion, or possibly being terminated (fired) from a job:

- Alcohol or illegal drug use on the job.
- Inability or refusal to perform the work required.
- Being habitually tardy or missing work repeatedly without adequate reasons.
- Inability to work with supervisors or peers.
- Fighting with or making threats to fellow workers or supervisors.
- Inability to work as a team member.
- Theft of company property, or using company resources for personal projects without permission.
- Inappropriate use of social media.

2.4 Keeping Your Skills Current

The completion of your formal schooling does not mean the end of your training and study. To keep a job and advance in it, you will have to keep up-to-date with the knowledge and new skills that advanced technologies demand, **Figure 2-19**. Keen competition from foreign-made products and the ever-changing nature of technology make this a very real necessity.

Another important consideration in maintaining a competitive edge in the marketplace is to pursue professional certifications and credentials. The National Institute for Metalworking Skills (NIMS) provides several credentialing opportunities for people working in the metalworking field, including the machine tool profession. Listed below are the credentials offered for machining professionals, along with the standards associated with each credential.

Machining Level I

Measurement, Materials & Safety
Job Planning, Benchwork & Layout
Manual Milling Skills I
Turning Operations: Turning Between Centers
Turning Operations: Turning Chucking Skills
Grinding Skills I
Drill Press Skills I
CNC Turning: Programming Setup & Operations
CNC Milling: Programming Setup & Operations
CNC Turning: Operations
CNC Milling: Operations

Machining Level II

Manual Milling Skills II
Turning II (manual)
Drill Press Skills II
Grinding Skills II
CNC Milling Skills II
CNC Turning Skills II
EDM — Wire
EDM — Plunge

Machining Level III

CNC Turning Skills III
CNC Milling Skills III

NIMS offers additional credentials in specialty areas such as screw machining, die-making, and machine maintenance, service, and repair. These credentials are intended to be follow-up certifications to the machining credentials.

Chuck Rausin/Shutterstock.com

Figure 2-19. Rapidly advancing technologies often require workers to learn new skills in order to perform their jobs well. This worker attended workshops provided by the manufacturer to learn how to use this electrical discharge wire cutting machine (EDWC).

Chapter Review

Summary

- Machining careers can be divided into four main categories—semiskilled, skilled, technical, and professional—based on training, level of education, and job requirements.
- Information about machining-related careers can be found through a school's career center, from instructors, the Internet, community colleges or career and technical education centers, state employment services, the armed forces, or the Department of Labor.
- Employers are looking for very specific traits in their employees, including work-related skills, knowledge, and ability, but also honesty, dependability, teamwork and communication, initiative, and proper grooming and dress.
- Employees have the right to expect certain things from their employer, including a safe, clean workplace that is adequately lighted, heated, and ventilated; availability of proper safety clothing and equipment; and enforcement of all safety regulations and precautions.
- Your résumé should include all relevant information about your education and training, skills, and work experience, as well as any credentials you have earned.
- What is on your résumé may get you hired, but in order to keep and advance in your chosen career it is very important to keep up-to-date with new skills and advances in technology.

Review Questions

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

1. List the four general categories of metalworking occupations.
2. ______ workers are those who perform operations that do *not* require a high degree of skill or training.
3. The ______ worker usually starts his or her career as an apprentice.
4. Other than apprenticeship programs, where can specialized career training be obtained?
5. Describe what an all-around machinist is expected to do.
6. What does a layout specialist do?
7. List five areas of knowledge a part programmer should have to perform his or her job properly.
8. List the areas of study usually included in a technician's educational program.
9. What are some of the duties of a technician?
10. List three sources of information on metalworking occupations.
11. List five traits an employer wants in a prospective employee.
12. List five factors that can lead to rejection for employment.
13. What is a job résumé?
14. Why should you prepare your résumé before filling out a job application?
15. List three questions an employee should ask when selecting a place of employment.
16. What are five factors that can lead to job termination?
17. What is necessary if you want to keep your job and advance in your chosen career?

CHAPTER 3
Shop Safety

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Explain why it is important to develop safe work habits.
- Identify common shop safety procedures.
- Dress in the proper safety equipment and clothing for a machine shop.
- List general guidelines for machine and tool safety.
- Select the appropriate fire extinguisher for a particular type of fire.
- Explain the purpose and use of safety data sheets.

Technical Terms

approved respirator
combustible material
machine shield
Occupational Safety and Health Administration (OSHA)
protective clothing
safety data sheet (SDS)
safety equipment
spontaneous combustion
ventilation

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 3-1**.

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

SAFETY NOTE

Throughout the text, Safety Notes are included to emphasize important tips and warn against potential hazards.

3.1 Safety in the Shop

There are several general precautions you can take to work safely in the shop. Remember that the shop is a place to work. It is not a place for horseplay. A "joker" in a machine shop is a walking hazard to everyone. Daydreaming also increases your chance of injury. Understanding this and the other basic guidelines in this section can help you stay safe and enjoy your time in the shop.

US Navy photo by Mass Communication Specialist 2nd Class Andrea Perez

Figure 3-1. None of the pilots in the Blue Angels flight demonstration squad would consider performing until all the plane's systems were thoroughly checked. In the same way, you should never operate a machine tool until you have determined that it is in safe operating condition.

3.1.1 Know the Materials

Do not machine a material until you know what it is and how it should be handled, and exercise extreme care when you are machining unfamiliar materials. For example, inhaling fumes or dust from some of the more advanced and exotic materials can cause serious respiratory ailments.

Magnesium is another 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.

3.1.2 Keep the Shop Clean

Place metal scraps in the scrap bin. Never allow them to remain on the bench or floor. These metal scraps can be very sharp and may cause injury if left on the bench or floor.

SAFETY NOTE

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.

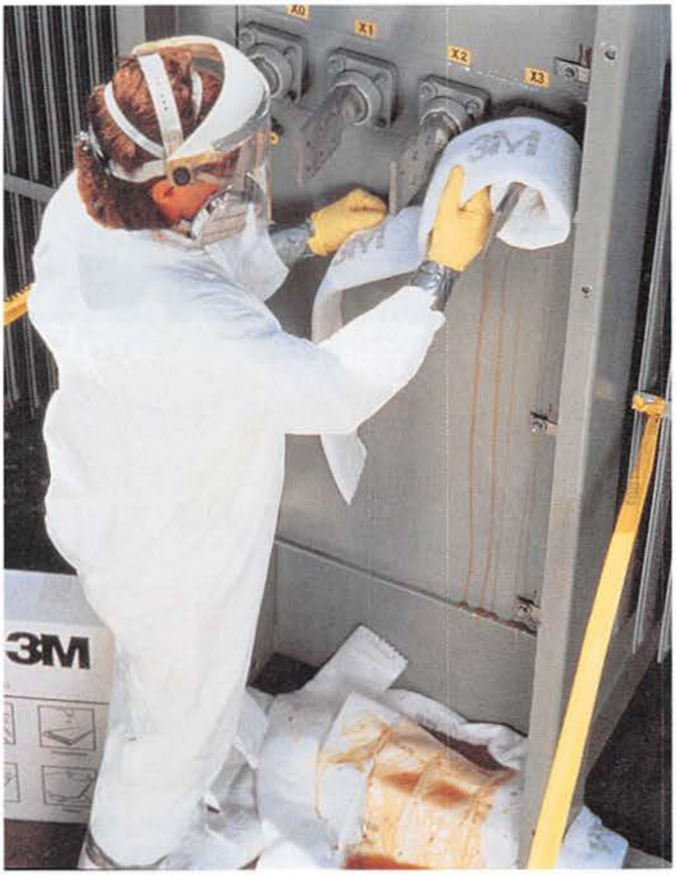
Return solvents and oils to proper storage place after use. Wipe up spilled oil or solvent right away. If the spill area is extensive, use an approved oil absorbent. See **Figure 3-2**.

Place oily rags in an approved safety container. See **Figure 3-3**. Rags or waste used to clean machines may also have metal slivers embedded in them, posing an additional hazard. Placing them in a safety container will help ensure that 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).

3.1.3 Wear Protective Clothing and Equipment

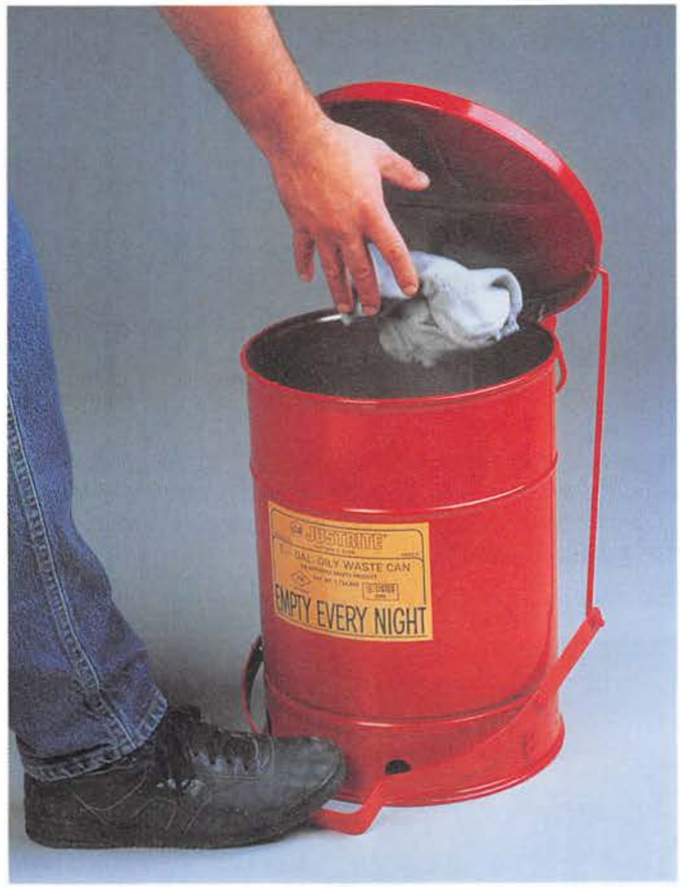
Dress properly for working around machinery. Severe injuries or even death can result if clothing, hair, or jewelry gets caught in moving parts. Avoid wearing loose-fitting clothing that could catch in machinery. A snug-fitting shop coat or apron can be worn to protect your street clothes, **Figure 3-4**. 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. When operating a machine with a rotating spindle, do not wear gloves. If a glove gets caught in the spindle, the operator's hand could be crushed.

An approved respirator and special ***protective clothing*** must be worn when machining some materials. For jobs where dust and fumes are a hazard, also ensure that there is adequate ***ventilation***. Machines must be fitted with effective vacuum systems as needed.



3M Company

Figure 3-2. 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.



Justrite Manufacturing Company

Figure 3-3. 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.

Wear appropriate ***safety equipment,*** **Figure 3-5**. 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. An ***approved respirator*** must be worn in areas where machining operations produce a mist of oil or coolants. See **Figure 3-6**. Suitable personal protective equipment must also be worn when handling sharp, hot, or contaminated materials.

Always protect your eyes. Wear eye protection whenever you are in the shop. Take no chances. Eyesight that has been damaged or destroyed cannot be replaced. Wear safety glasses, goggles, or face shields approved by the ***Occupational Safety and Health Administration (OSHA).***

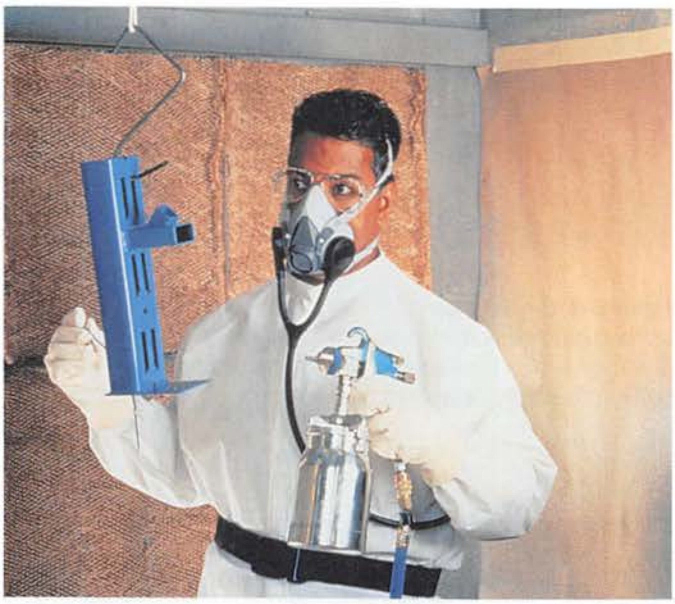
It is good practice to have your own personal safety glasses or face shield. The cost is reasonable. Your instructor can help you determine the style best suited for your needs, **Figure 3-7**. If you wear glasses, special safety lenses are available that can be ground to your prescription. Your eye doctor or optician can help get them.

Millersville University

Figure 3-4. 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.

Goodheart-Willcox Publisher

Figure 3-5. 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.



3M Company

Figure 3-6. 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.

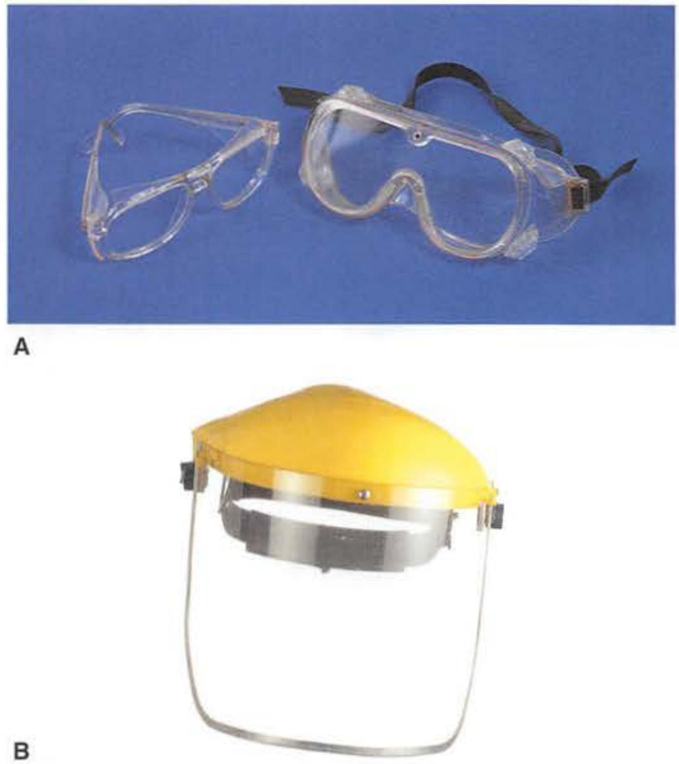
3.1.4 Use Common Sense

Other safety rules are a matter of common sense. First, *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.

Also, use care when handling long sections of metal stock. Accidentally contacting a light fixture with the stock, for example, could cause an electric shock, severe electrical burns, or even death.

Perform all lifting and moving carefully to avoid injury. When moving heavy machine accessories or large pieces of metal stock, always ask for help. The back injuries that can result from improper lifting are usually long-term injuries.

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. Even many over-the-counter cold remedies recommend that you do not operate machinery while taking the medication because of possible drowsiness.



A

B

Goodheart-Willcox Publisher; Eldred Lim/Shutterstock.com

Figure 3-7. A—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. B—A safety face shield protects the entire face from flying debris and splatters.

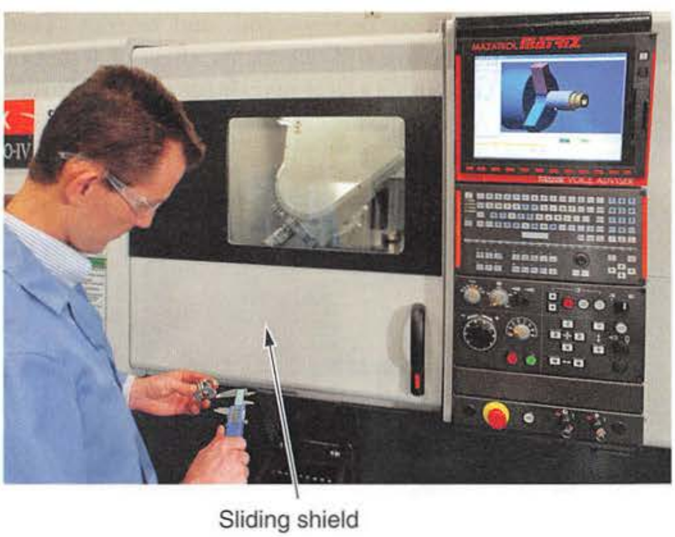
3.1.5 Use Safety Aids

Not all barriers provide physical protection from machine hazards. Awareness barriers serve to remind the operator that an area is dangerous. In its 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. See **Figure 3-8**.

Jenoptik

Figure 3-8. The yellow bars on each side of the front of the safety fence are a safety light curtain. A beam of light passes from one bar to the other. If anything or anyone moves between the bars, breaking the light beam, the safety system automatically shuts off the machine.

Machine shields 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 3-9**.



Mack Molding Co., Arlington, Vermont

Figure 3-9. Many CNC machine tools use sliding shields over the machining area to protect the machinist from flying chips and vaporized coolant.

Many different types of warning signs are used to notify workers of potential hazards, **Figure 3-10**. *No Smoking* signs must be posted in areas where flammable or combustible materials are used and stored.

3.2 General Machine Safety

In addition to general safety rules, some safety guidelines are specifically designed to keep workers safe while working with machines. These guidelines include:

- Never operate a machine until all guards are in place.
- Keep the floor around the machine clear of oil, chips, and metal scrap.
- Do not 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 3-11**. 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.
- 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.

Matee Nuserm/Shutterstock.com

Figure 3-10. Signs are often used to remind students and workers to wear personal protective equipment or be alert for potential hazards.

SAFETY NOTE

Always stop the machine before making adjustments or taking measurements. Resist the urge, while the machine is running, to touch a surface that has been machined. Severe lacerations can result.

3.3 General Tool Safety

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. In addition, follow these tool safety guidelines:

- 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 3-12**.
- Make sure tools are properly sharpened, in good condition, and fitted with suitable handles. Tools that are properly sharpened and maintained are less likely to cause injury than dull, poorly-maintained tools.

3.4 Fire Safety

Combustible materials are materials that are capable of burning. Fires are classified according to the type of combustible material that is burning, **Figure 3-13**. Extinguishers should have color-coded symbols to identify their appropriateness for a particular type of fire.

- **Class A fires.** Those involving ordinary combustible materials, such as paper, wood, and textiles. 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 Class D fires.

Photo courtesy of Weiler Corporation

Figure 3-11. Use a brush, not your hands, to remove accumulated chips.

Goodheart-Willcox Publisher

Figure 3-12. Arrange sharp-pointed tools on the bench in a way that they will not injure you when you reach to pick them up.

Fires
Class A Fires Ordinary Combustibles (Materials such as wood, paper, textiles.) *Requires: cooling/quenching* Old: A — New
Class B Fires Flammable Liquids (Liquids such as grease, gasoline, oils, and paints.) *Requires: blanketing or smothering* Old: B — New
Class C Fires Electrical Equipment (Motors, switches, etc.) *Requires: a nonconducting agent* Old: C — New
Class D Fires Combustible Metals (Flammable metals such as magnesium and lithium.) *Requires: blanketing or smothering* Old: D — New

Type	Use	Operation
Soda-acid Bicarbonate of soda solution and sulfuric acid	Okay for use on: A Not for use on: B C D	Direct stream at base of flame.
Pressurized Water Water under pressure	Okay for use on: A Not for use on: B C D	Direct stream at base of flame.
Carbon Dioxide (CO_2) Carbon dioxide (CO_2) gas under pressure	Okay for use on: B C Not for use on: A D	Direct discharge as close to fire as possible, first at edge of flames and gradually forward and upward.
Foam Solution of aluminum sulfate and bicarbonate of soda	Okay for use on: A B Not for use on: C D	Direct stream into the burning material or liquid. Allow foam to fall lightly on fire.
Dry Chemical	Multi-purpose type — Okay for: A B C; Not okay for: D Ordinary BC type — Okay for: B C; Not okay for: A D	Direct stream at base of flames. Use rapid left-to-right motion toward flames.
Dry Chemical Granular-type material	Okay for use on: D Not for use on: A B C	Smother flames by scooping granular material from bucket onto burning metal.

Goodheart-Willcox Publisher

Figure 3-13. Symbols coded by color and shape identify the four classifications of fire and the extinguishers that can be used on them.

- **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 non-conducting extinguishing agents that will smother the flames. Do not use Class A extinguishers on electrical fires.
- **Class D fires.** Extinguishers containing heat-absorbing dry powder are used on flammable metals, such as magnesium and lithium. Do not use Class A extinguishers on flammable metal fires.

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. Think before acting. It costs nothing, and you may be saved from painful injury that could result in a permanent disability.

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.

Fire blankets are sometimes an appropriate alternative to fire extinguishers. Fire blankets intended for industrial applications are generally made of wool, and are often chemically treated with a flame retardant fluid. They can be used on any type of fire where oxygen is needed to maintain the flame, because the blanket deprives the fire of oxygen, if placed properly. These blankets are particularly useful in smothering small fires that can be completely covered with the blanket, such as a fire contained inside a pan.

In some cases, fire blankets are the preferred method of putting out a fire. For example, fire extinguishers can spread out the fuel source, if not properly applied, and make a small fire into a larger one. A fire blanket can cover such a fire and extinguish it more effectively. Also, using chemical fire extinguishers on a person can pose serious health risks when the burns are exposed to the chemicals. If inhaled, the chemical from a fire extinguisher can be even more hazardous, even deadly. Fire blankets can be wrapped around a person whose clothing is on fire to extinguish the fire quickly.

GREEN MACHINING

Green Fire Safety

Fire safety is a concern in any machine shop, especially those that use flammable oil-based machining fluids. Recent advances in automated manufacturing have improved the manufacturing process but may leave combustible material and expensive equipment unattended. To protect against fires, some machine shops use on-machine fire suppression systems. These systems are set up to automatically detect fire and release a chemical fire extinguishing agent. Traditional fire extinguishers contain halon, a chemical that can cause ozone depletion when released into the atmosphere. New potassium-based extinguishing agents are more environmentally friendly. They do not cause ozone depletion, are nontoxic, and are easier for shop workers to clean up.

WORKPLACE SKILLS

Staying Safety Conscious

Safety on the job is everyone's responsibility. Many workplace accidents occur because of careless behavior. Often poor attitudes can cause unsafe behavior, too. Common causes of accidents include the following:

- Taking chances.
- Showing off.
- Forgetting safety details.
- Disobeying company rules.
- Daydreaming.
- Losing your temper.
- Falling asleep.

Practicing good safety habits is essential for preventing accidents and injuries on the job. A healthy worker is more alert and less likely to make accident-prone mistakes. Knowing how to use machines and tools properly is the responsibility of both the employer and employees. Wearing protective clothing and using safety equipment correctly helps keep workers safe. Your employer will emphasize the safety practices that employees must follow in your workplace.

The government agency that promotes safety in the workplace is the Occupational Safety and Health Administration (OSHA). You will be required to follow the specific OSHA regulations that apply to your workplace.

3.5 Safety Data Sheets

Safety data sheets (SDS), formerly called material safety data sheets (MSDS), provide information about the various chemicals, chemical compounds, and mixtures that are used in an industrial setting. A typical SDS has a leading paragraph identifying the manufacturer and describing the proper use of the information in the SDS. The sheet also contains 16 sections, as described in the table in **Figure 3-14.**

SDSs must be kept in a general location that is easily accessible by employees and by emergency response personnel in the event of an emergency. Some companies subscribe to an online service that makes SDS information immediately available on tablets and other mobile devices, **Figure 3-15.**

Required Sections in a Safety Data Sheet

Section	Description
Section 1. Identification	Provides a unique product identifier, as well as the contact information for the manufacturer, recommended uses, and any restrictions on the use of the chemical.
Section 2. Hazard(s) identification	Describes hazards associated with the chemical and required label elements.
Section 3. Composition/information on ingredients	Specifies chemical ingredients and trade secret claims.
Section 4. First-aid measures	Describes both acute (immediate) and delayed symptoms and effects, as well as required treatment.
Section 5. Fire-fighting measures	Describes chemical hazards from fire and lists appropriate equipment and techniques for extinguishing the fire.
Section 6. Accidental release measures	Explains proper measures for clean-up and containment of an accidental spill or release of the chemical. Also specifies emergency procedures and protective equipment needed.
Section 7. Handling and storage	Describes precautions for safe handling and storage, as well as incompatibilities.
Section 8. Exposure and controls/personal protection	Specifies exposure limits, including OSHA's Permissible Exposure Limits (PELs), ACGIH Threshold Limit Values (TLVs), and any other exposure limits recommended by the manufacturer, importer, or employer. Also includes recommended engineering controls and personal protective equipment (PPE).
Section 9. Physical and chemical properties	Specifies the physical and chemical characteristics of the chemical.
Section 10. Stability and reactivity	Lists the stability of the chemical and the possibilities for hazardous reactions with other substances.
Section 11. Toxicological information	Describes the toxicity of the chemical, including routes of exposure, symptoms of toxicity, acute and chronic effects of toxicity, and numerical measures of toxicity.
Section 12. Ecological information	Lists potential ecological effects of the chemical.
Section 13. Disposal considerations	Explains how to dispose of the chemical safely.
Section 14. Transport information	Describes transportation requirements and safety precautions for transport.
Section 15. Regulatory information	Lists regulations pertaining to the chemical.
Section 16. Other information	Lists the date the SDS was written or last revised.

Goodheart-Willcox Publisher

Figure 3-14. Required sections for a safety data sheet.

Safety Data Sheet

Section 1: Product and Company Identification

Product Name:	**Solvent Based Anti-Spatter (Aerosol)**
Product Identifier:	Anti-Spatter
Product Use:	Prevents Spatter Build Up in Welding Operations
Item Code(s):	BDAS-S-16OZC
SDS Code:	5001
Manufacturer:	Welding Material Sales
Physical Address:	1340 Reed Road Geneva, IL 60134
Business Phone:	630-232-6421
Business Fax:	888-733-1512
E-mail Address:	info@weldingmaterialsales.com
Emergency Phone:	800-424-9300
Date of Preparation:	May 10, 2015
OSHA Regulatory Status:	D1B, D2A, D2B, A
WHMIS Classification:	

Section 2: Hazards Identification

CLASSIFICATION OF THE SUBSTANCE OR MIXTURE:

CLP/GHS Classification (1272/2008):

Physical	Health	Environmental
None	Eye Irritation Category 2A Skin Irritation Category 2 Specific Target Organ Toxicity - Single Exposure Category 3 (H335, H336) Carcinogen Category 2	None

EU CLASSIFICATION (67/548/EEC): Xn R40 (Carcinogen Category 2)

Label Elements

WARNING! Contains methylene chloride

Hazard Phrases

H315	Causes skin irritation.
H319	Causes serious eye irritation.
H335	May cause respiratory irritation.
H336	May cause drowsiness or dizziness.
H351	Suspected of causing cancer.

Revised May 10, 2015 **Solvent Based Anti-Spatter Aerosol SDS** Page 1 of 6

Goodheart-Willcox Publisher

Figure 3-15. Safety data sheet information for every chemical in use at a facility must be made easily available to all employees.

Chapter Review

Summary

- Shop safety should be maintained throughout your machining career, not just in the training program, to help prevent injuries.
- Wear proper clothing that will not be caught in the machine.
- Use the appropriate safety equipment when working with machines to protect yourself from injury.
- Awareness barriers and machine shields are put in place to protect machine users.
- Get proper instruction before operating any machine.
- Warning signs are placed around a shop to identify potential hazards.
- Fires are classified into four different categories (A, B, C, and D) according to the type of material that is burning.
- Safety data sheets (SDS) are required for every chemical present in the machine shop and provide information about the chemical, including emergency information.

Review Questions

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

1. Why is shop safety so important?
2. Most shop accidents are caused by _____ or by not observing _____.
3. Why should compressed air *not* be used to clean chips from machine tools?
4. Why should oily rags be placed in a safety container?
5. When working in an area contaminated with dust or solvent fumes, wear a dust mask and be sure that there is adequate _____.
6. 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.
7. Why is it necessary to take special precautions when handling long sections of metal stock?
8. Why is it important to ask for help when moving heavy machine accessories or large pieces of stock?
9. What should you do before operating a machine tool if you are taking medication of any sort?
10. What is the purpose of a machine shield?
11. Use a(n) _____, *not* your hand, to remove chips and shavings.
12. Secure prompt _____ for any cut, bruise, scratch, or burn.
13. Always stop machine tools before making _____ or taking measurements.
14. Why should tools be kept properly sharpened and maintained?
15. What type of fire extinguisher should be used to put out an electrical fire?
16. Briefly describe the information available on a safety data sheet.

CHAPTER 4 Understanding Drawings

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Read drawings that are dimensioned in fractional inches, decimal inches, and metric units.
- Explain how and why coordinate dimensioning is used.
- List the information found on a typical drawing.
- Explain how prints are generated.
- Describe how detail, subassembly, and assembly drawings differ.
- Explain the purpose of a parts list or bill of materials.
- Recall standard drawing sheet sizes.

Technical Terms

American National Standards Institute (ANSI)
bilateral tolerance
bill of materials
dimensions
dual dimensioning
International System of Units (SI)
limit dimensions
microinch
micrometer (micron)
parts list
profilometer
scale drawing
SI Metric system
tolerance
unilateral tolerance
US Conventional system
working drawing
X-Y coordinate system

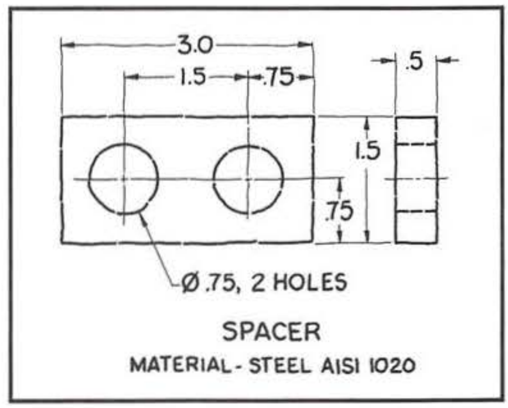
Many products manufactured today are an assembly of parts supplied by a number of different industries located all over the world. It would not be possible for industry to manufacture a complex product like the passenger plane in **Figure 4-1** without using drawings. Drawings show the machinist what 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.

IM_photo/Shutterstock.com

Figure 4-1. Thousands of drawings were required in the design and construction of aircraft like this passenger aircraft. Standards and specifications had to be exact because the components were manufactured in several geographic locations.

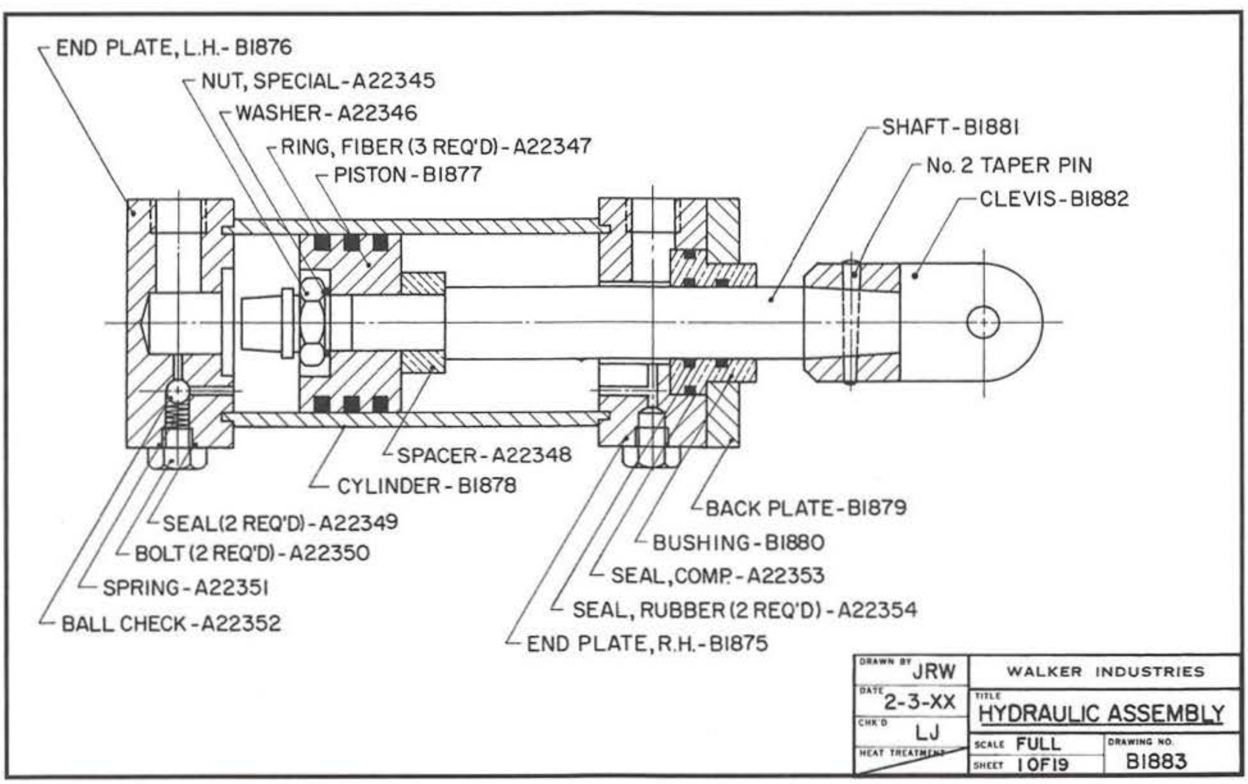
Drawings range from simple freehand sketches, **Figure 4-2**, to detailed drawings for complex products, **Figure 4-3**. Lines of different types and thicknesses are used to give drawings meaning, **Figure 4-4**. This "Alphabet of Lines," devised by the ***American National Standards Institute (ANSI)***, is standardized so that each type of line has the same meaning wherever drawings are made and used.

- Visible lines are used to draw the edges of objects and features that can be seen in each view. Visible lines are drawn with a thick lineweight so the part features are easily visible after other linetypes are added.



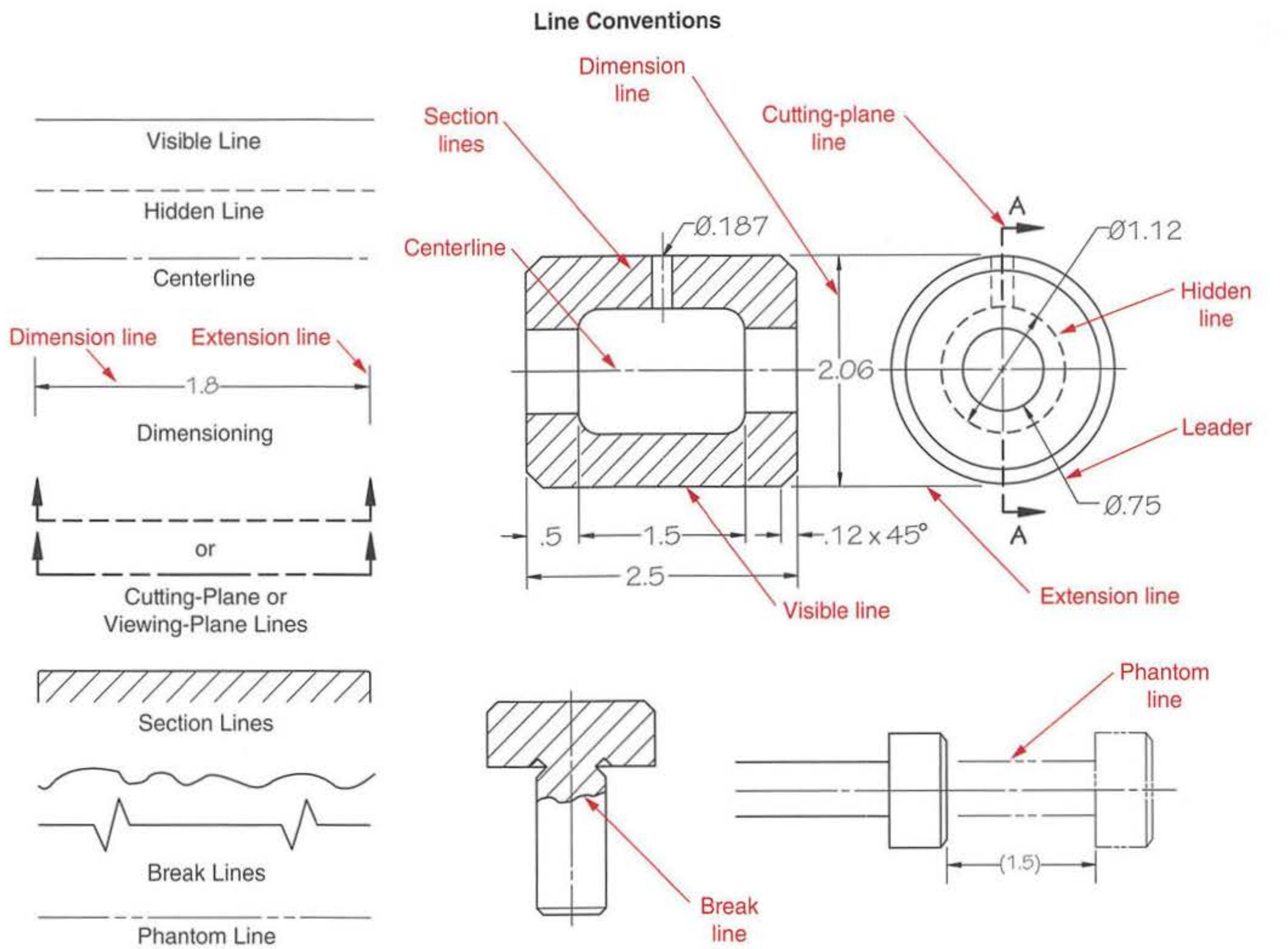
Goodheart-Willcox Publisher

Figure 4-2. Some drawings are as simple as this freehand sketch.



Goodheart-Willcox Publisher

Figure 4-3. Each manufactured product may require dozens of drawings, one for each part. Even the smallest screw, washer, or pin may require a drawing.

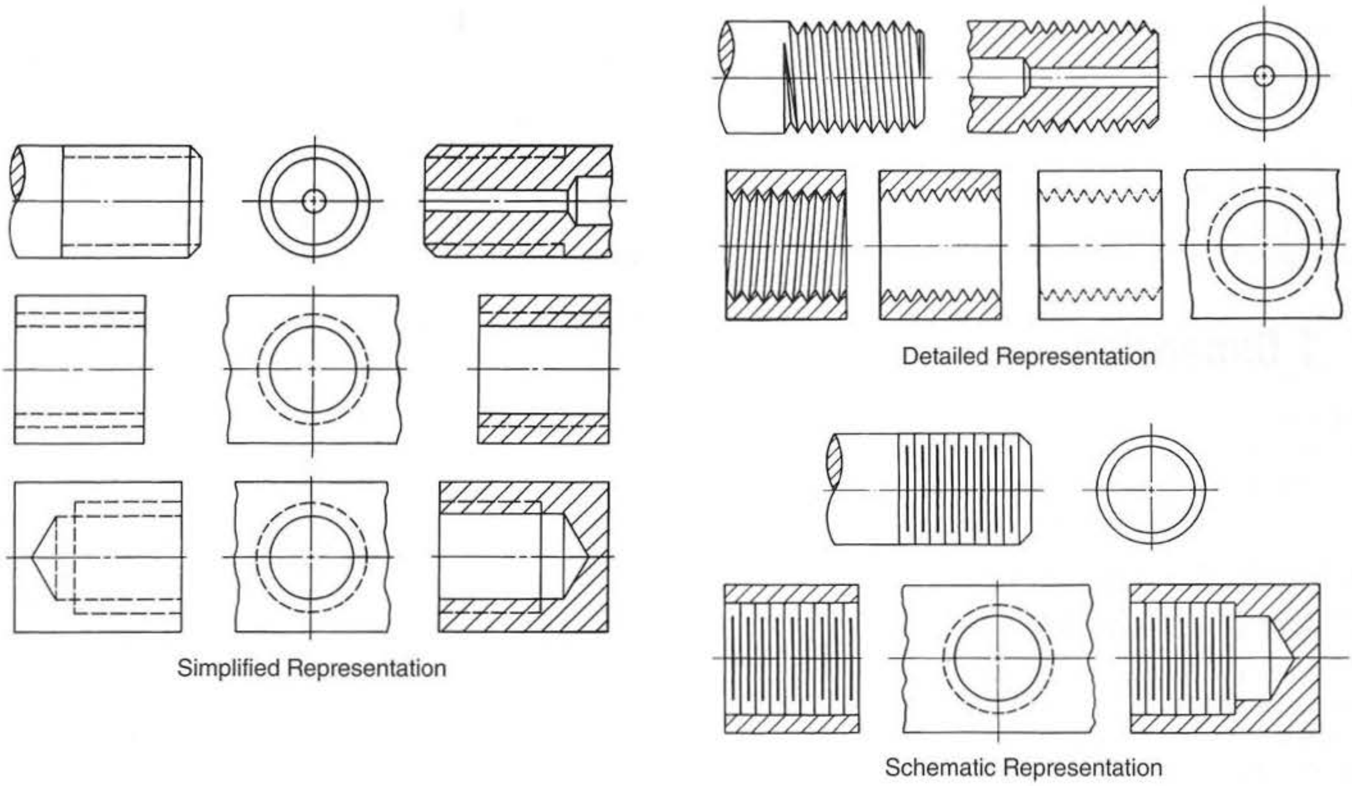


Goodheart-Willcox Publisher

Figure 4-4. Lines, symbols, and other elements give meaning to a drawing and indicate how parts should be assembled. The "Alphabet of Lines" is a standardized list of lines used on drawings. Each linetype has a definite pattern and form, and a specific purpose and meaning.

- Hidden lines are dashed lines of medium width that indicate edges that are not visible in the view.
- Centerlines indicate the center axes of holes, keyways, and similar features. Centerlines cross at the center of a hole that is visible as a circle to indicate the location of the hole's center point.
- Extension lines extend from edges or key points on the drawing and form the endpoints for dimension lines. Extension lines do not touch the object lines that define part edges; a slight gap is left between them so the reader will know the lines are not part of the actual part edge.
- Dimension lines extend between extension lines to show the actual dimension or size of the item or part being dimensioned. The size measurement appears on or above the dimension line. Arrowheads on the ends of the dimension line indicate the span of the dimension value included with the dimension line.
- Cutting-plane lines are used to define an imaginary cutting plane that defines a section view. Section views help show internal features that are usually shown with hidden lines. When a part has multiple internal features, it is difficult to distinguish the individual features. A section view is a view that shows the interior with a section of the part "cut away" to provide a direct view of the features.
- Section lines define the surfaces that are "cut" by the cutting-plane line when the section view is created.
- Break lines are used to show where part of a long, uniform item has been "cut out" of a drawing to make the drawing fit better on the drawing or to show the edge of a broken-out section.
- Phantom lines are generally used to show an alternate placement or position of a part.

In addition to various types of lines, drawings include other information needed to make the product. **Figure 4-5**



Goodheart-Willcox Publisher

Figure 4-5. Methods used to depict threads on drawings. The style must be used consistently within a drawing. The simplified version is the most common style.

CAREER CONNECTION

CAD Technician

What does a CAD technician do?

CAD technicians work with engineers, architects, and other designers to convert designs into digital technical drawings. They use computer-aided-drafting (CAD) to draw and structure designs digitally. Building information modeling (BIM) is also used to render interactive 3D models from the original technical specifications. Once designs become digital technical drawings or models, they can be stored, shared, and edited via computer as the project progresses. Information collected with BIM can even be used to maintain a structure during its lifetime.

What education and skills are needed to be a CAD technician?

Highly specialized training from technical schools or community colleges is required for CAD technicians. Most jobs will require an associate's degree in drafting or a certificate from a technical program. Coursework in these programs often covers design fundamentals, sketching, and creating drawings with CAD. High school students interested in drafting should prepare themselves with courses in mathematics, science, computer technology, drafting, and design.

What is it like to be a CAD technician?

CAD technicians most often work in the architectural, engineering, manufacturing, and construction industries. While they primarily work in offices, CAD technicians may travel to jobsites to monitor the project and refine the design, if needed.

According to the *Occupational Outlook Handbook*, annual wages for CAD technicians range from $49,000 to $59,000.

shows the three different conventions for drawing threads. Of these, the simplified representation is most often used.

Periodically, ANSI changes standard drawing symbols. Machinists must be familiar with past and present practices because only recently made drawings follow the new standards. It is too expensive to revise the millions of drawings made before the new standards were adopted. **Figure 4-6** shows past and present dimensioning symbols used for various types of holes on a drawing.

4.1 Dimensions

A proper drawing includes all ***dimensions*** (exact sizes or measurements) needed to produce the part or object. The parts and dimensions are shown in proper relation to one another and provide a detailed blueprint for creating the part.

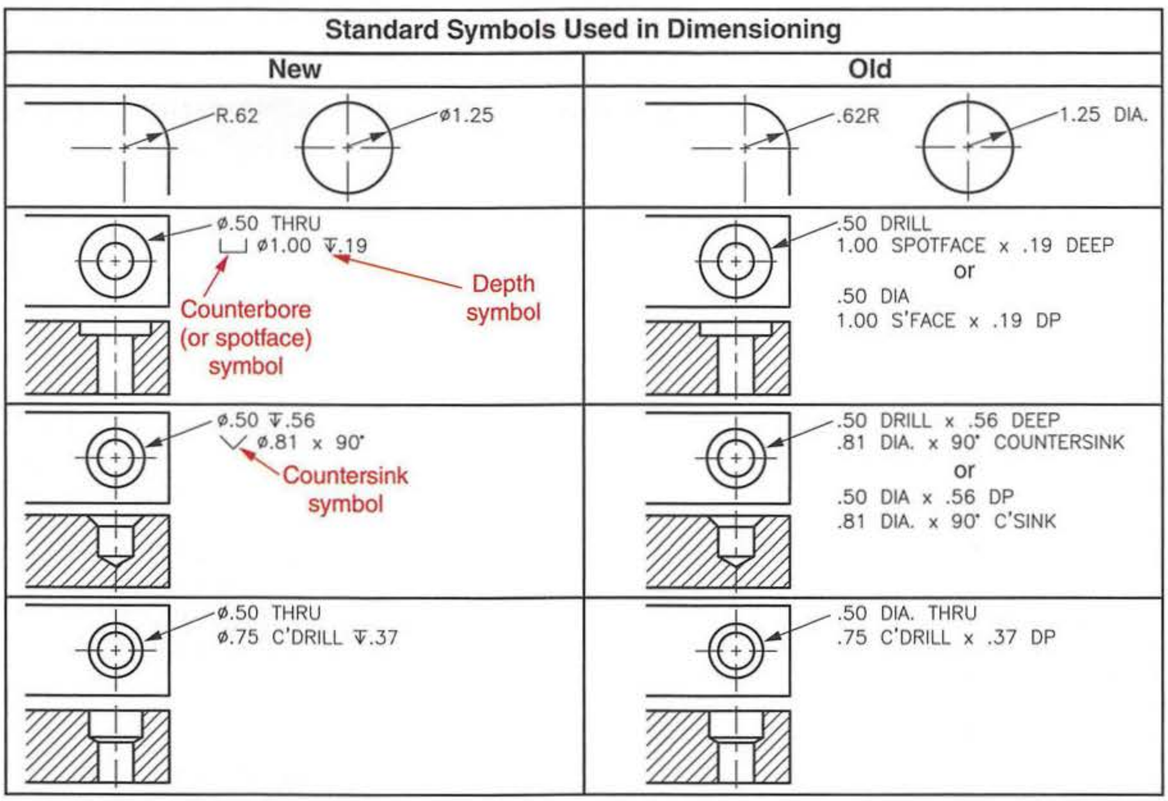
4.1.1 US Conventional and Metric Dimensioning

Drawings prepared using the ***US Conventional system*** of measurement are typically dimensioned in either fractional or decimal inches. Drawings using fractional dimensioning usually show objects that do not require a high degree of precision in their manufacture. For parts that require greater precision, dimensions are given in decimal parts of an inch. Fractional dimensioning is rarely used in mechanical drawings today, but older drawings may still use this format.

In metric dimensioning, all of the dimensions on the drawing are in the ***SI Metric system***. *SI* stands for the French words *Système International*), and this system is also called the ***International System of Units (SI)***. For most applications, the dimensions are in millimeters.

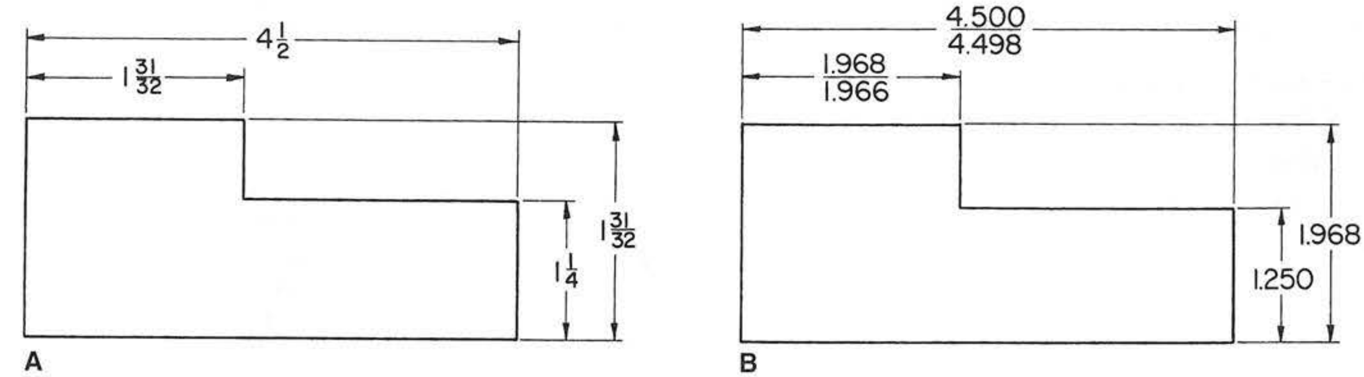
4.1.2 Dual Dimensioning

Until recently in the United States, drawings were dimensioned using only US Conventional units, **Figure 4-7**. However, because most of the rest of the world uses the metric system, US industries that compete internationally may produce drawings using metric measurements as well. ***Dual dimensioning*** is a system that uses both US Conventional units and metric dimensions on the same drawing, **Figure 4-8**. If the drawing is intended primarily for use in the United States, the decimal inch appears above the metric dimension, as in **Figure 4-9A**. The reverse is true if the drawing is to be used in a metric-oriented country, as in **Figure 4-9B**. Some companies place the metric dimension within brackets, as in **Figure 4-9C**.



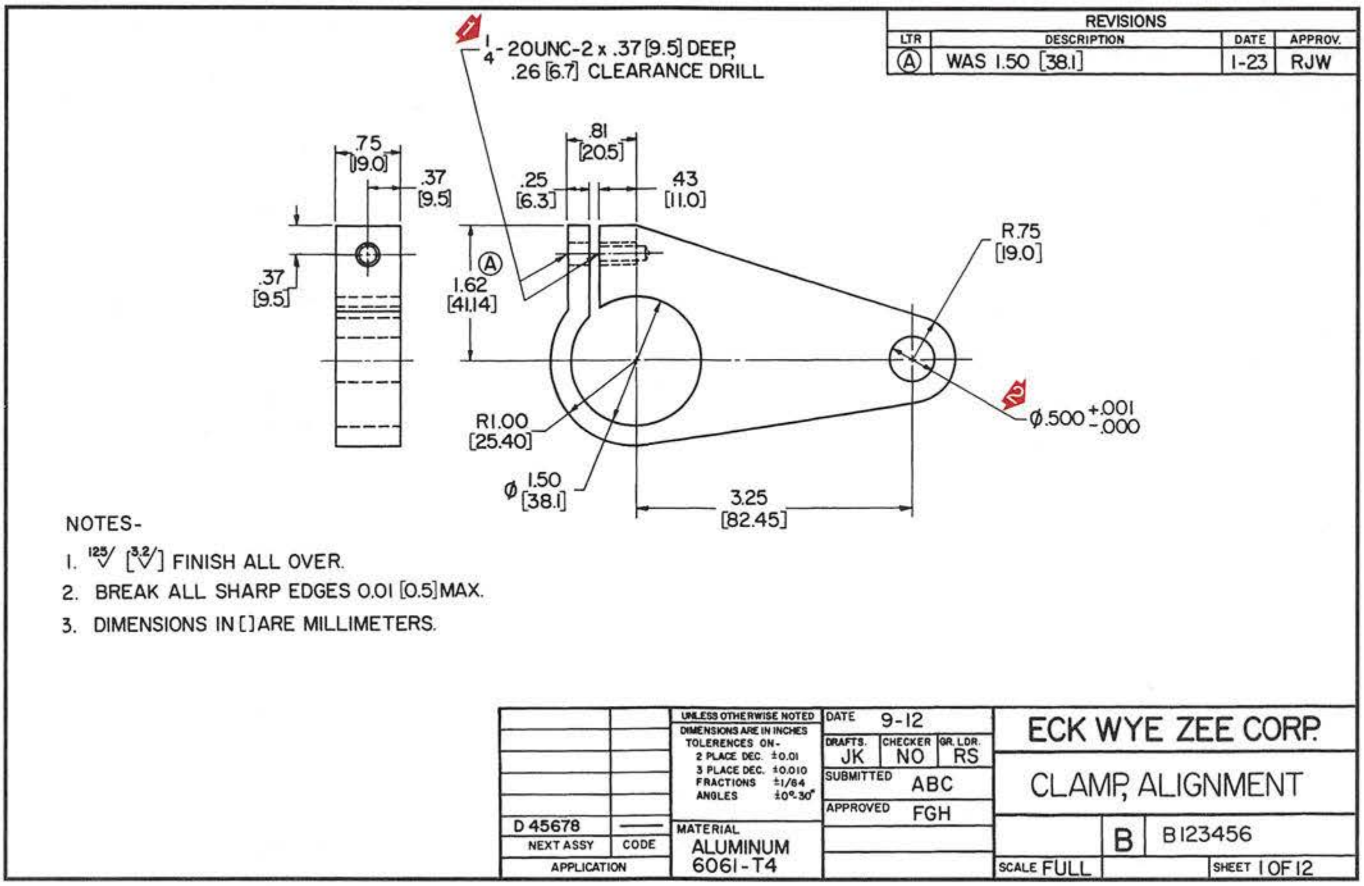
Goodheart-Willcox Publisher

Figure 4-6. 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 of hole representations.



Goodheart-Willcox Publisher

Figure 4-7. Dimension styles. A—Fractional dimensions do not require tolerances closer than ±1/64″. B—Decimal dimensions normally have tolerances of ±0.010″ or 0.015″.

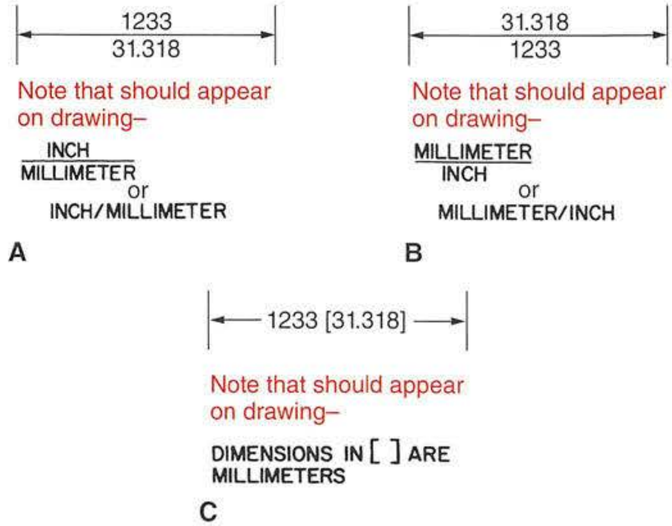


Goodheart-Willcox Publisher

Figure 4-8. 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.

4.1.3 Coordinate Dimensioning

Coordinate dimensioning is used on drawings with several features that, dimensioned individually, would require multiple dimension lines, extension lines, and dimensions. The crowding that would result would make it difficult to interpret the drawing properly. To prevent the clutter and make the drawing more easily readable, an ***X-Y coordinate system*** can be used to identify the exact location of each feature in terms of its location along an X (horizontal) axis and a Y (vertical) axis. First, a feature on the part, such as a corner or the center of a hole, is defined as point 0,0. Then each feature is labeled on the drawing. A table lists each feature, along with its position on the X and Y axes. Positive distances are given to the right on the X axis and upward on the Y axis.



Goodheart-Willcox Publisher

Figure 4-9. 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.

For example, the center of a hole located at coordinates 3,2 is three units to the right of point 0,0 and two units above it. Negative coordinates represent distances to the left (X axis) and downward (Y axis). The coordinate table provides all the information needed to create the part while keeping the drawing clear of confusing markings. **Figure 4-10** shows an example of coordinate dimensioning.

4.2 Information Included on Drawings

Drawings contain additional information to inform the machinist of the material to be used, required surface finish, tolerances, units used, scale, assembly and subassembly instructions, past revisions, and the name and part number of the object. Most of this information is located in the title block, which is usually found in the lower-right corner of the drawing.

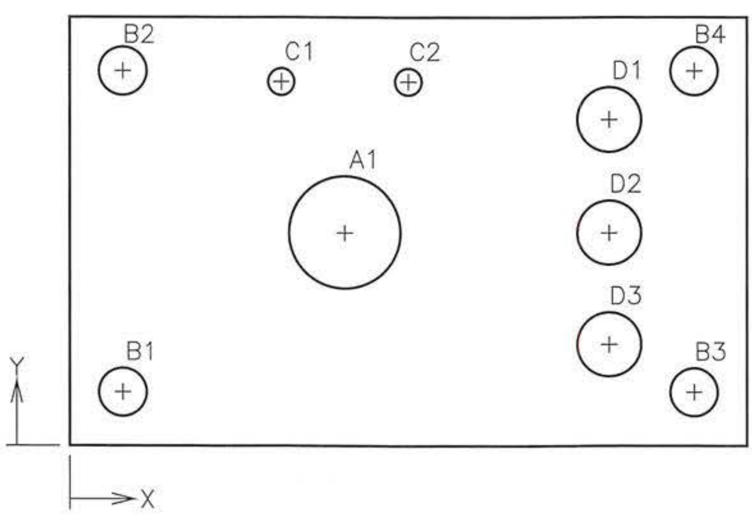
4.2.1 Materials

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 4-11**. Exact material specification is included in a section of the title block. See item A in **Figure 4-12**. Sometimes, the material specification may be found in notes shown elsewhere on the drawing.

4.2.2 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. Smoother finishes usually require additional operations or more precise machining operations, which makes the parts more expensive to manufacture.

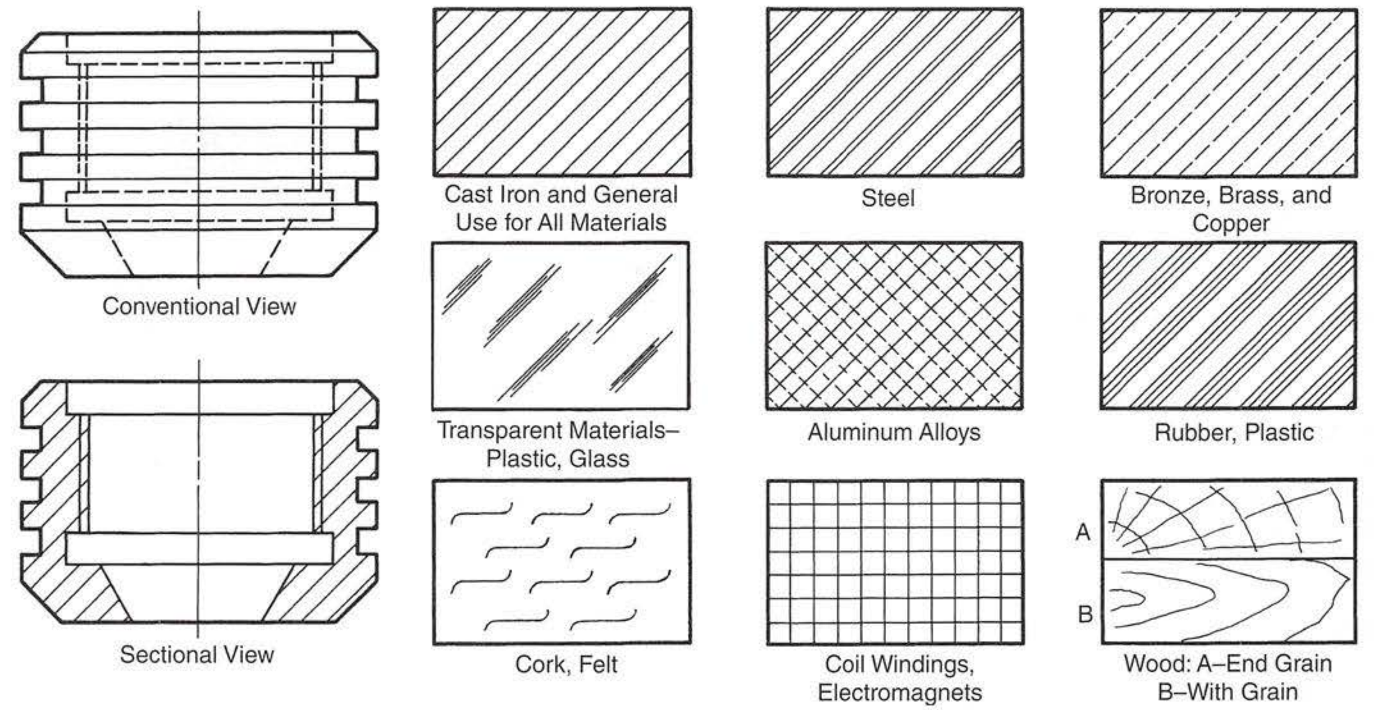
In the past, symbols were used to indicate machined surfaces, **Figure 4-13**. These symbols may still be found on some older drawings. With so many machining techniques now in use, symbols such as these do not provide sufficient detail about the surface finish required on a part.



ID	X	Y	ø
A1	3.25	2.50	1.3125
B1	.625	.625	.5625
B2	.625	4.375	.5625
B3	7.375	.625	.5625
B4	7.375	4.375	.5625
C1	2.50	4.25	.3125
C2	4.00	4.25	.3125
D1	6.375	3.8125	.75
D2	6.375	2.50	.75
D3	6.375	1.1875	.75

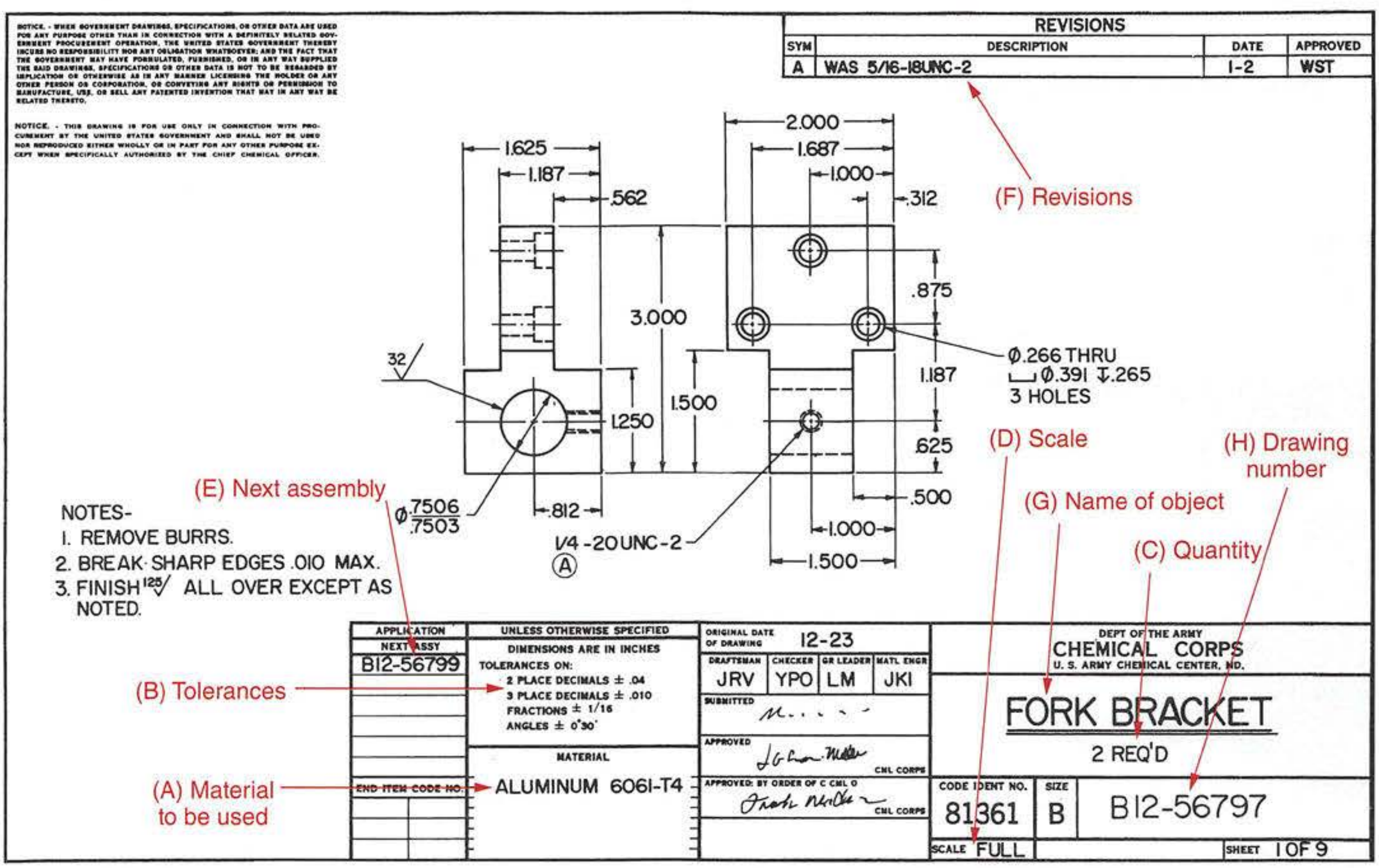
Goodheart-Willcox Publisher

Figure 4-10. Coordinate dimensioning removes the clutter from a drawing to make it more easily readable. Using a table for the coordinates is also useful when a part is to be created in several different sizes. The drawing can remain the same, and only the dimensions in the table change.



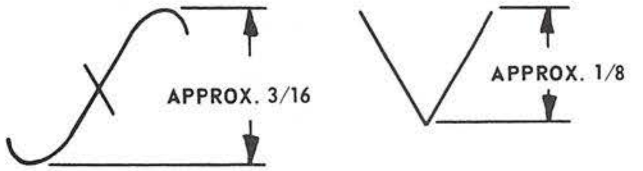
Goodheart-Willcox Publisher

Figure 4-11. Sectional views make a drawing easier to understand because internal details are shown more clearly. Various materials may be identified with unique section lines that have specific meanings. However, many sectional views are created using general section lining regardless of the material being used.



Goodheart-Willcox Publisher

Figure 4-12. A great deal of information is located in the drawing's title block. The items highlighted here are standard on most drawings.

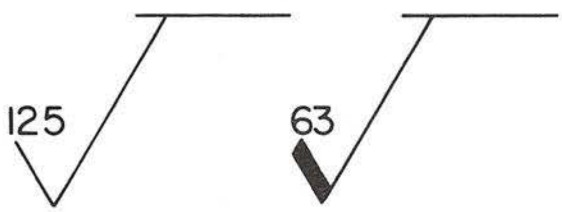


Goodheart-Willcox Publisher

Figure 4-13. These older-style finish marks do not indicate precisely enough the degree of smoothness required; they simply specify that the surface is machined. These finish marks are still found on older drawings.

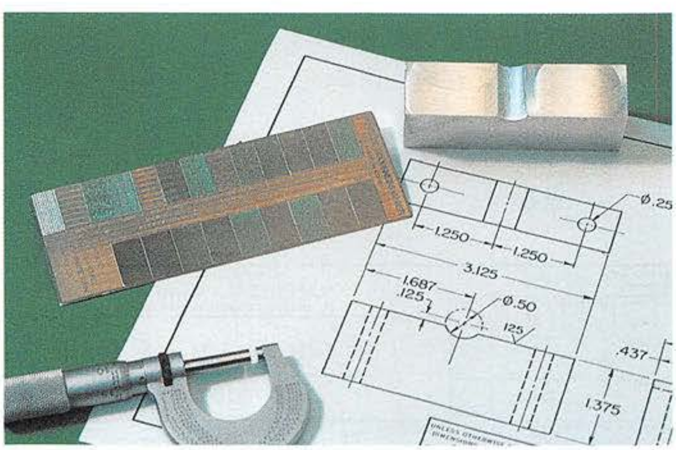
The method currently used provides more complete surface information. Shown in **Figure 4-14**, a check mark and number are used to indicate surface roughness in microinches or micrometers. A ***microinch*** is one-millionth of an inch (0.000001″). A ***micrometer (micron)*** is one-millionth of a meter (0.000001 m) and is abbreviated μm.

A machinist compares surface finishes to a surface roughness comparison standard to determine whether it meets the required specifications, **Figure 4-15**. 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 4-16**.



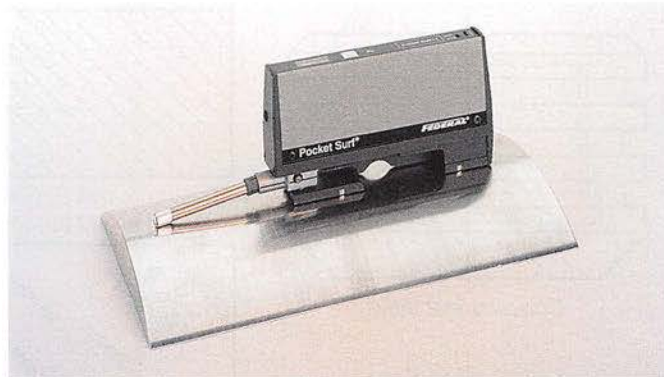
Goodheart-Willcox Publisher

Figure 4-14. Current surface finish marks. The number indicates the degree of smoothness in microinches—the larger the number, the rougher the finish.



Goodheart-Willcox Publisher

Figure 4-15. A surface roughness comparison standard is used to check whether a milled surface meets the required specifications.



Federal Products Co.

Figure 4-16. Surface roughness is most accurately determined with a profilometer or electronic surface roughness gage. The probe on the unit measures surface roughness electronically as it is moved across the surface of the work. A digital display presents the measured roughness value in microinches or micrometers.

4.2.3 Tolerances

The control of dimensions to achieve interchangeable parts is known as *tolerancing*. A ***tolerance*** controls the size of a part's features. It is an allowance greater than and/or less than a stated dimension size that is permitted when machining or making a part. Refer to item B in **Figure 4-12**. Acceptable tolerances are shown on drawings in several different ways.

When the dimension is given in fractional inch units, the tolerance is 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. 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, than the stated dimension, the dimension on the drawing might read as follows:

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

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

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

When the tolerance is allowed in one direction only (plus or minus, but not both), it is called a ***unilateral tolerance***.

For drawings dimensioned in decimal inches, unless otherwise indicated, the tolerances are assumed to be ±0.001″. The part is acceptable if the machined dimensions measure within these limits. A *plus* tolerance may be shown as follows:

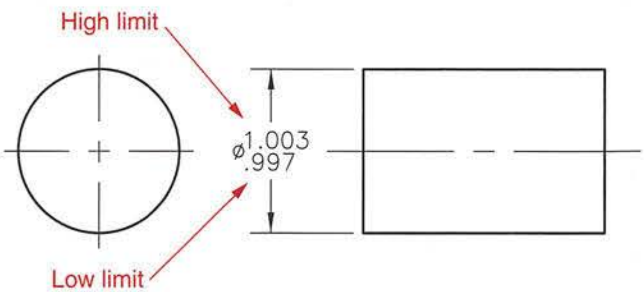
$$2.500^{+.001} \text{ or } 2.501/2.500$$

A *minus* tolerance may be shown as follows:

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

Often, to make it easier for the machinist to read the drawing, ***limit dimensions*** are used, **Figure 4-17**. Instead of providing the dimension and a tolerance, the dimension provides the largest and smallest acceptable size. For example, for a dimension of 2.500 with a plus tolerance of +.001, the dimension would be shown as 2.501–2.500 instead of $2.500^{+.001}$. This takes up more space on the drawing, but it eliminates the possibility of the machinist making an error in calculating the limits.

Metric tolerances are presented in the same way as decimal tolerances. **Figure 4-18** shows an example of a metric drawing with limit dimensions.



Goodheart-Willcox Publisher

Figure 4-17. An example of limits dimensioning. The base dimension is not included. Only the lowest and highest possible sizes are shown in the dimension.

4.2.4 Other Information

Other important information is also included in the title block and revision block of a drawing. These items are briefly described in this section.

Quantity of Units

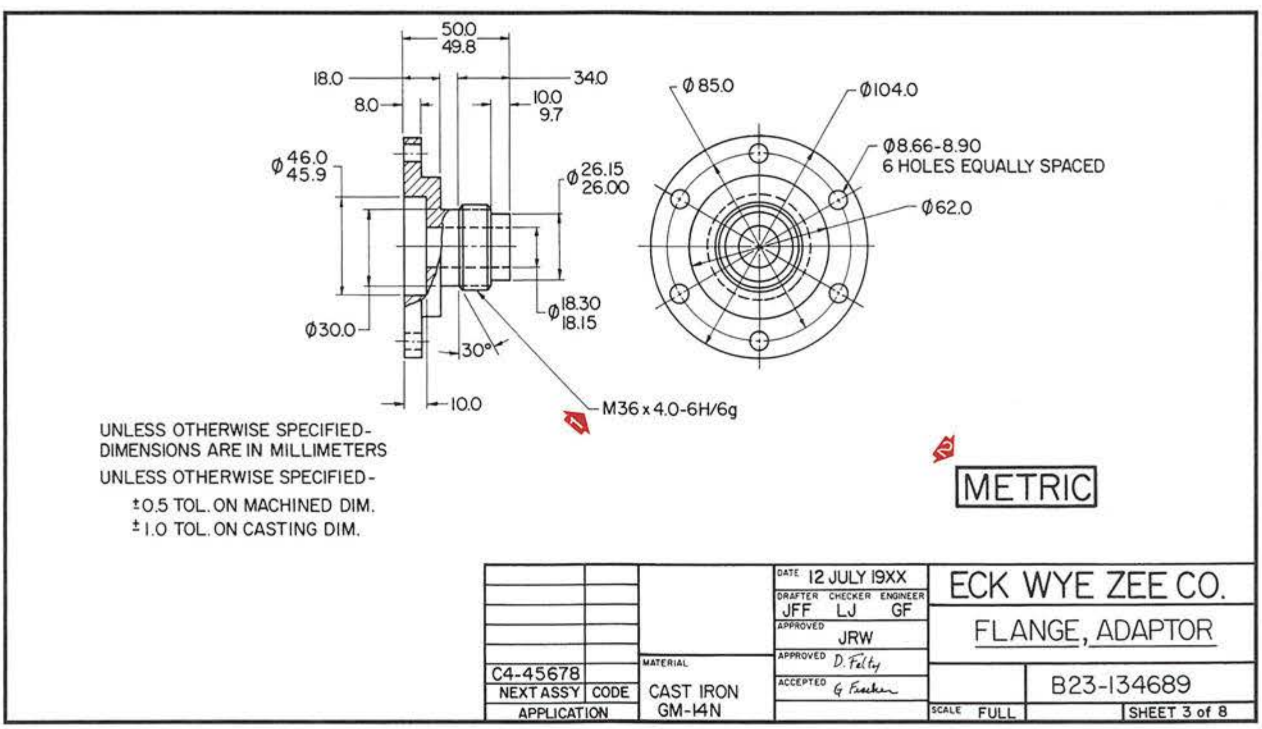
The number of parts (quantity) needed in each assembly is included on the drawing. Refer to item C in **Figure 4-12**. A work order, included with the job information received by the shop, gives the total number of units to be manufactured. This facilitates ordering the necessary materials and helps determine the most economical way to manufacture the pieces.

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 (item D in **Figure 4-12**). 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.

Assembly or Subassembly

Assembly or subassembly information is necessary to correctly fit the various parts of a multiple-part object together.



Goodheart-Willcox Publisher

Figure 4-18. 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, the word *METRIC* is placed on the drawing in large letters.

Notice the assembly code at item E in **Figure 4-12**. The term *application* is sometimes used in place of the term *next assembly*.

Revisions

Revisions indicate what changes were made to the original drawing and when they were made. The revisions are often shown in a separate revisions box. Refer to item F in **Figure 4-12**.

Name and Part Number of the Object

The title block includes both the name of the part and its assigned part number. Refer to item G in **Figure 4-12**.

4.3 Prints

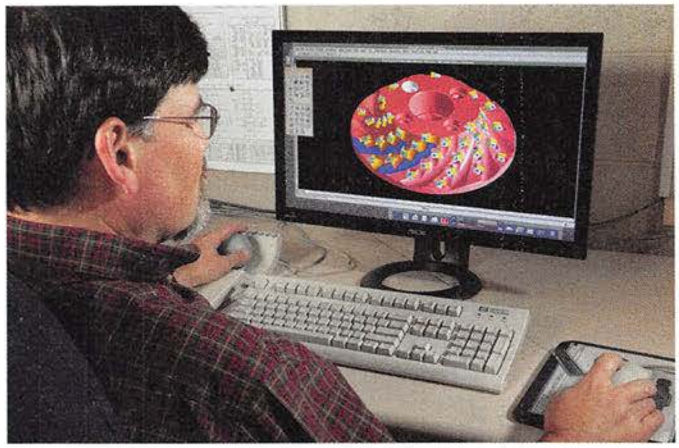
On many jobs, several sets of plans are required. In the past, a variety of techniques existed to reproduce drawings. Today, most drawings are created using computer-aided drafting (CAD) software. Both 2D drawings and 3D models can be created using CAD software. Prints can be generated from the drawings or models and are then viewed electronically or printed using a printer or plotter (automatic drafting machine), **Figure 4-19**.

This same information can also be used to control machine tools, using computer-aided manufacturing (CAM) software. When these methods are used, the overall manufacturing technique is called *computer integrated manufacturing (CIM)*. Computer numerical control (CNC) machining uses CAD models to carry out precisely the sequence of operations needed to produce a part. More information about the use of computers in manufacturing is provided in later chapters of this book.

GREEN MACHINING

Electronic Drawings and CAD

Digital or electronic blueprints have become more common for companies in construction and machining. This greener practice of blueprinting can be used to increase jobsite efficiency, cut costs, and significantly reduce paper consumption. More than 42,000 trees are used each year to create blueprints. When there are miscalculations or errors in prints, more paper consumption is required for corrections and reprints. Digital blueprints can be distributed and accessed through either PDF or CAD files. Digital prints allow designers to make changes, calculations, and status updates to their drawings without extra time for printing and additional paper use. CAD programs can also produce 3D sketches of blueprints to provide a more accurate view of the design.



Lovejoy Tool Company, Inc.

Figure 4-19. A computer-generated 3D model of an indexable insert cutting tool.

4.4 Types of Drawings Used in the Shop

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

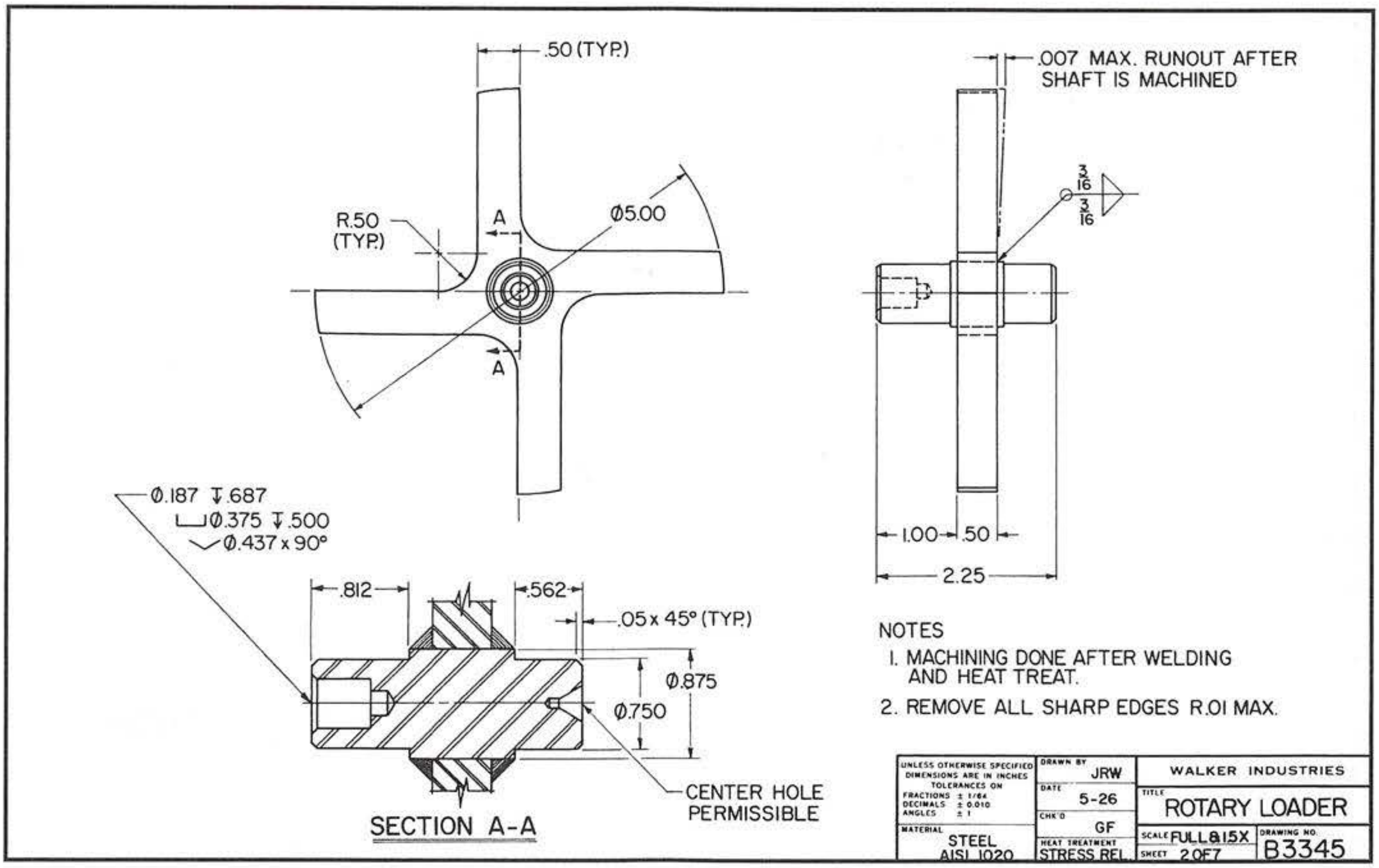
- **Detail drawings.** These consist of a drawing (usually multiview) of the part with dimensions and other information for making the part, **Figure 4-20**.
- **Assembly drawings.** These drawings show where and how the parts described on the detail drawings fit into the completed assembly. See **Figure 4-21**. On large or complex products, subassembly drawings are used to show the assembly of a small portion of the completed object, **Figure 4-22**.

Some assembly and subassembly drawings are shown as exploded pictorial drawings (a drawing with parts separated, but in proper relationship to one another). An example of an exploded pictorial drawing is shown in **Figure 4-23**.

In most instances, a detail drawing provides information on just one item or part. 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 4-24**.

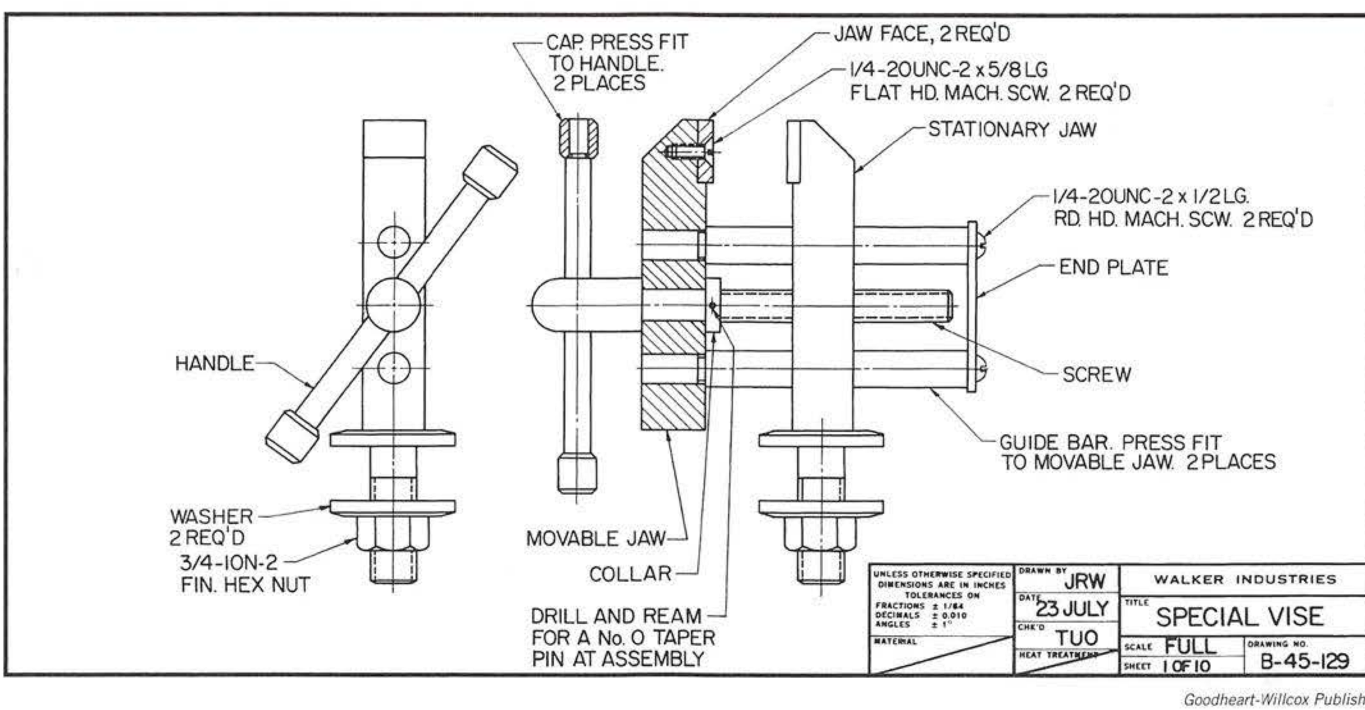
4.5 Parts List

Parts are identified by circled numbers and are listed in a note. For assemblies that include many parts, a ***parts list*** or ***bill of materials*** is developed. Both a parts list and a bill of materials list the name of the part, assign it an item number, and list the quantity needed to produce the item. Therefore, many people use the terms *parts list* and *bill of materials*



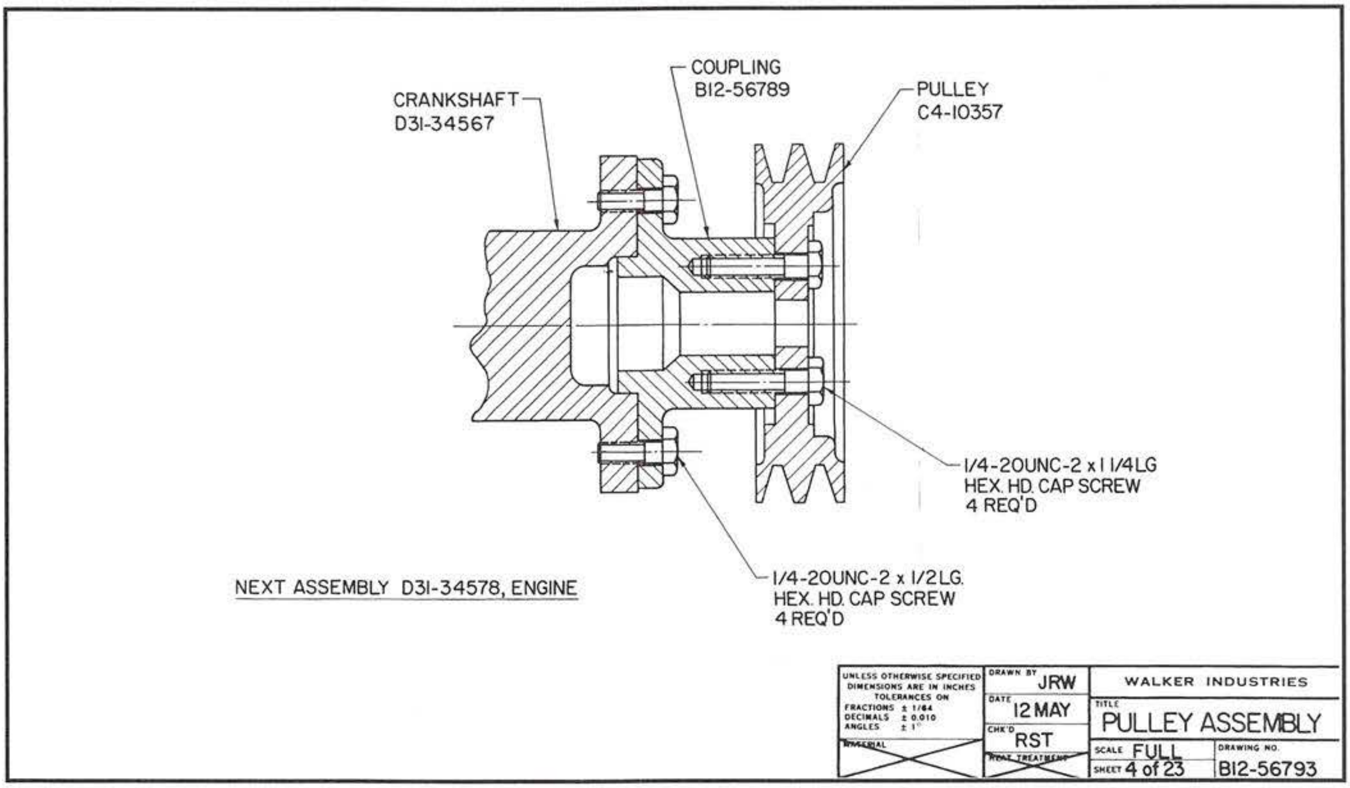
Goodheart-Willcox Publisher

Figure 4-20. A detail drawing contains all of the information needed to produce the part.



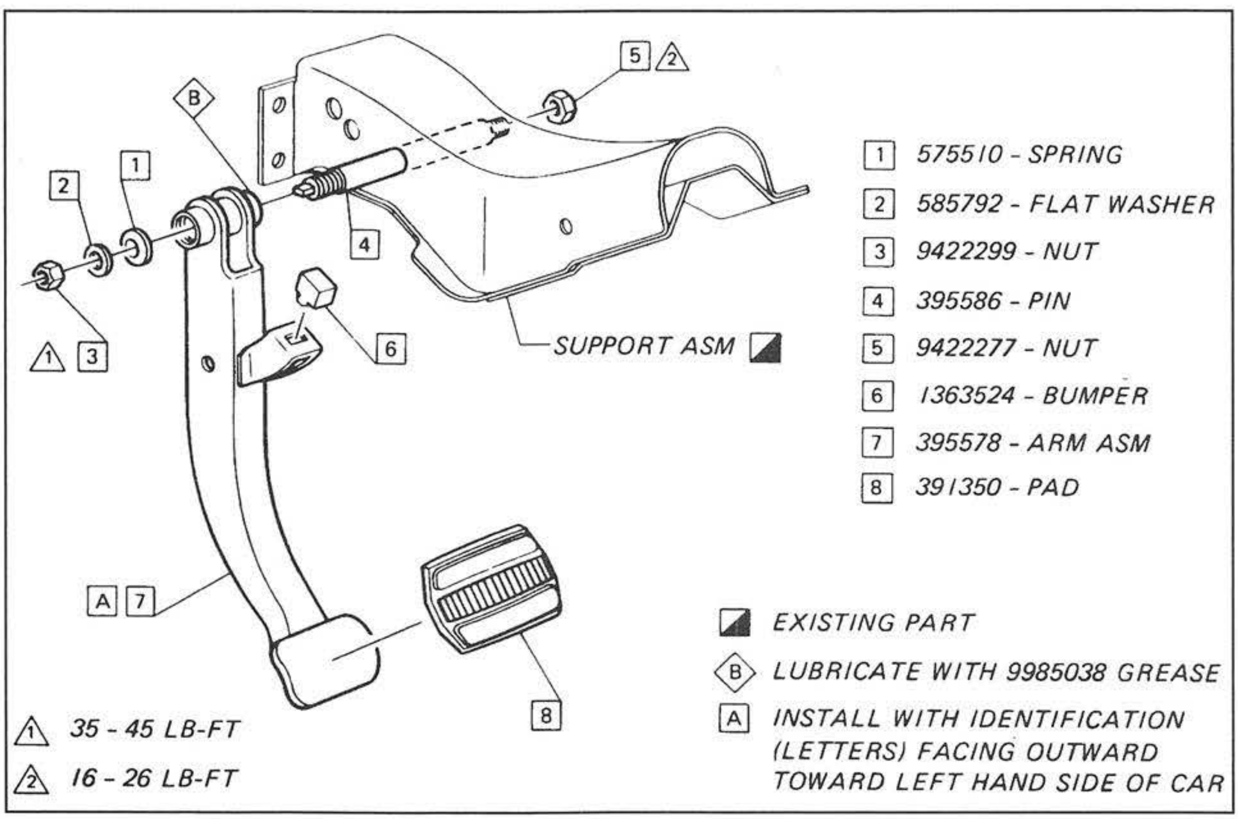
Goodheart-Willcox Publisher

Figure 4-21. An assembly drawing shows how various parts fit together.



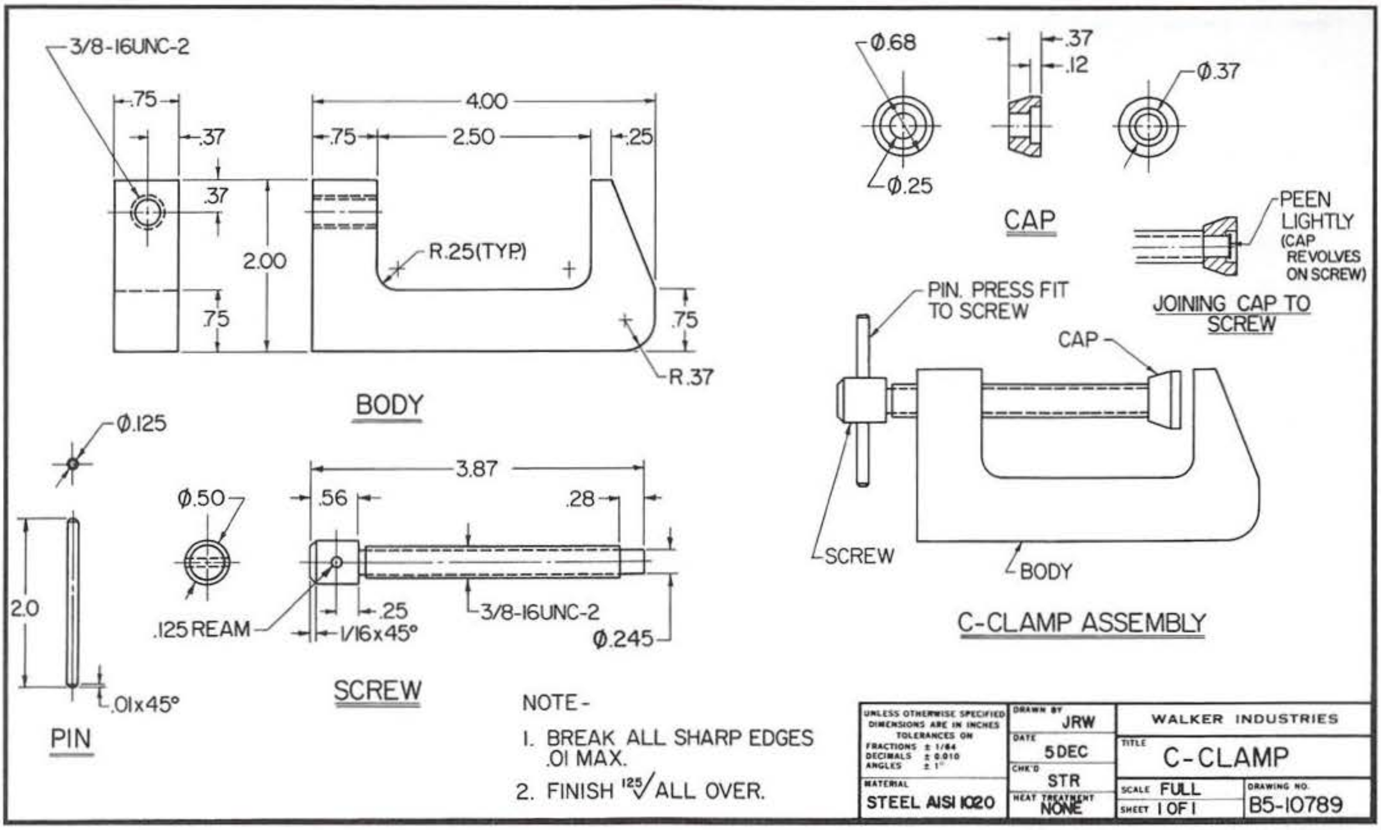
Goodheart-Willcox Publisher

Figure 4-22. A subassembly drawing contains the assembly of only a portion of the entire product.



General Motors Corp.

Figure 4-23. Exploded pictorial drawings are often created for semiskilled workers who have received a minimum of training in print reading.



Goodheart-Willcox Publisher

Figure 4-24. A combined detail and assembly drawing.

interchangeably. A bill of materials, however, is more likely to include more information, such as the material from which the part is made and an actual part number from a supplier, as opposed to a simple list of parts. See **Figure 4-25.**

Also, for convenience in filing and locating drawings in storage, each drawing has an identifying number. See item H in **Figure 4-12.**

4.6 Drawing Sizes

Most firms centralize the preparation and storage of drawings in the engineering department. Generally, engineers and drafters prepare drawings on standard-size sheets. This simplifies stocking, handling, and storing the completed drawings.

Standard sizes for drawing sheets include the following:

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

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″

A

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

B

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			

Goodheart-Willcox Publisher

Figure 4-25. A list of parts or a bill of 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.

Chapter Review

Summary

- Drawings may be dimensioned in fractional or decimal inches or in metric units.
- For parts that have several features requiring multiple dimensions, coordinate dimensioning is sometimes used to clarify the drawing.
- The "Alphabet of Lines" is a standardized system of lines and line symbols developed by the American National Standards Institute (ANSI).
- Drawings include a wealth of information, both in the drawing itself and in the title block, revision block, and other standard features, such as drawing notes.
- Prints can be generated from 2D drawings or from 3D CAD models.
- Working drawings show the machinist exactly what to make and include the specifications for the product.
- Working drawings include a parts list or bill of materials that identify the items needed to create a part, and the number of each item required.
- Working drawings are usually created on standard-size sheets.

Review Questions

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

1. Drawings are used to ______.
   A. show, in multiview, what an object looks like before it is made
   B. identify the standards to be used to make an object
   C. show what to make and the sizes to make it
   D. All of the above.
   E. None of the above.
2. The standardized list of lines and line symbols developed by ANSI to give drawings meaning is called the ______.
3. What is the purpose of a hidden line?
4. What is dual dimensioning?
5. A microinch is ______ of an inch.
6. A micrometer (μm) is also known as a(n) ______.
7. How can the surface roughness of a machined part be checked against specifications on the drawing? How can it be measured electronically?
8. Tolerances are ______.
   A. the different materials that can be used to create an item
   B. acceptable allowances greater than and/or less than a stated dimension size
   C. dimensions
   D. All of the above.
   E. None of the above.
9. When both plus and minus allowances are given with a dimension, the tolerance is called a(n) ______ tolerance.
10. When only a positive (plus) or only a negative (minus) allowance follows a dimension, the tolerance is called a(n) ______ tolerance.
11. Drawings made other than actual size are called ______ drawings.
12. How are most drawings created today?
13. The type of drawing that gives the machinist all of the information needed to make a part is a(n) ______ drawing.
14. What does an assembly drawing show?
15. A subassembly drawing differs from an assembly drawing by ______.
    A. showing only a small portion of the complete object
    B. requiring a larger drawing sheet
    C. showing the object without all needed dimensions
    D. All of the above.
    E. None of the above.
16. What is the purpose of a parts list or bill of materials?
17. Why are standard-size drawing sheets used?

CHAPTER **5**

Measurement

Chapter Outline

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.
- Use the various helper measuring tools found in a machine shop.

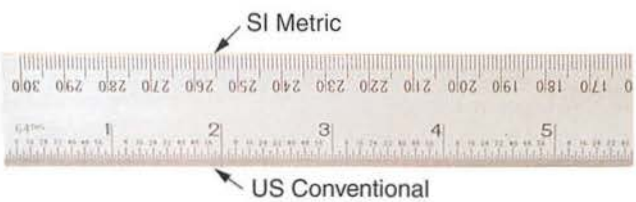
Technical Terms

dial indicator	micrometer caliper
gage blocks	steel rule
gaging	universal bevel protractor
graduations	vernier caliper
metrology	vernier scale

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.

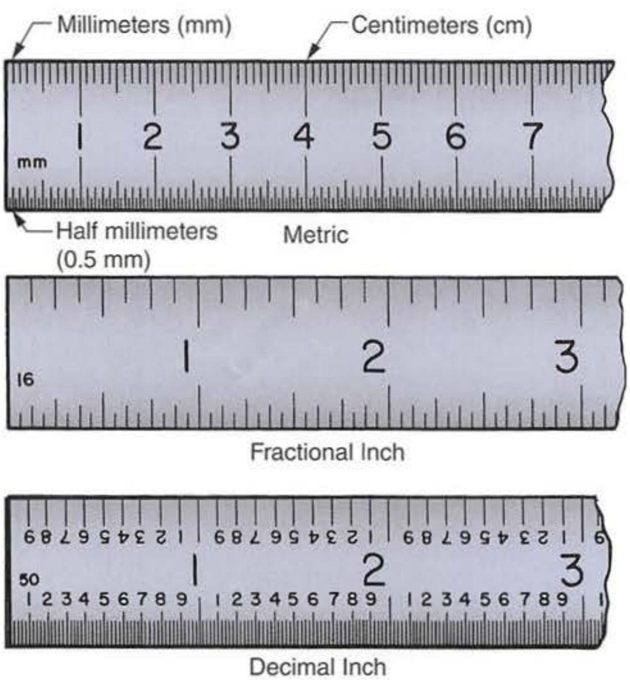
Industry uses both US Conventional (inch and foot) and SI Metric (millimeter and meter) units of measure. Recall that a microinch is one-millionth of an inch (0.000001″), and a micrometer or micron is one-millionth of a meter (0.000001 m).

Many of the familiar measuring tools are available with scales graduated both in US Conventional units and in metric units, **Figure 5-1**. A metric rule is compared with conventional fractional and decimal rules in **Figure 5-2**.



Goodheart-Willcox Publisher

Figure 5-1. This rule can be used to make measurements in both US Conventional and SI Metric units.



Goodheart-Willcox Publisher

Figure 5-2. Compare the metric (millimeter-graduated) rule with the rules graduated in fractional and decimal inch units.

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 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 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.

5.1 The Rule

The ***steel rule***, often incorrectly called a *scale*, is the simplest of the measuring tools found in the shop. **Figure 5-2** shows the three basic types of rule graduations. A few of the many rule styles are shown in **Figure 5-3**.

5.1.1 Reading a US Conventional Rule

A careful study of the enlarged rule sections in **Figure 5-4** shows the fractional divisions of the inch from 1/8 to 1/64. The lines representing the divisions are called ***graduations***. On many inch-based rules, every fourth graduation is numbered on the 1/32 edge, and every eighth graduation on the 1/64 edge. 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.

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 decimal inches and have graduations in 10ths, 20ths, 50ths, and 100ths.

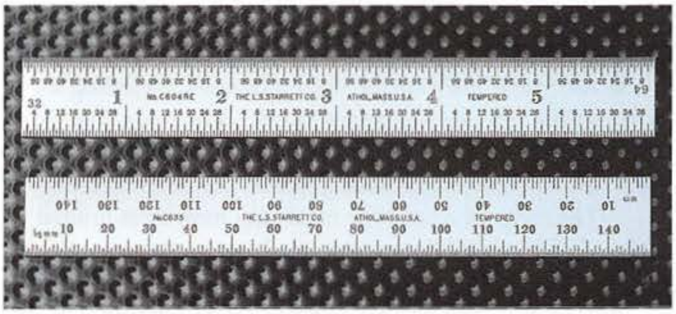
5.1.2 Reading a Metric Rule

Metric rules are typically divided into millimeter or half-millimeter graduations. They are numbered every 10 mm. See **Figure 5-5**. The measurement is determined by counting the number of millimeters.

5.1.3 Care of the Rule

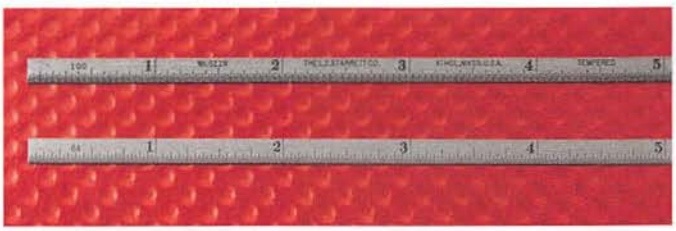
The steel rule is precision-made and, like all tools, its accuracy depends on the care it receives. Here are a few suggestions:

- Use the rule for measurements only. Do not adjust screws or open paint cans with it. Be careful not to bend 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 preventive.
- Clean the rule lightly with steel wool to keep the graduations legible.

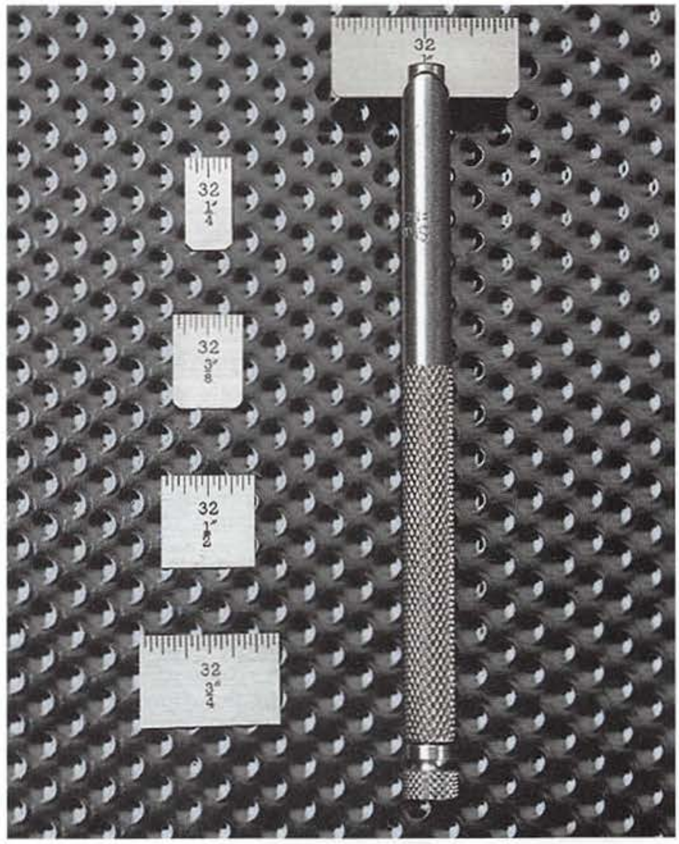


6″ Steel Rule

Rule with Adjustable Hook



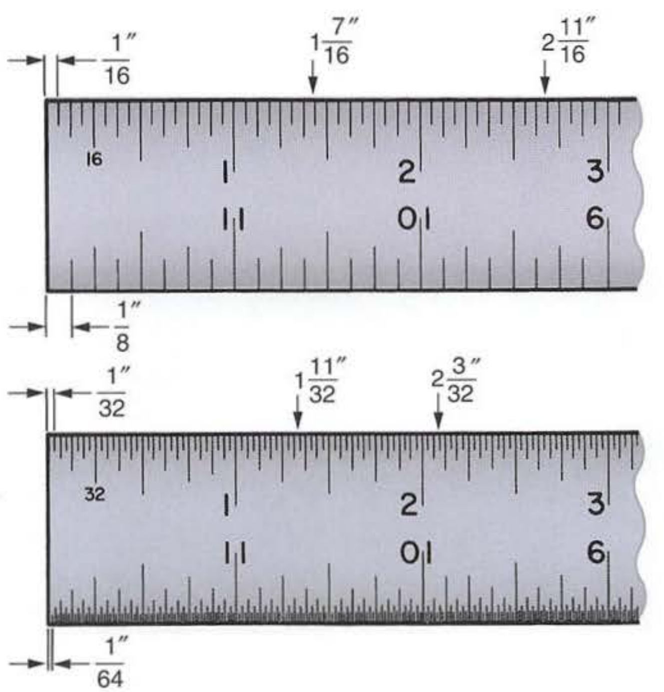
Narrow Rule



Small Rules with Holder

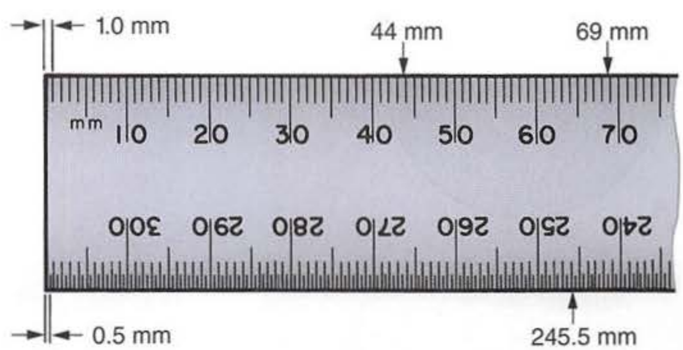
L. S. Starrett Co.

Figure 5-3. Many different types of rules are used to make measuring faster and more accurate.



Goodheart-Willcox Publisher

Figure 5-4. These are the fractional graduations found on a standard inch-based rule. Measurements are taken by counting the number of graduations.



Goodheart-Willcox Publisher

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

- 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.

> **SAFETY NOTE**
>
> 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.

5.2 The Micrometer Caliper

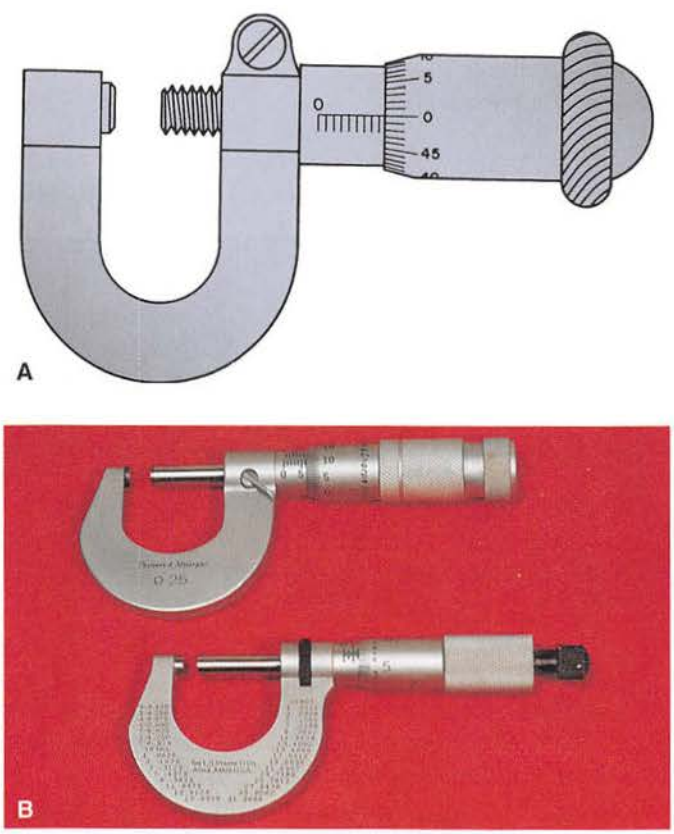
A Frenchman, Jean Palmer, devised and patented a measuring tool that made use of a screw thread, making it possible to read measurements quickly and accurately without calculations. It incorporated a series of engraved lines on the sleeve and around the thimble. The device, called *Systeme Palmer*, is the basis for the modern micrometer caliper, **Figure 5-6**.

The ***micrometer caliper*** 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.

5.2.1 Types of Micrometers

Several types of micrometers are available. Many have a digital display that makes measuring easier. Some of the most popular models include:

- An outside micrometer measures external diameters and thickness, **Figure 5-7**.
- 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 5-8**.
- A micrometer depth gage measures the depths of holes, slots, and projections. See **Figure 5-9**. The measuring range can be increased by changing to longer spindles. Measurements are read from right to left.

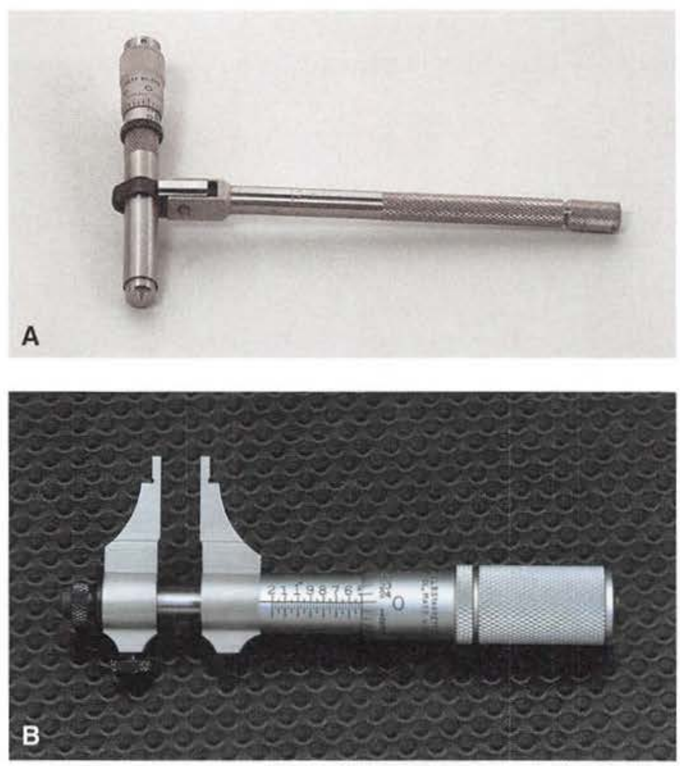


Goodheart-Willcox Publisher

Figure 5-6. 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.

Mitutoyo/MTI Corp.

Figure 5-7. This digital outside micrometer can be used to measure in both US Conventional and SI Metric units.



L. S. Starrett Co.

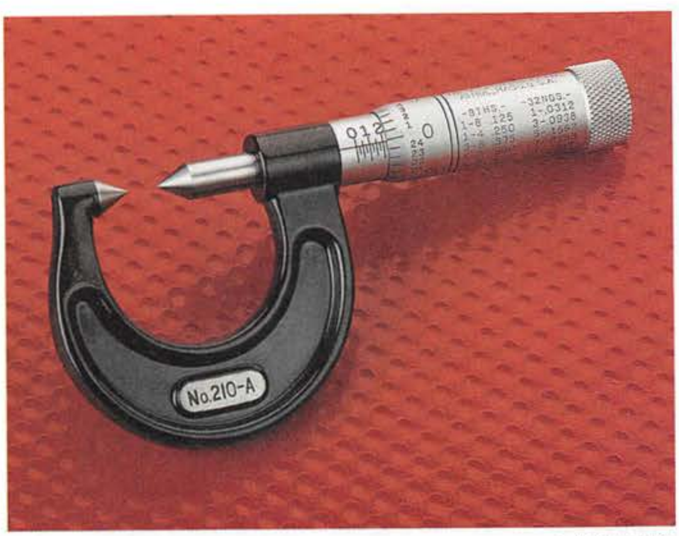
Figure 5-8. Inside micrometers. A—A conventional inside micrometer. B—The caliper jaws on this inside micrometer allow quick and accurate measurements. The divisions on the sleeve are numbered in the reverse order of a conventional outside micrometer.

- A screw thread micrometer has a pointed spindle and a double-V anvil shaped to contact the screw thread, **Figure 5-10**. 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.
- A chamfer micrometer accurately measures countersunk holes and other chamfered measurements. Because fastener 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.

Goodheart-Willcox Publisher

Figure 5-9. A standard micrometer depth gage.



L. S. Starrett Co.

Figure 5-10. This screw thread micrometer can measure threads as wide as 7/8″.

5.2.2 Reading a 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 5-11**.

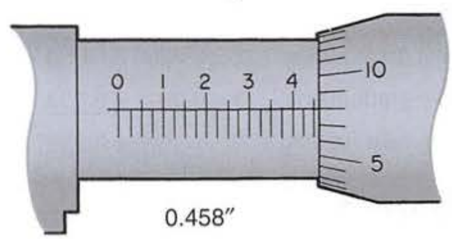
Reading an Inch-Based Micrometer

The threaded section has 40 threads per inch; therefore, each revolution of the thimble moves the spindle 1/40″ (0.025″). 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″, and so on.

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; on others, every fifth division is numbered.

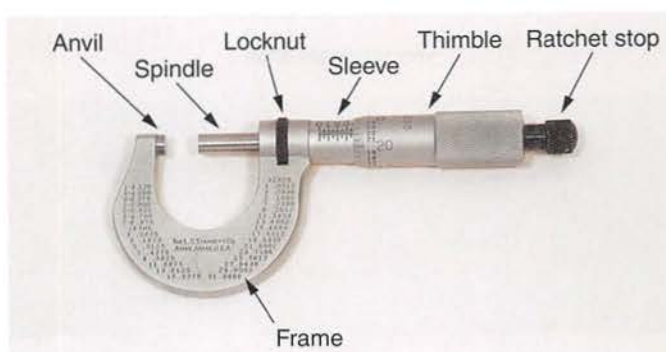
To read a micrometer, record the highest number on the sleeve, where 1 = 0.100, 2 = 0.200, and so on. To this number, add the number of vertical lines visible between the number and thimble edge (1 = 0.025, 2 = 0.050, and so on). To this total, add the number of thousandths indicated by the line that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

Example 1



Add the readings from the sleeve and the thimble:

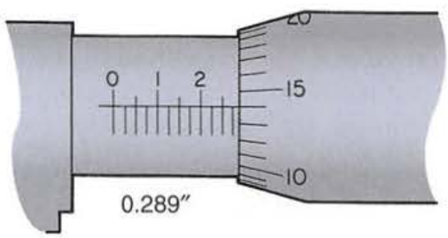
4 large graduations:	4 × 0.100	= 0.400
2 small graduations:	2 × 0.025	= 0.050
8 thimble graduations:	8 × 0.001	= 0.008
Total mike reading		= 0.458″



Goodheart-Willcox Publisher

Figure 5-11. Basic parts of a micrometer caliper.

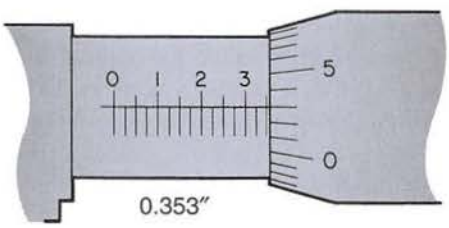
Example 2



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	= 0.003
Total mike reading		= 0.353″

Reading an Inch-Based Vernier Micrometer

A vernier micrometer caliper is used when it is necessary to measure more precisely than 0.001″. This micrometer has a third scale around the sleeve that supplies the 1/10,000″ (0.0001″) reading. See **Figure 5-12**.

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 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 it is line 2, add 0.0002 to the reading, and so forth. See **Figure 5-13**.

Reading a Metric Micrometer

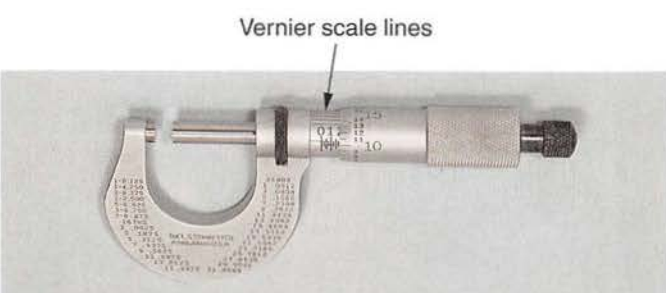
The metric micrometer is read as shown in **Figure 5-14**. If you are able to read the conventional inch-based micrometer, reading the metric tool will offer no difficulties. The procedure is the same.

Reading a Metric Vernier Micrometer

Metric vernier micrometers are read in the same way as inch-based vernier micrometers. Obtain the basic reading, then use the vernier scale on the sleeve to obtain an additional reading of two-thousandths of a millimeter, **Figure 5-15**.

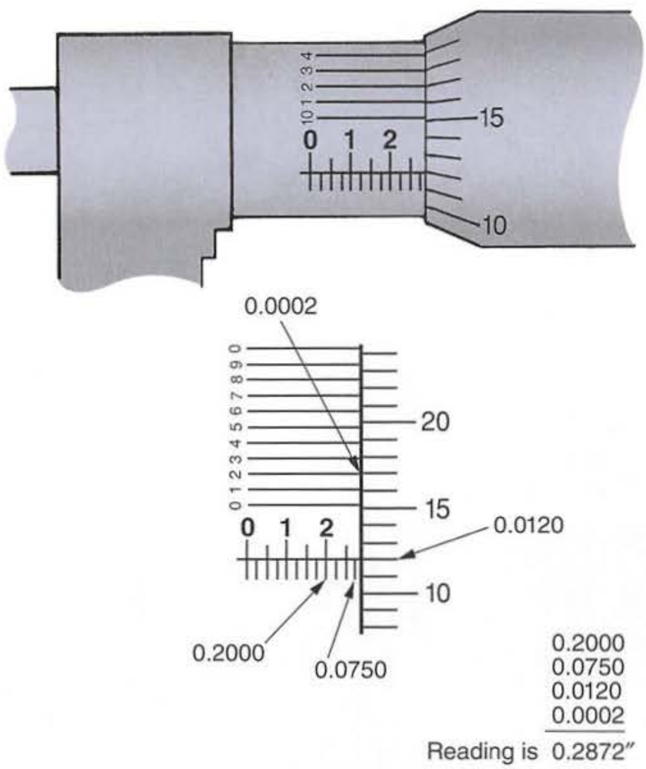
5.2.3 Using an Outside Micrometer

The proper way to hold an outside micrometer when making a measurement is shown in **Figure 5-16**. The work is placed into position, and the thimble is rotated until the part is clamped lightly between the anvil and spindle. Guard against



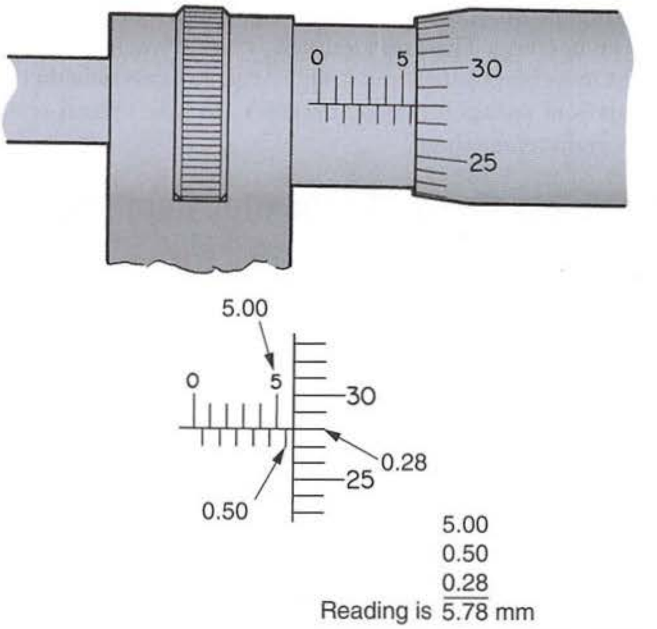
Goodheart-Willcox Publisher

Figure 5-12. A vernier micrometer caliper includes a vernier scale on the sleeve.



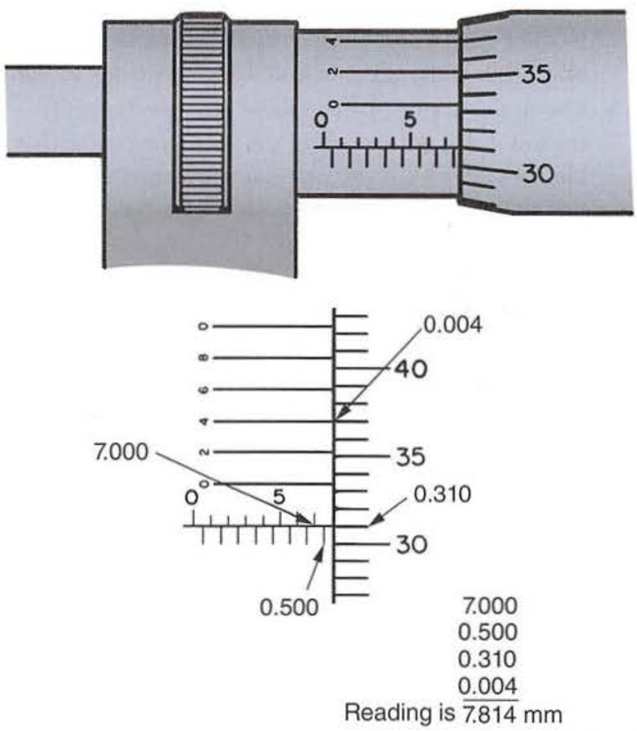
Goodheart-Willcox Publisher

Figure 5-13. 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.



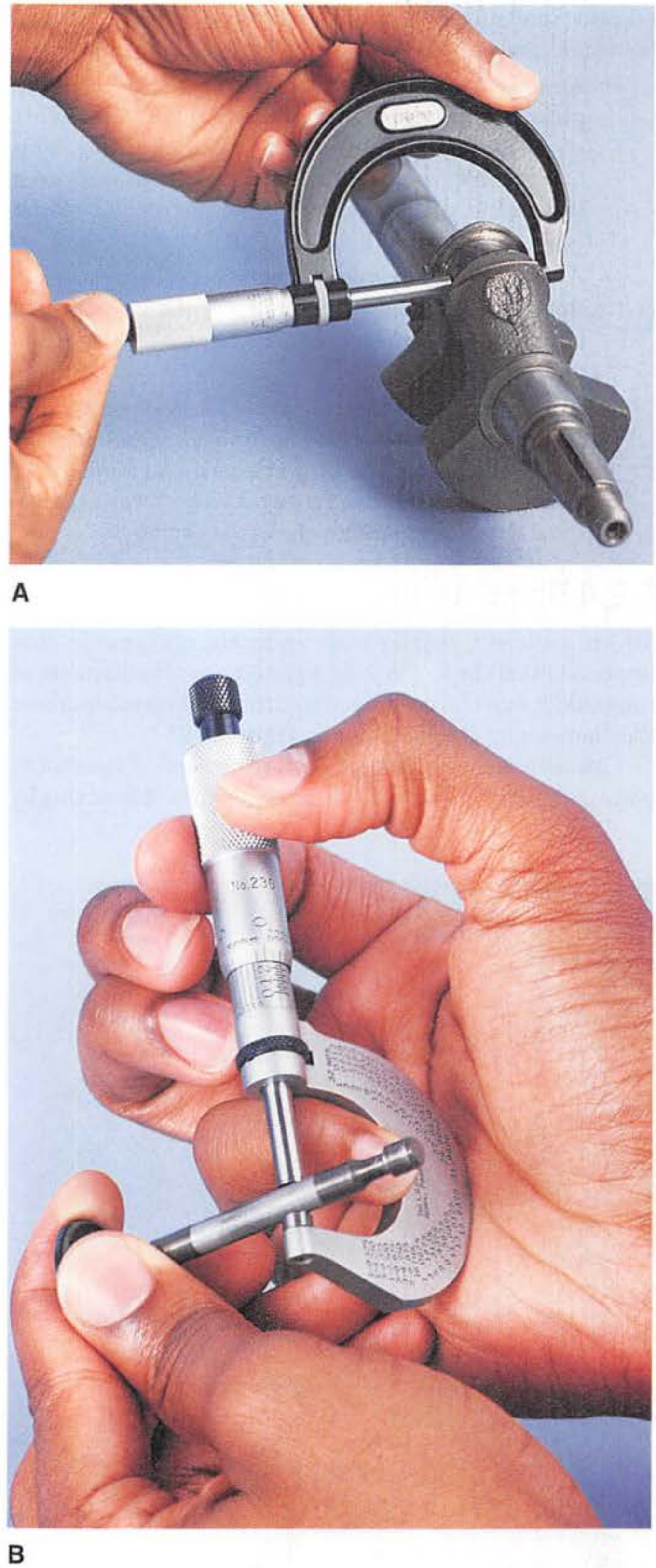
Goodheart-Willcox Publisher

Figure 5-14. 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.



Goodheart-Willcox Publisher

Figure 5-15. Reading a metric 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).



Goodheart-Willcox Publisher

Figure 5-16. 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.

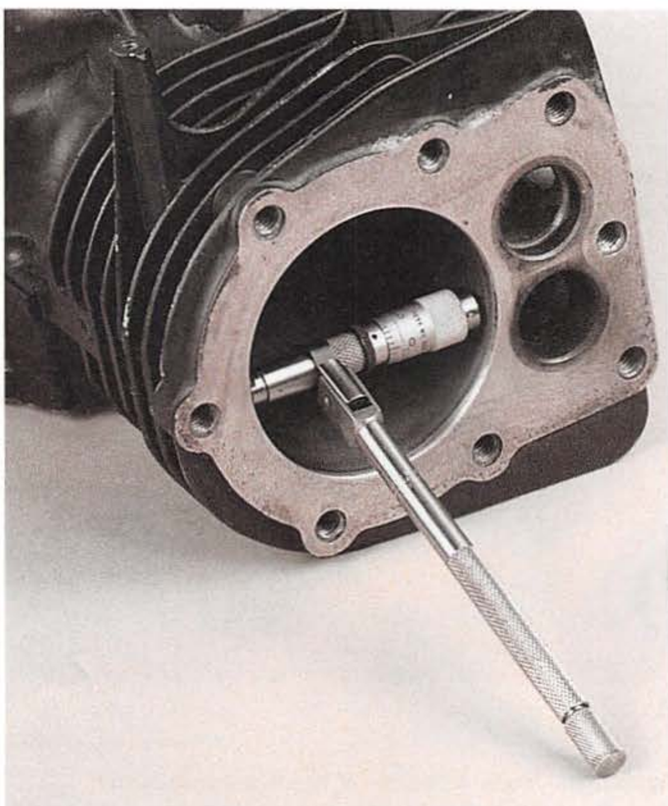
excessive pressure, which will cause an erroneous reading. Some micrometers have features to help regulate pressure:

- 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 5-11**.
- 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.
- A locknut is used when several identical parts are to be gaged. Refer again to **Figure 5-11**. 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.

5.2.4 Using 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 5-17**.

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.



Goodheart-Willcox Publisher

Figure 5-17. Using an inside micrometer. Extension rods can be added to increase the tool's measuring range.

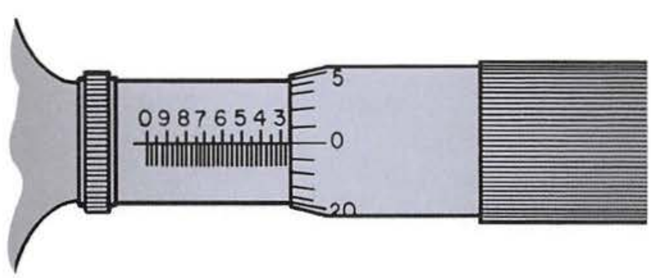
5.2.5 Reading a Micrometer Depth Gage

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 5-18**. The graduations under the thimble must be read, rather than those that are exposed.

5.2.6 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 5-19**.



Goodheart-Willcox Publisher

Figure 5-18. 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".

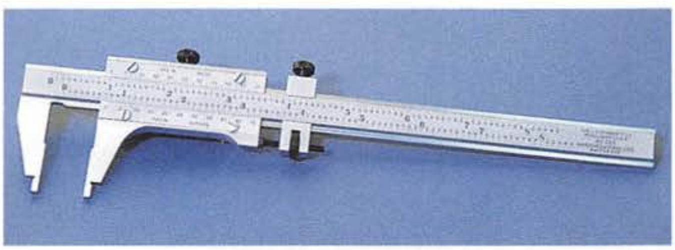
Goodheart-Willcox Publisher

Figure 5-19. 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.

- 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.

5.3 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 5-20**.



A

B

Goodheart-Willcox Publisher

Figure 5-20. 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 in US Conventional lengths of 6″, 12″, 24″, 36″, and 48″. 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 5-21**.

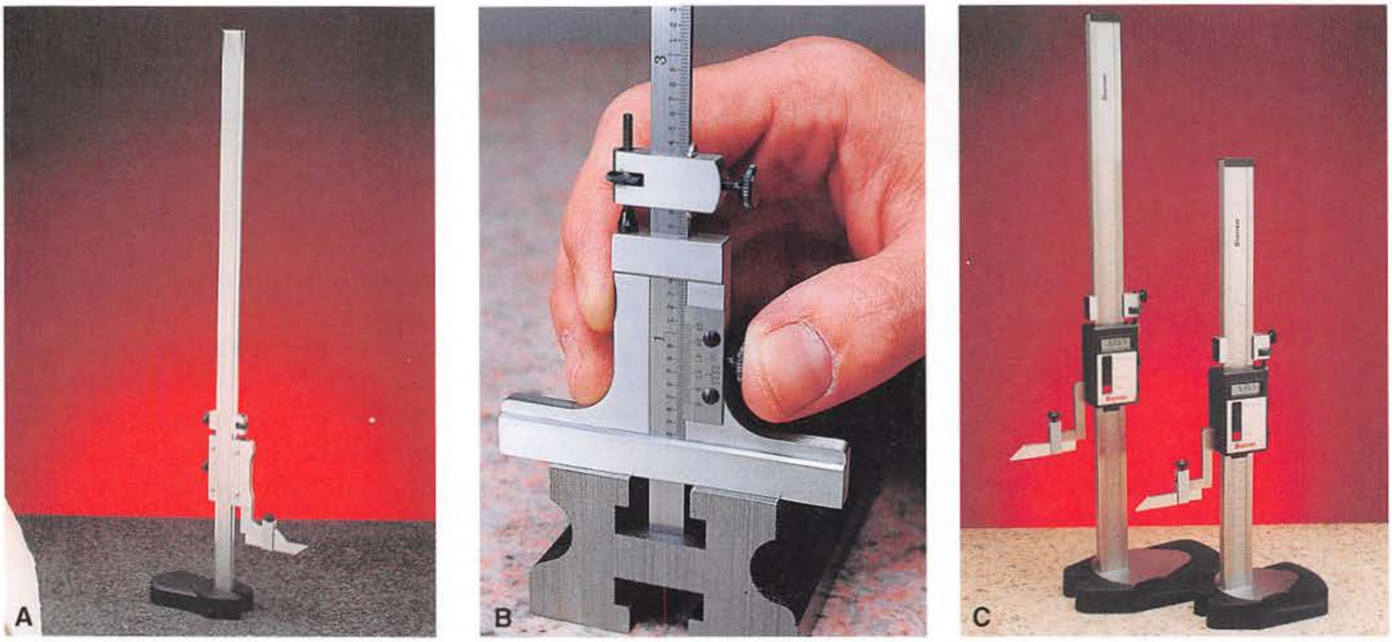
The following measuring instruments may include a vernier scale:

- Height and depth gages are used for layout work and to inspect the locations of features. See **Figure 5-22**.
- Gear tooth calipers are used to measure gear teeth and threading tools, **Figure 5-23**.
- Universal vernier bevel protractors are used for the layout and inspection of angles, **Figure 5-24**.

Vernier measuring tools, with the exception of the vernier bevel protractor, consist of a graduated beam with a 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.

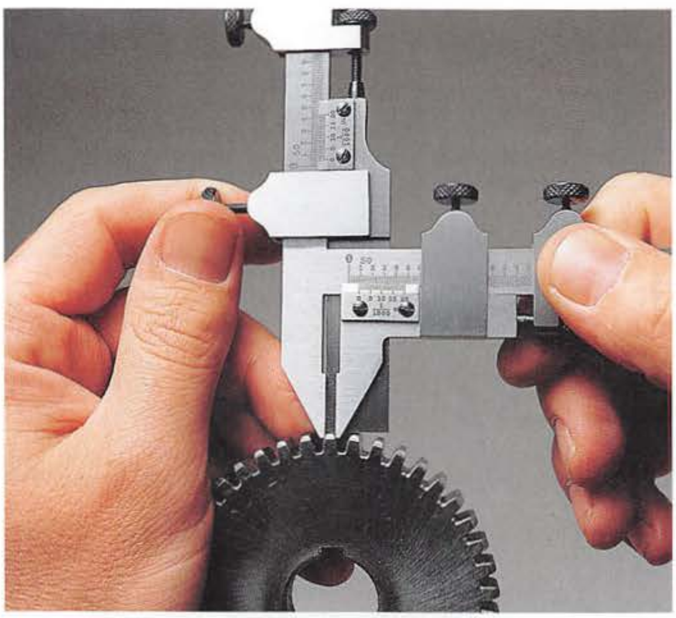
Mahr Federal Inc.

Figure 5-21. Vernier calipers can be used to make both internal and external measurements.



L. S. Starrett Co.

Figure 5-22. 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 5-23. Gear tooth vernier calipers are used to measure gear teeth, form tools, and threaded tools.

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.

Mahr Federal Inc.

Figure 5-24. A digital universal vernier bevel protractor is used to accurately measure angles.

5.3.1 Reading an Inch-Based Vernier Scale

Inch-based vernier scales are available with either 25-division or 50-division vernier plates. Both plates can be read to 0.001″.

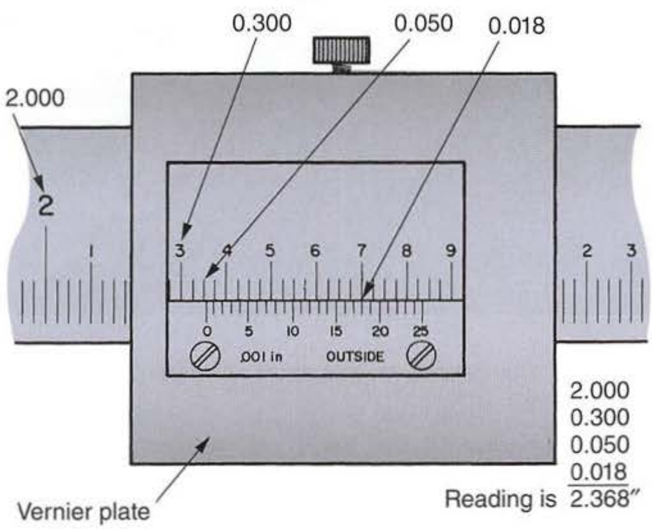
On scales that have a 25-division vernier plate, every inch section on the beam is graduated into 40 equal parts. Each graduation is 1/40″ (0.025″). Every fourth division, representing 0.100″, is numbered.

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, tenths, and fortieths (0.025, 0.050, or 0.075) are between the "0" on the vernier scale and the "0" line on the beam, and add them together. 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 corresponds exactly with a line on the beam. Only one line will line up exactly. Add this to the previous total for the reading.

Example 4



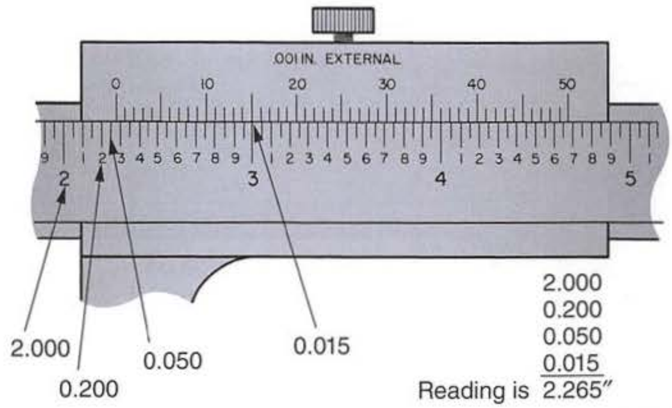
The "0" line on the vernier plate is:

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, and twentieths (0.050) there are between the "0" line on the beam, and the "0" line on the vernier plate, and 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 previous total.

Example 5

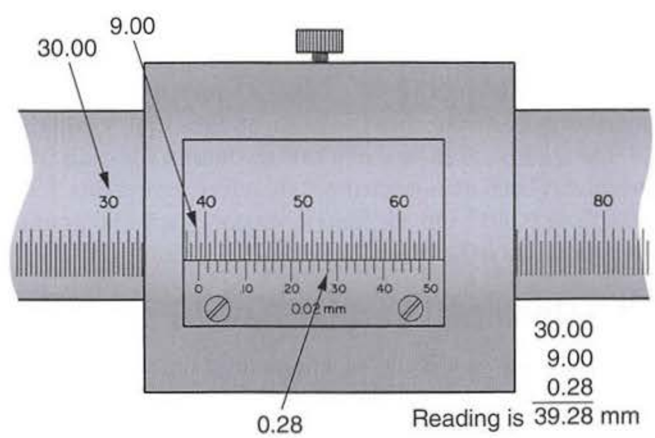


The "0" line on the vernier plate is:

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″

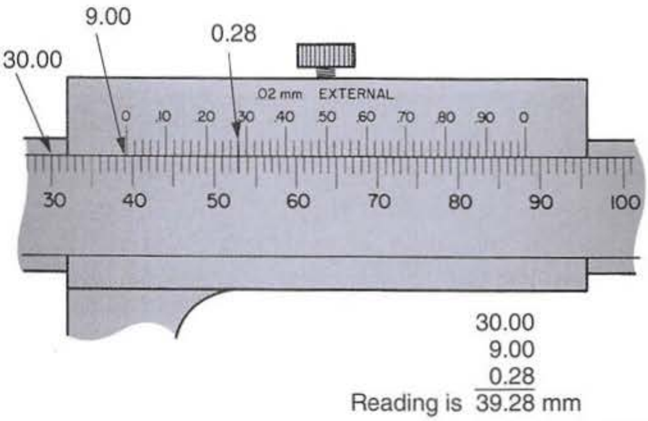
5.3.2 Reading a Metric Vernier Scale

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 5-25**, and a 50-division scale is described in **Figure 5-26**.



Goodheart-Willcox Publisher

Figure 5-25. How to read a 25-division metric vernier scale. Readings on the scale are obtained in units of two hundredths of a millimeter (0.02 mm).



Goodheart-Willcox Publisher

Figure 5-26. How to read a 50-division metric vernier scale. Each division equals two hundredths of a millimeter (0.02 mm).

5.3.3 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.

5.3.4 Using a Dial Caliper

A dial caliper is a direct-reading instrument that resembles a vernier caliper. See **Figure 5-27**. It can be used to make outside, inside, and depth measurements (with the addition of a depth attachment). A lock permits the tool to be used for repetitive measurements.

The beam is graduated in 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.

5.3.5 Universal Vernier Bevel Protractor

A quick review of the circles, angles, and units of measurement associated with a universal vernier bevel protractor will help you understand how to read this instrument:

- **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 (for example: 30°15′).

L. S. Starrett Co.

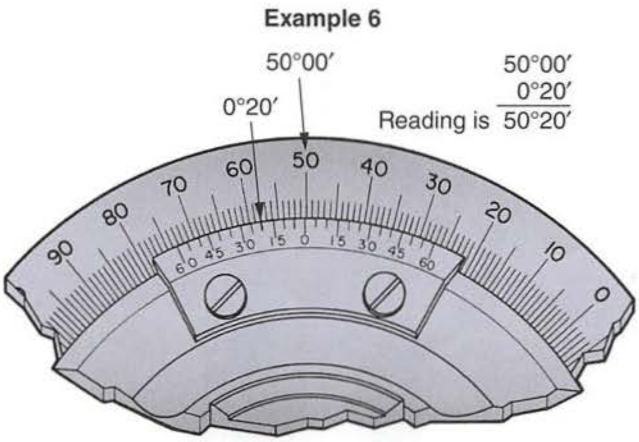
Figure 5-27. Dial calipers provide direct readings of measurements.

- **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 10° division is numbered, and every 5° is indicated by a fine line longer than those on either side. The vernier scale is divided into 12 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′.

5.3.6 Care of Vernier Tools

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

- 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, gage block, or ground parallel. Return the tool to the manufacturer for adjustments and repairs when needed.
- Lay vernier height gages on their side when not in use to prevent them from being knocked over and damaged.

5.4 Gages

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.

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. Gages provide a method of checking your work.

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.

5.4.1 Plug Gage

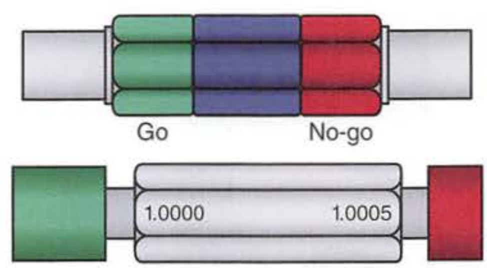
Plug gages are used to check whether hole diameters are within specified tolerances. The double-end cylindrical plug gage has two gaging members known as *go* and *no-go plugs*, **Figure 5-28**. The go plug should enter the hole with little or no interference. The no-go plug should not fit.

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 5-29**.

5.4.2 Ring Gage

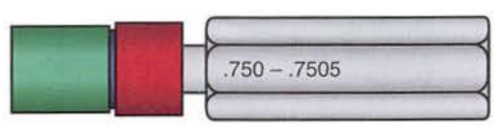
External diameters are checked with ring gages, **Figure 5-30**. The go and no-go ring gages are separate units, and can be distinguished from each other by a groove cut on the knurled outer surface of the no-go gage.

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.



Goodheart-Willcox Publisher

Figure 5-28. A double-end cylindrical plug gage.



Goodheart-Willcox Publisher

Figure 5-29. A step plug gage can check for oversize and undersize in a single test.

Kimtaro/Shutterstock.com

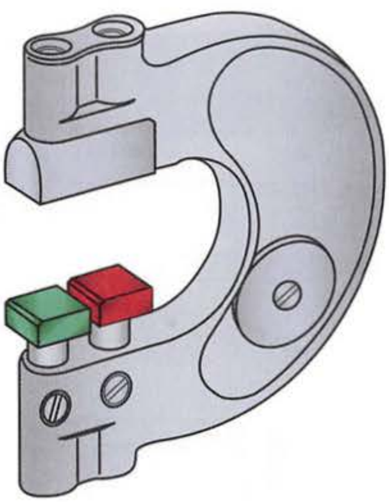
Figure 5-30. A ring gage is used to check external diameters.

5.4.3 Snap Gage

A snap gage serves the same purpose as a ring gage, except that 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 5-31**.
- A nonadjustable snap gage is made for one specific size.
- 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 non-adjustable indicating snap gages are available. See **Figure 5-32**.

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



Goodheart-Willcox Publisher

Figure 5-31. An adjustable snap gage.

Mahr Federal Inc.

Figure 5-32. A dial indicator snap gage.

5.4.4 Thread Gages

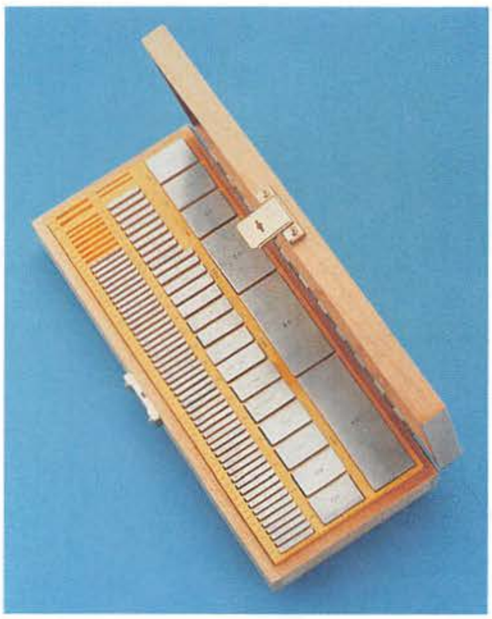
Several types of gages are used to check screw thread fits and tolerances. These gages are similar to the gages already discussed:

- A thread plug gage is similar to a normal plug gage except that the plugs are threaded. Threaded plug gages are used to check internal threads. The go plug should enter the threaded hole, while the no-go plug should not.
- A thread ring gage is similar to a ring gage with internal threads. Thread ring gages are used to check external threads. The no-go ring gage features a groove cut into the knurled surface to distinguish it from the go ring gage.
- A thread snap gage features thread-shaped anvils or rollers. Thread snap gages operate much like normal snap gages. Dial thread snap gages are also available.

5.4.5 Gage Blocks

Gage blocks, commonly known as *Jo-blocks* or *Johansson blocks*, are precise measuring standards made of steel. Gage blocks can be purchased in various sets ranging from a few commonly used block sizes to more complete sets. See **Figure 5-33**.

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 5-34**.



Federal Products Co.

Figure 5-33. A typical set of gage blocks.

Federal Accuracy Grades			
Grade		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)

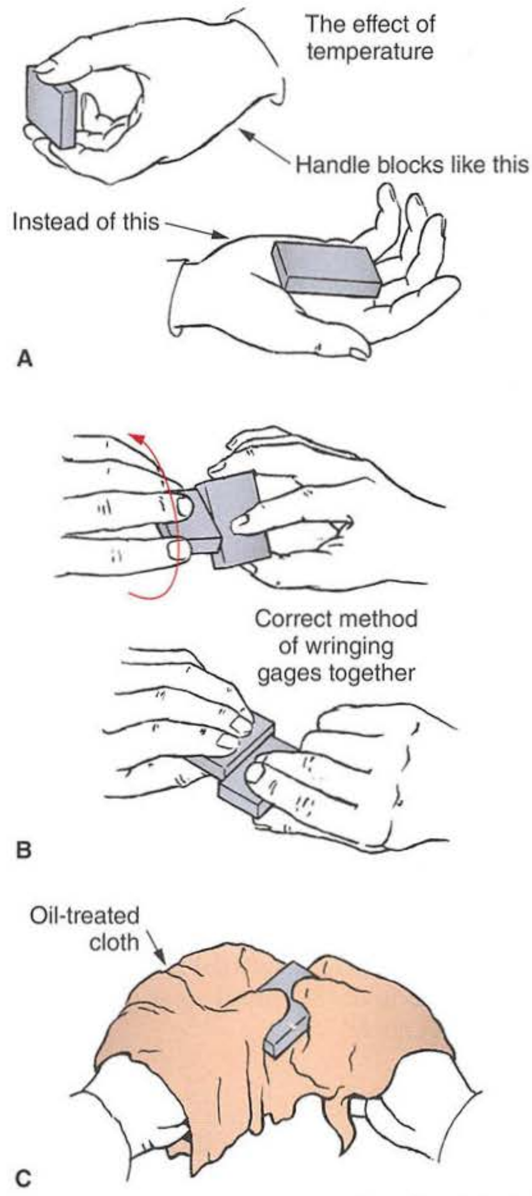
One inch = 25.4 millimeters exactly

Goodheart-Willcox Publisher

Figure 5-34. Federal Accuracy Grades for gage blocks.

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 5-35A**.
- When wringing gage blocks together to build up to desired size, wipe the blocks and then carefully slide them together. See **Figure 5-35B**. They should adhere to each other strongly. Separate the blocks when



Webber Gage Div., L. S. Starrett Co.

Figure 5-35. 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.

you are finished. Leaving gage blocks together for extended periods may cause the contacting surfaces to corrode.

- 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 5-35C**.

5.4.6 Other Gaging Tools

Industry makes wide use of other types of gaging tools. Most of these tools are used for special purposes.

Air Gage

An air gage uses air pressure to measure hole sizes and hard-to-reach shaft diameters, **Figure 5-36**. This type of gage is especially helpful when measuring deep internal bores. The basic operation of an air gage is illustrated in **Figure 5-37**.

There is no actual contact between the measuring gage and the 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 airstream, or by a manometer (U-shaped tube in which the height of fluid in the tube indicates pressure).

Electronic Gage

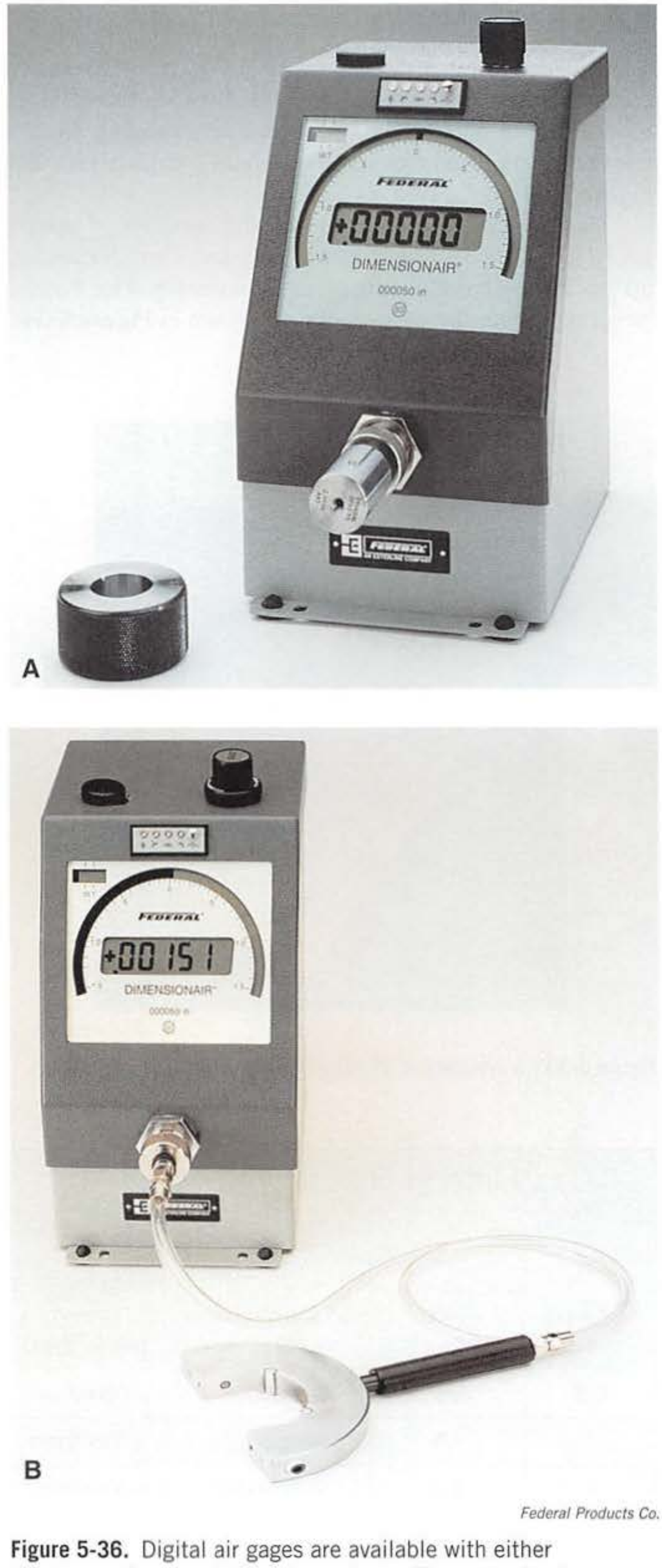
An electronic gage, **Figure 5-38**, is another type of gaging tool used to make extremely precise measurements. Electronic gages are comparison gages: they compare the size of the work to a reference size. Some are calibrated using master gage blocks and others use replaceable gaging probes. These instruments measure in both US Conventional and SI Metric units.

Laser Gaging

A laser is a device that produces a very narrow beam of extremely intense light. *Laser* is an acronym for "light amplification by stimulated emission of radiation." Lasers are used in communication, medical, and industrial applications. When employed for industrial inspection purposes, it can check the accuracy of critical areas in machined parts quickly and accurately.

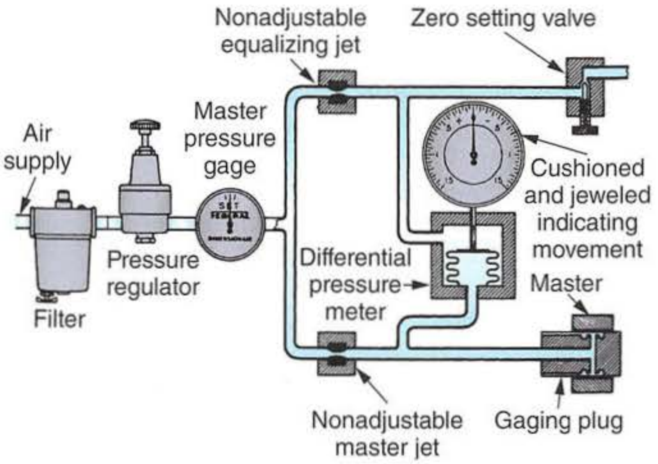
Optical Comparator

The optical comparator uses magnification to inspect parts, **Figure 5-39**. An enlarged image of the part is projected on a screen for inspection. The part image is superimposed on 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.



Federal Products Co.

Figure 5-36. 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 that is used to check hard-to-reach diameters, such as crankshaft journals.



Goodheart-Willcox Publisher

Figure 5-37. This diagram illustrates the operation of an air gage.

Sunnen Products Company

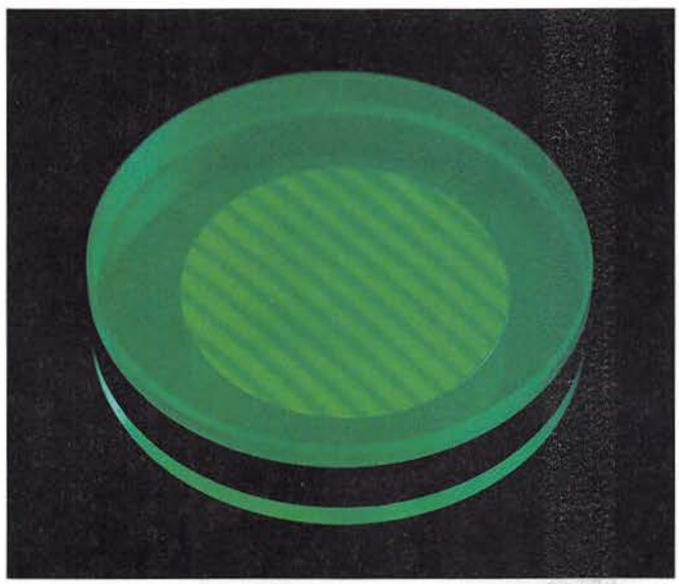
Figure 5-38. This electronic bore gaging system can deliver electronic resolution as fine as 0.00001″ (0.0002 mm). Using replaceable gaging probes, the self-contained unit measures diameters ranging from 0.370″ to 2.900″. It also measures in millimeters. It can be linked to a computer for statistical process control (SPC) data collection.

BIG Kaiser Precision Tooling Inc.

Figure 5-39. A technician uses an optical comparator to quickly and efficiently inspect a microdrill.

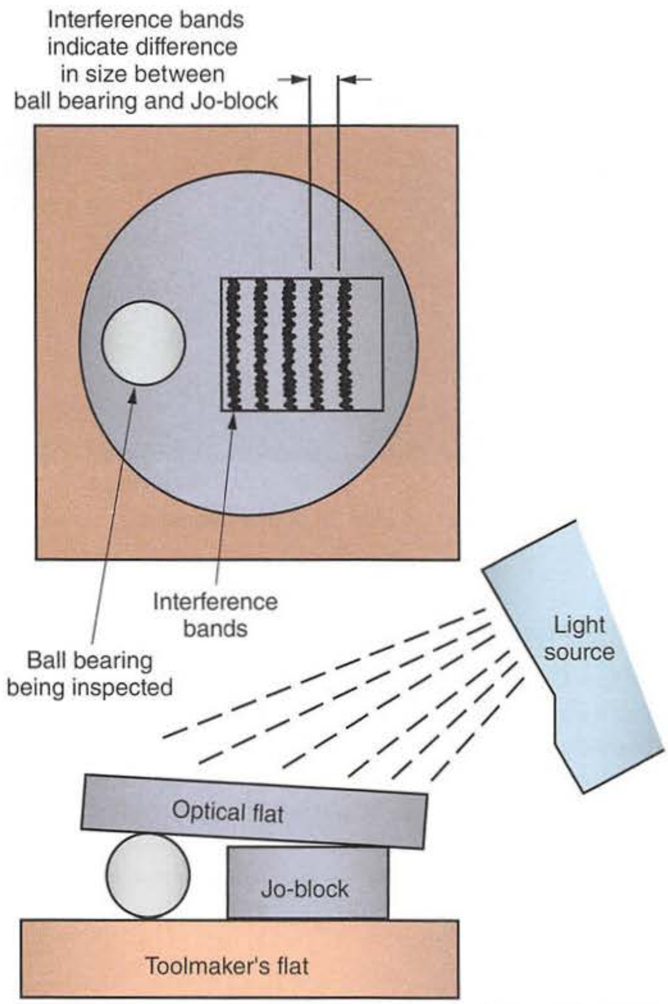
Optical Flats

Optical flats are precise measuring instruments that use light waves as a measuring standard, **Figure 5-40**. 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 5-41**. The shape of these bands indicate to the inspector the accuracy of the part. See **Figure 5-42**.



L. S. Starrett Co.

Figure 5-40. Optical flats are used for precision flatness, parallelism, size, and surface variations.



Goodheart-Willcox Publisher

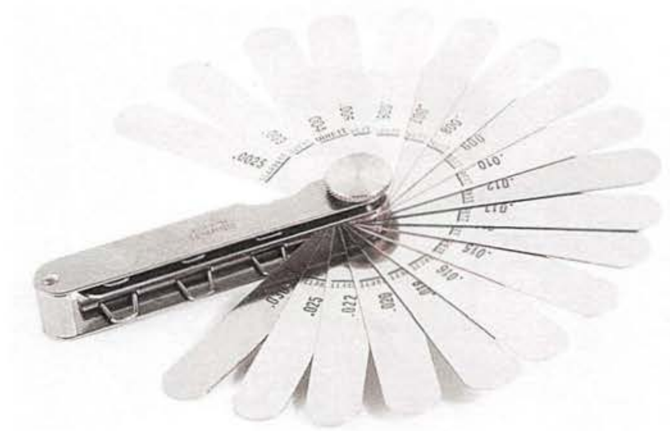
Figure 5-41. Optical flat setup. The optical flat is placed on top of the work and the light is positioned above the flat.

Thickness (Feeler) Gage

Thickness gages, also called *feeler gages*, are pieces or leaves of metal manufactured to precise thickness, **Figure 5-43**. Thickness gages are made of tempered steel and are usually 1/2″ (12.7 mm) wide. They are ideal for measuring narrow slots, setting small gaps and clearances, determining fit between mating surfaces, and checking flatness of parts in straightening operations. See **Figure 5-44**.

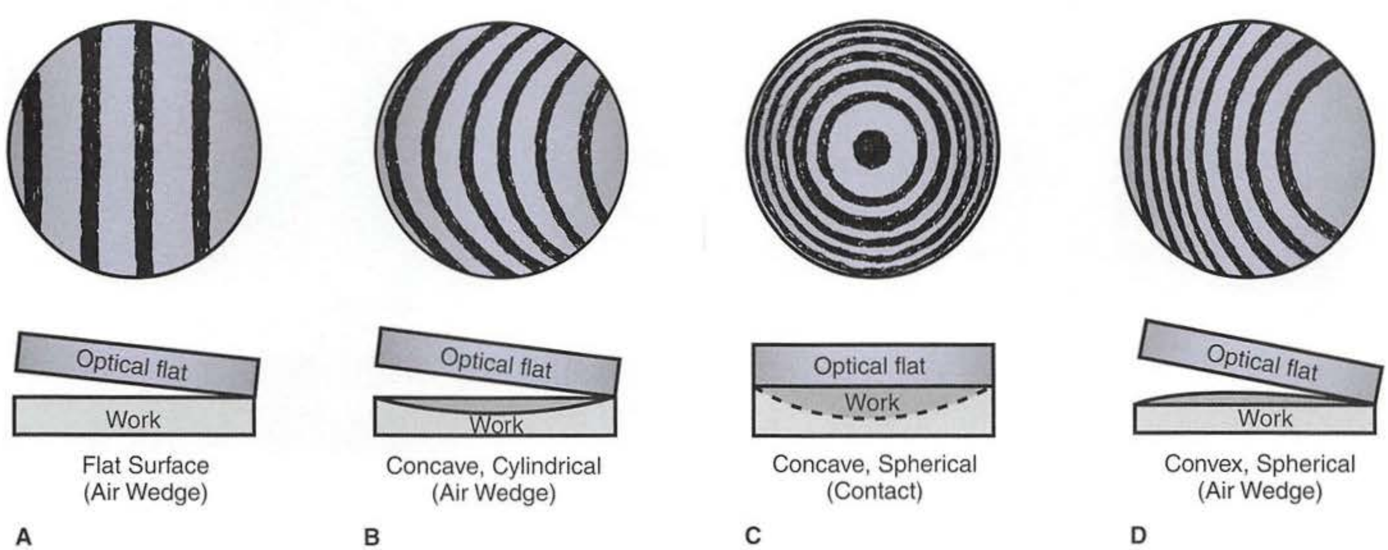
Screw Pitch Gage

Screw pitch gages are used to determine the pitch (number of threads per inch) on a screw, **Figure 5-45**. Each blade is stamped with the pitch. Screw pitch gages are available in US Conventional and SI Metric thread sizes.



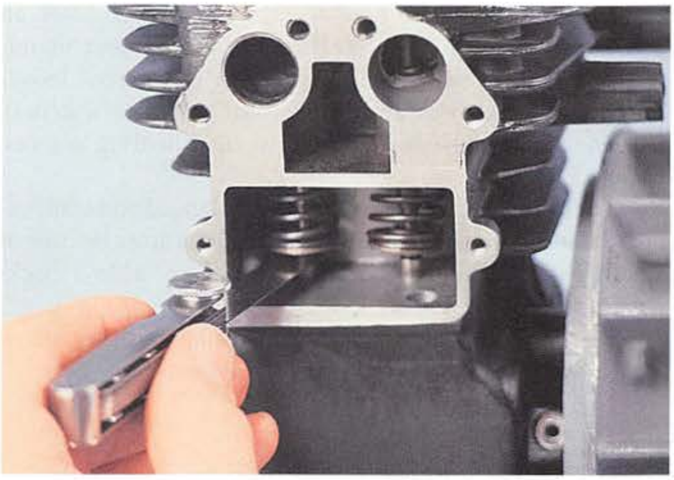
Goodheart-Willcox Publisher

Figure 5-43. Thickness or feeler gages.



Goodheart-Willcox Publisher

Figure 5-42. Interference band patterns indicate surface flatness and variations.



Goodheart-Willcox Publisher

Figure 5-44. A thickness gage is used to check part clearance.

L. S. Starrett Co.

Figure 5-45. Screw pitch gages are made for both inch-based and metric threads.

Fillet and Radius Gage

The thin steel blades of fillet and radius gages, **Figure 5-46**, 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 5-47**. In a set of fillet and radius gages, gage radius increases in 1/64″ (0.5 mm) increments.

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.

CAREER CONNECTION

Civil Engineer

What does a civil engineer do?

Civil engineers design, build, and maintain structures and systems for the private and public sectors. These works may include roads, bridges, buildings, transportation hubs (such as airports or harbors), water supply and sewerage systems, and other infrastructures. Because civil engineering projects are complex, these engineers often specialize in one area of civil engineering, such as construction, transportation, geotechnical, or structural engineering.

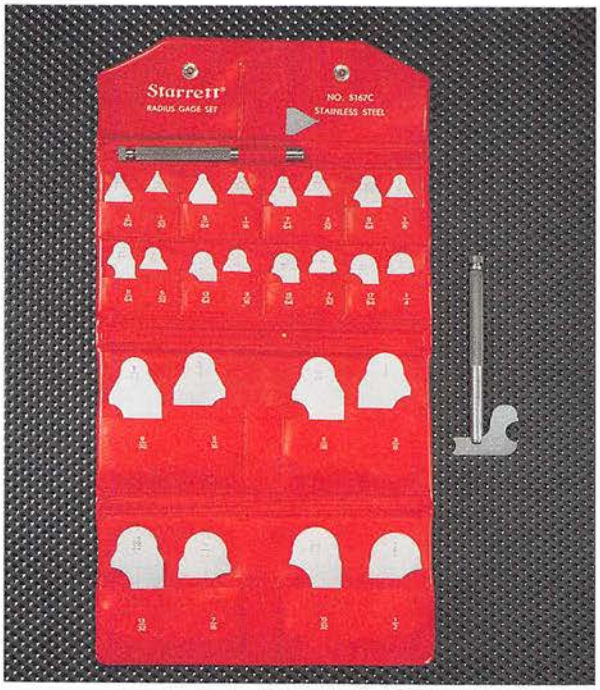
What education and skills are needed to be a civil engineer?

Prospective students earn a bachelor's degree in civil engineering or one of its specialties through an ABET-accredited program. Coursework includes statistics, mathematics, engineering mechanics, and other topics that vary by specialty. Professional licensure and a postgraduate degree are desirable for promotion. Because of the role civil engineers play in upholding local and federal regulations, different levels of licensure may be needed depending on location.

What is it like to be a civil engineer?

Civil engineers must be willing to travel to as many locations as is required to meet the needs of their projects. In the same day, they may travel from office to jobsite to monitor a project's progress and solve any problems directly.

According to the *Occupational Outlook Handbook*, civil engineers earn an annual median wage of $82,220. Wages range from $78,000 to $91,000 annually. Professionals with the highest median wages often work for federal and local governments.



L. S. Starrett Co.

Figure 5-46. A set of radius and fillet gages.

5.5 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 now commonly used.

Dial indicators are used to center and align work on machine tools, check for eccentricity, and inspect manufactured parts. They are designed with shockproof movements and have jeweled bearings (similar to fine watches). Dial indicators must be mounted to rigid holding devices, **Figure 5-48.**

There are two types of indicators: balanced and continuous. Balanced indicators take measurements on either side of a zero line. Continuous indicators read from "0" in a clockwise direction. See **Figure 5-49.**

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

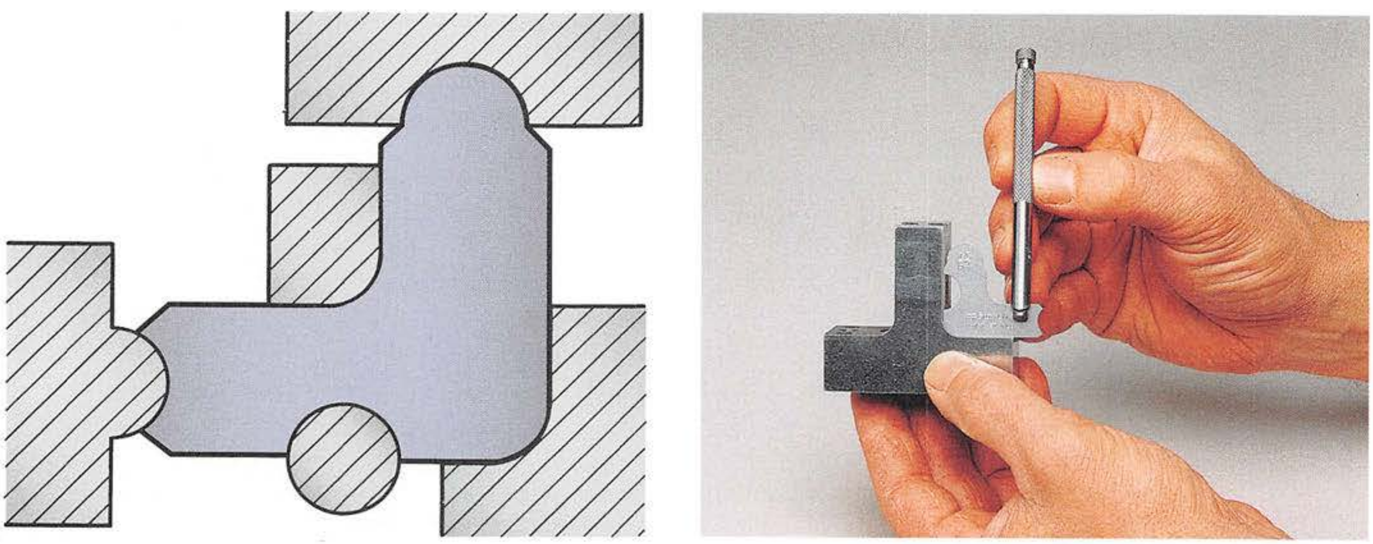
- 1/1000″ (0.001″).
- 1/100 mm (0.01 mm).
- 1/10,000″ (0.0001″).
- 2/1000 mm (0.002 mm).

A digital electronic indicator, **Figure 5-50,** 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.

5.5.1 Using a Dial Indicator

The hand on the dial of a dial indicator is actuated by a sliding plunger. Place the plunger lightly against the work until the hand moves. Turn the dial face until the "0" line coincides with the hand. As you slowly move the work touching the plunger, the indicator hand measures movement.

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



A B

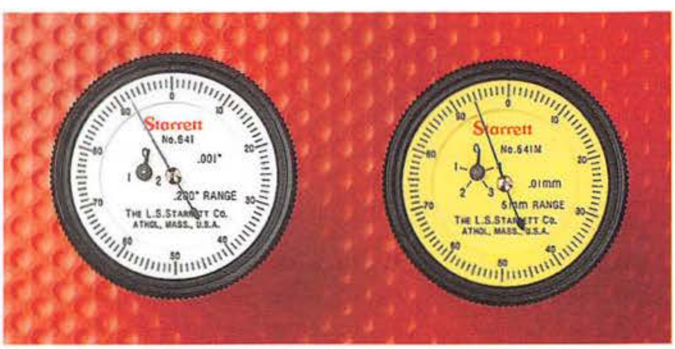
L. S. Starrett Co.

Figure 5-47. Using a radius gage. A—Various ways a radius gage can be used. B—Using a radius gage holder.

Goodheart-Willcox Publisher

Figure 5-48. Mounting this dial indicator on a magnetic base permits it to be attached to any ferrous metal surface. A pushbutton releases the magnet.

A



B

L. S. Starrett Co.

Figure 5-49. The two basic varieties of dial indicators. A— Balanced indicators. B—Continuous indicators.

Federal Products Co.

Figure 5-50. This digital electronic indicator has numeric readouts and a conventional graduated dial.

5.5.2 Using a Dial Test Indicator

Dial test indicators are, in most ways, similar to basic dial calipers. However, test indicators are usually intended for precision work, such as machine setups and surface plate layouts in the tool shop and metrology (gaging) lab. Test indicators can be used with a magnetic base (**Figure 5-48**) when zeroing a vise on a milling machine table or replacing a part in a four-jaw chuck on a lathe. They can also be used with precision height gages to ensure that part-holding devices, such as V-blocks and 1-2-3 blocks, are parallel to the surface plate before performing precision layouts. In addition, dial test indicators can be used to perform precise part sampling measurements in a metrology lab.

5.6 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*.

5.6.1 Calipers

External or internal measurements with an accuracy of up to 1/64″ (0.4 mm) can be made with outside or inside calipers, **Figure 5-51**. 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 an outside 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, **Figure 5-52**. Caliper weight should cause the caliper to slip over the diameter. Hold the caliper next to the rule to make the reading.

An inside caliper is used to make internal measurements when 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, **Figure 5-53**. Hold one leg firmly against the hole wall, and adjust the thumbscrew until the other leg lightly touches the wall exactly opposite the first leg. The legs should drag slightly when moved in and out, or from side to side.

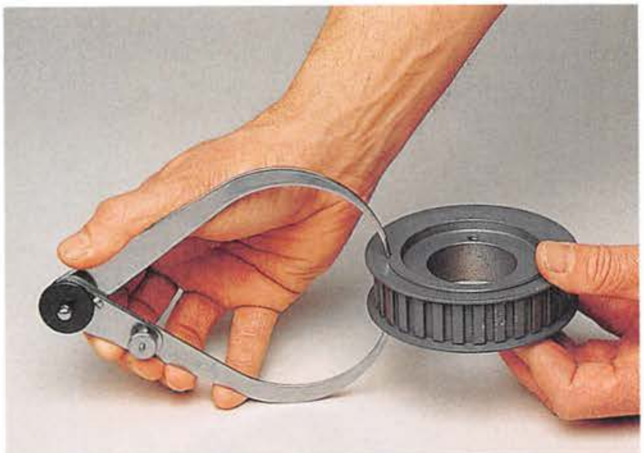
Considerable skill is required to make accurate measurements with a caliper. Much depends on the machinist's sense of touch. With practice, measurements with accuracy of 0.003″ (0.07 mm) can be made. However, a micrometer or vernier caliper is preferred and must be used when greater accuracy is required.

5.6.2 Telescoping Gage

A telescoping gage is used with a micrometer to determine internal dimensions, **Figure 5-54**. Sets of telescoping gages with varying ranges are available, **Figure 5-55**.

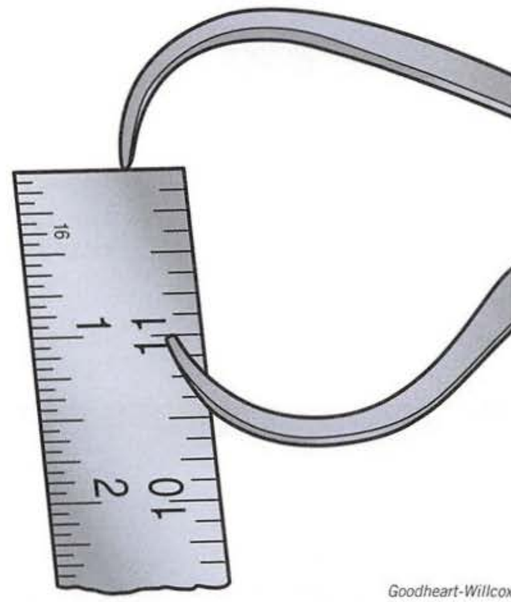
L. S. Starrett Co.

Figure 5-51. Inside and outside calipers.



A

L. S. Starrett Co.



B

Goodheart-Willcox Publisher

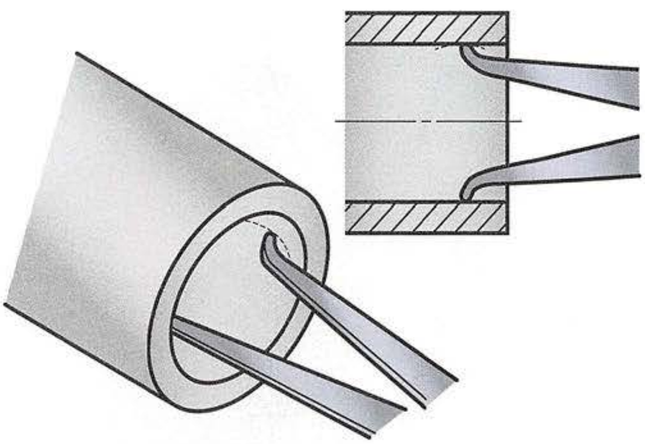
Figure 5-52. Using outside calipers. A—Adjust the calipers to bear lightly on the center line of the stock. B—Use a steel rule to measure the distance between the caliper legs.

To use a telescoping gage, compress the contact legs. The legs collapse within one another under spring tension. Insert the gage into the hole and allow the legs to expand, **Figure 5-56**. After the proper fitting is obtained, lock the contacts into position. Remove the gage from the hole and make the reading with a micrometer, **Figure 5-57**.

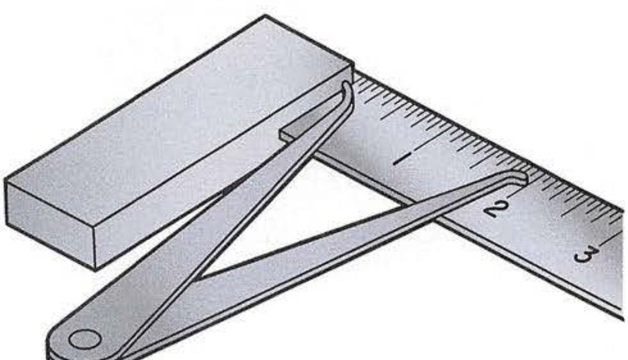
5.6.3 Small Hole Gage

A small hole gage is used to measure openings that are too small for a telescoping gage, **Figure 5-58**. The contacts are designed to allow accurate 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 5-59**.

L. S. Starrett Co.



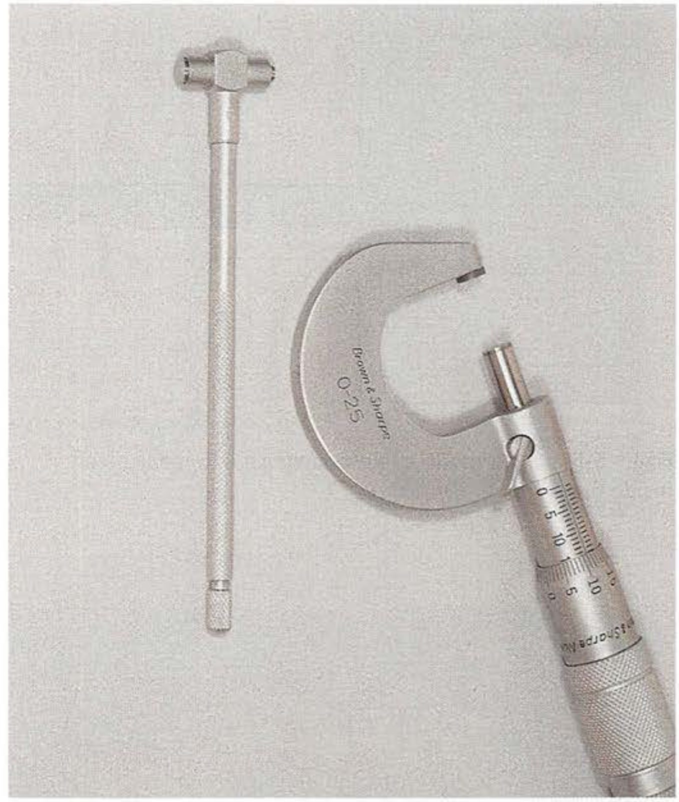
A



B

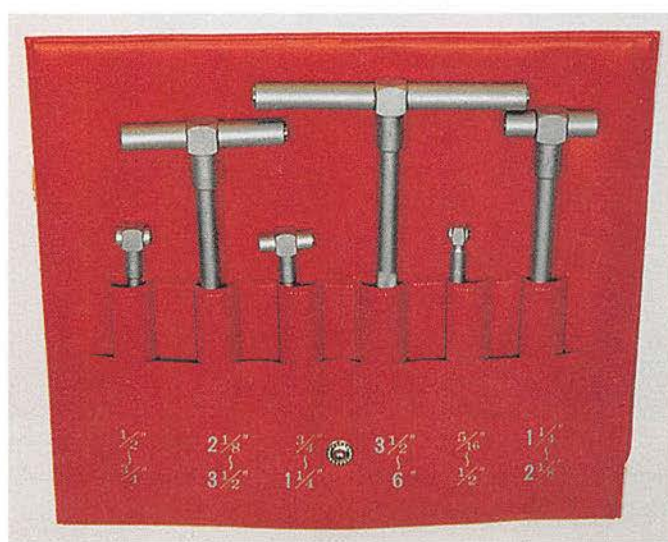
Goodheart-Willcox Publisher

Figure 5-53. Using inside calipers. A—Position the legs of the caliper against the inside walls of the opening. B—Use a steel rule to measure the distance between the caliper legs.



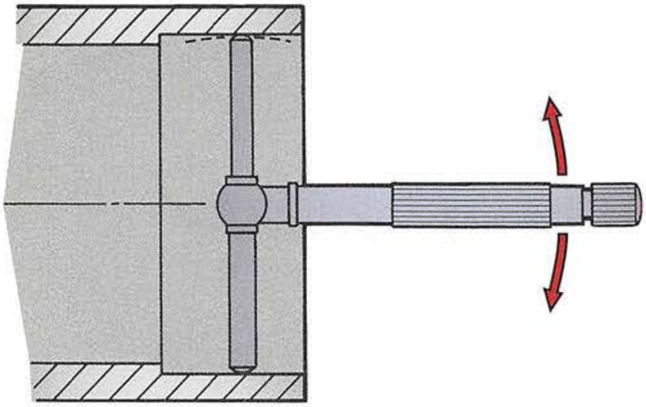
Goodheart-Willcox Publisher

Figure 5-54. A telescoping gage is used with a micrometer.



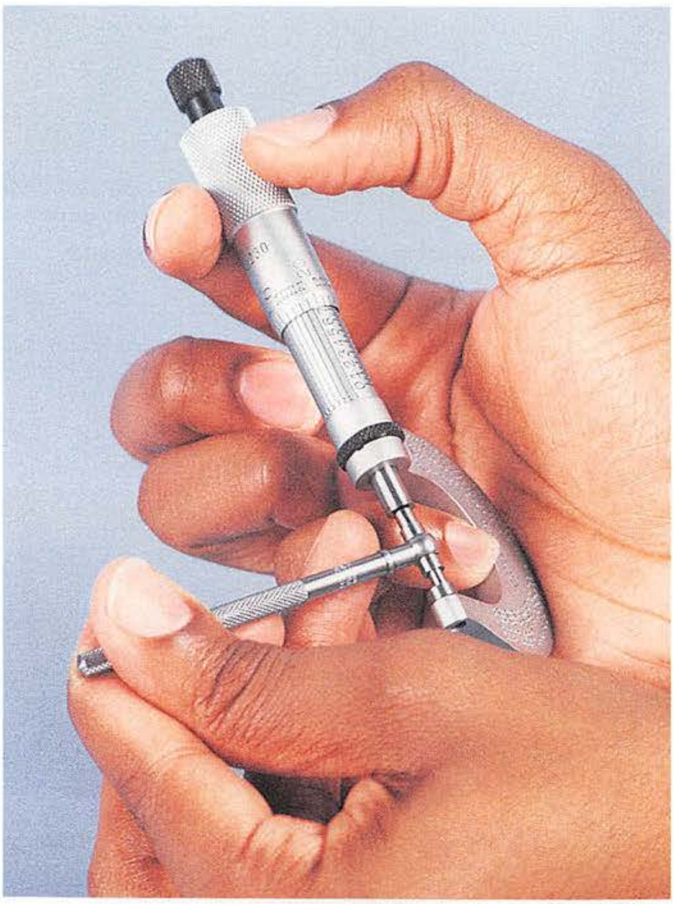
Goodheart-Willcox Publisher

Figure 5-55. A typical set of telescoping gages.



Goodheart-Willcox Publisher

Figure 5-56. Positioning a telescoping gage to measure an inside diameter.

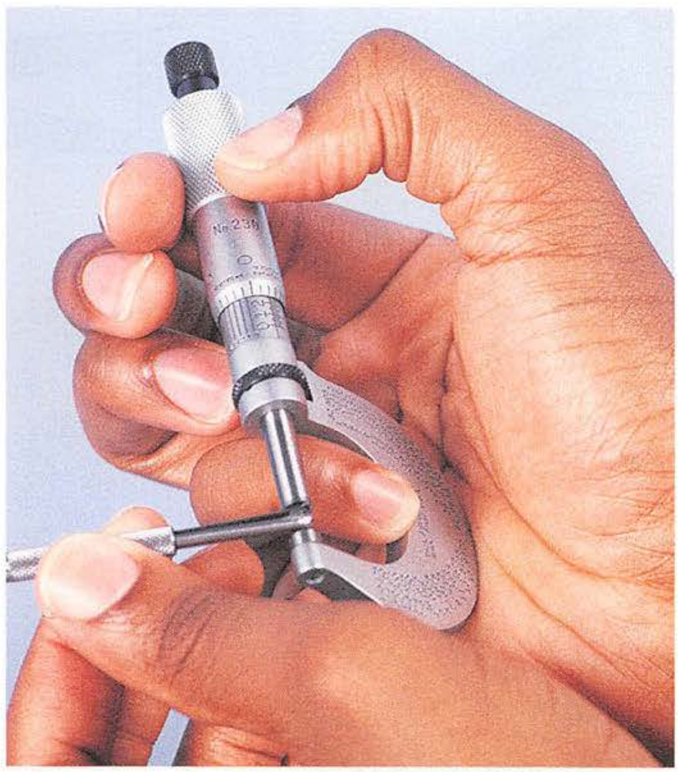


Goodheart-Willcox Publisher

Figure 5-57. After removing the locked telescoping gage, measure it with a micrometer.

Goodheart-Willcox Publisher

Figure 5-58. Small hole gages are used to measure the diameter of holes that are too small for telescoping gages.



Goodheart-Willcox Publisher

Figure 5-59. The correct way to measure a small hole gage with a micrometer.

Chapter Review

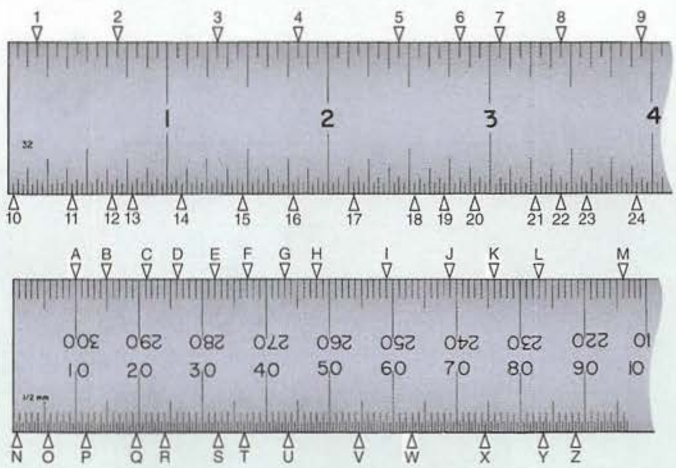
Summary

- Today's machinists need precise measurements that are accurate to a microinch or micrometer.
- The steel rule is the most basic measuring tool.
- Both the micrometer caliper and the vernier caliper are used to measure distances very accurately.
- Various gage tools can be used to make less conventional measurements accurately.
- Dial indicators and electronic gages are used to speed up the inspection of manufactured parts.
- All measuring tools must be cared for properly to maintain the accuracy of their measurements.
- Helper measuring tools are not direct reading and require the help of a rule, micrometer, or vernier caliper to determine the size of the measurement.

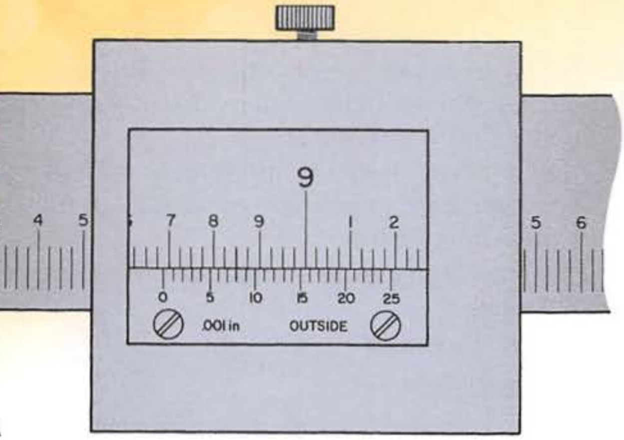
Review Questions

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

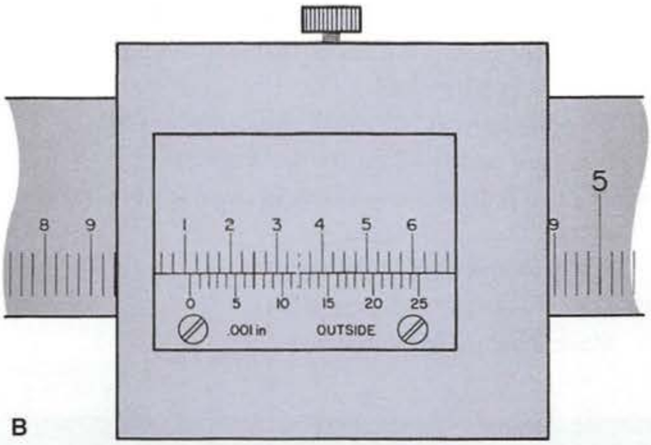
1. One-millionth part of a standard inch is known as a(n) _____.
2. One-millionth part of a meter is known as a(n) _____.
3. A micrometer is capable of measuring accurately to 0.0001″ or 0.002 mm when fitted with a(n) _____.
4. The pitch diameter of a thread can be determined with a(n) _____.
5. List four suggestions for keeping a micrometer or vernier caliper in good condition.
6. What are two advantages of a vernier caliper over a micrometer?
7. The vernier tool for measuring angles is called a(n) _____.
8. How does a double-end cylindrical plug gage differ from a step plug gage?
9. A ring gage is used to check whether external _____ are within the specified tolerance range.
10. Gage blocks are often referred to as _____ blocks.
11. 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.
12. The _____ is used for production inspection. An enlarged image of the part is projected on a screen where it is superimposed on an accurate drawing.
13. Name the measuring device that employs light waves as a measuring standard.
14. For what tasks are fillet and radius gages used?
15. What are some uses for the dial indicator?
16. The dial indicator is available in two basic types. List them.
17. What are helper measuring tools?
18. How is a telescoping gage used?
19. Make readings from the rules.



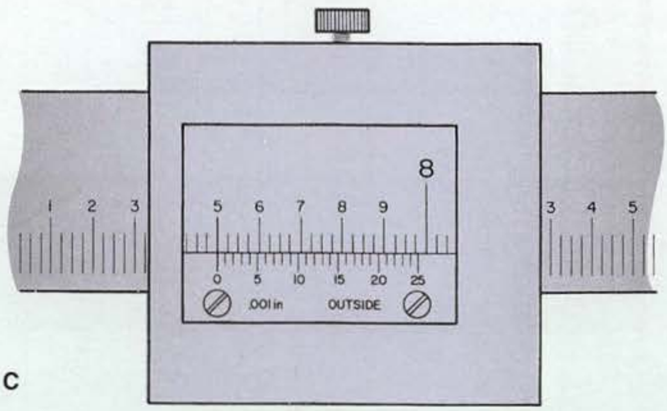
20. Make readings from the vernier scales shown below.



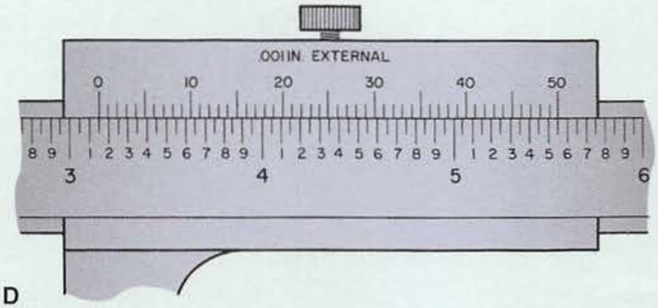
A



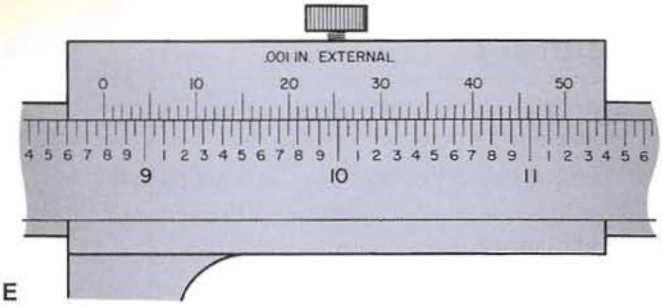
B



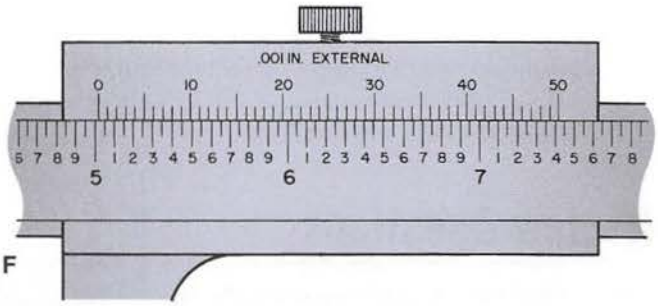
C



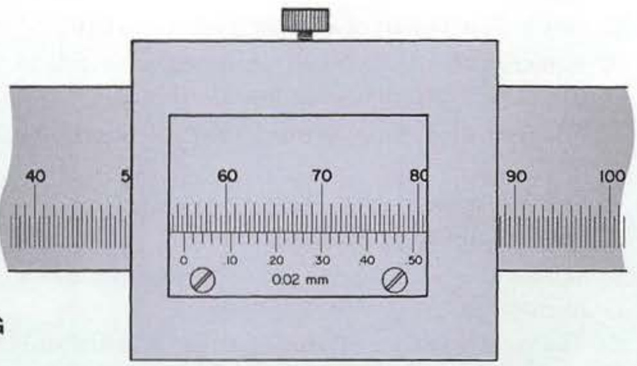
D



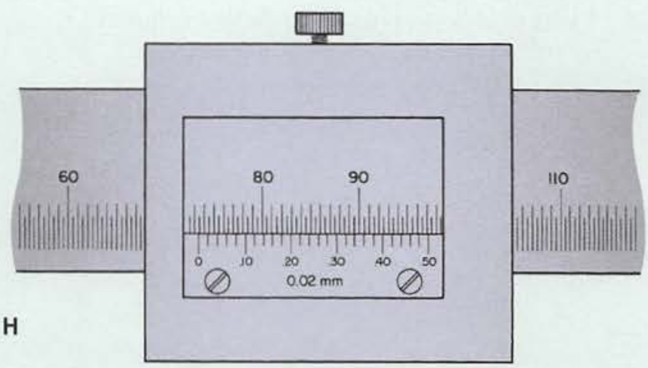
E



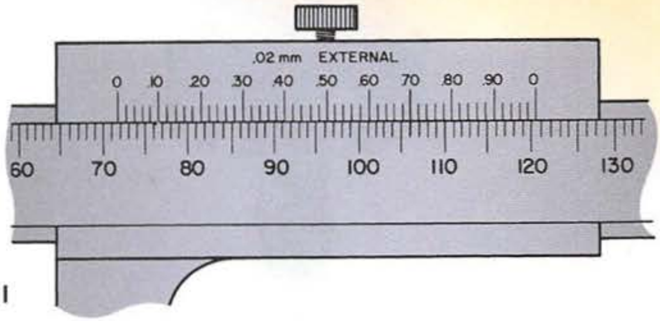
F



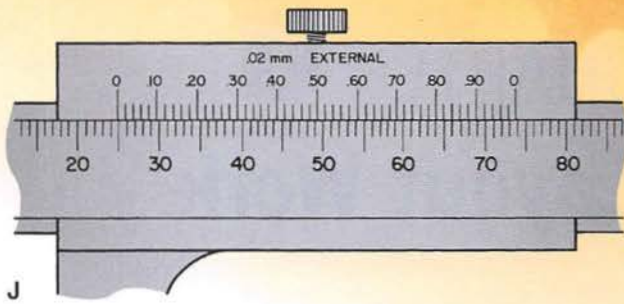
G



H



I

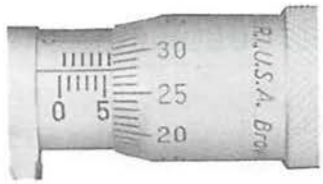


J

21. Make readings from the micrometer illustrations.

A

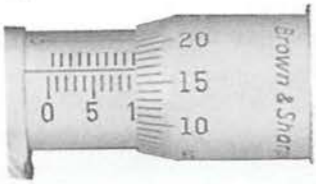
B



C

D

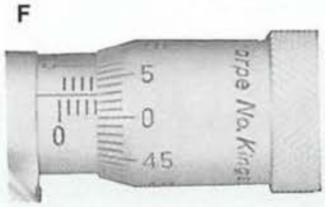
E



F

G

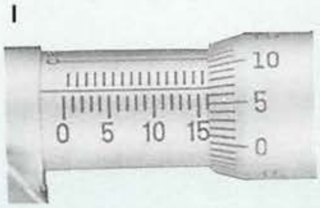
H



I

J

K



L

CHAPTER 6

Layout Work

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Explain why layouts are needed.
- Identify the purpose of common layout tools.
- Use layout tools safely.
- Follow the steps to create a basic layout.

Technical Terms

divider
hermaphrodite caliper
layout
layout dye
protractor
reference line
scriber
sine bar
square
straightedge
surface gage
surface plate
trammel
V-block
vernier protractor

In the shop, ***layout*** is the process of 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.

6.1 Layout Tools

The tools used for layout include many common hand tools. The accuracy of the job depends on the proper and careful application of these tools. See **Figure 6-1**. A good layout job is determined by its neatness, accuracy, and legibility.

6.1.1 Layout Dye

The shiny finish of most metals makes it difficult to distinguish layout lines. For this reason, a coating must be placed on the metal before layout. Many coatings may be used to make layout lines stand out better. Of these coatings, ***layout dye*** is probably the easiest to use. This fluid, which is usually blue, offers an excellent contrast between the metal and the layout lines. See **Figure 6-2**. 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 rolled steel as a layout background. A pencil should not be used because it makes too wide a mark and rubs off too easily.

Goodheart-Willcox Publisher

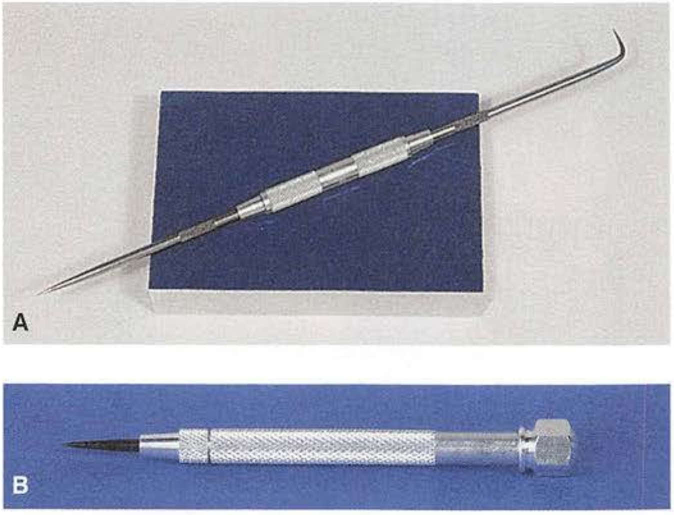
Figure 6-2. Layout dye makes the layout lines scribed into metal stand out.

6.1.2 Scriber

An accurate layout requires fine lines that must be scribed (scratched) into the metal. A ***scriber*** is used to produce straight lines. The point is made of hardened steel and is kept needle-sharp by frequent honing on a fine oilstone. Two of the many styles of scribers available are shown in **Figure 6-3**.

Goodheart-Willcox Publisher

Figure 6-1. A few of the tools needed to make a layout.



Goodheart-Willcox Publisher

Figure 6-3. Scribers are used to mark parts during layout and to draw straight lines. 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.

6.1.3 Divider

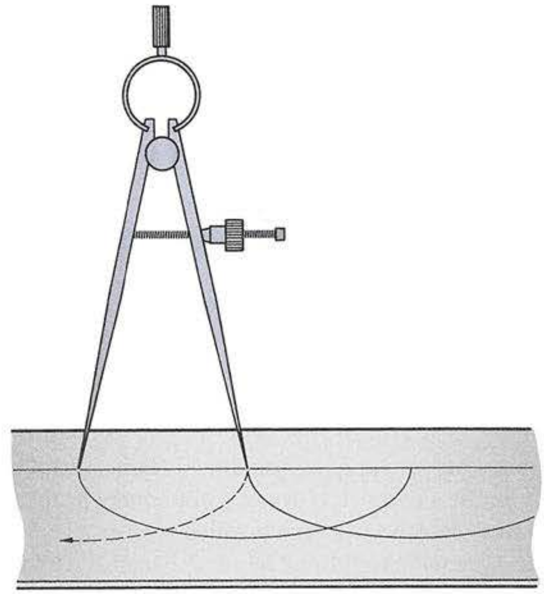
While a scriber is used to draw straight lines, a ***divider*** is used to draw circles and arcs, **Figure 6-4**. Both legs of the tool must be equal in length and must be kept pointed. Measured distances can be laid out with a divider, **Figure 6-5**. To set the tool to the correct distance, set one 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 6-6**.

Circles and arcs that are too large to be made with a divider are drawn with a ***trammel***, **Figure 6-7**. 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.

Goodheart-Willcox Publisher

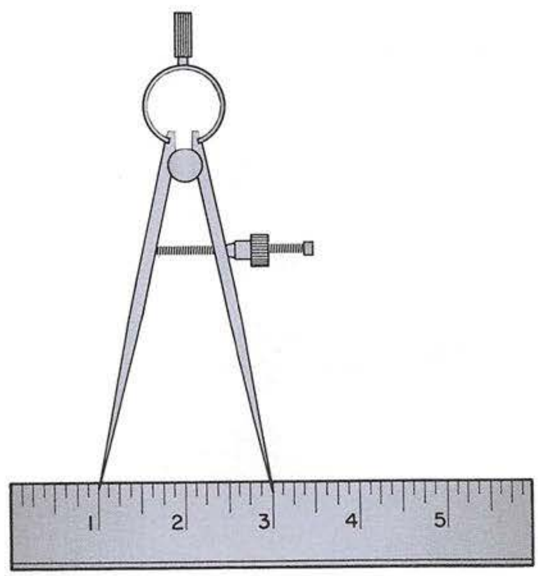
Figure 6-4. A divider is used to mark arcs and circles and to lay out distances.

The ***hermaphrodite caliper*** is a layout tool with one leg that is shaped like a caliper and the other pointed like a divider, **Figure 6-8**. Lines parallel to the edge of the material, either straight or curved, can be drawn with the tool, **Figure 6-9**. It can also be used to locate the center of irregularly shaped stock.



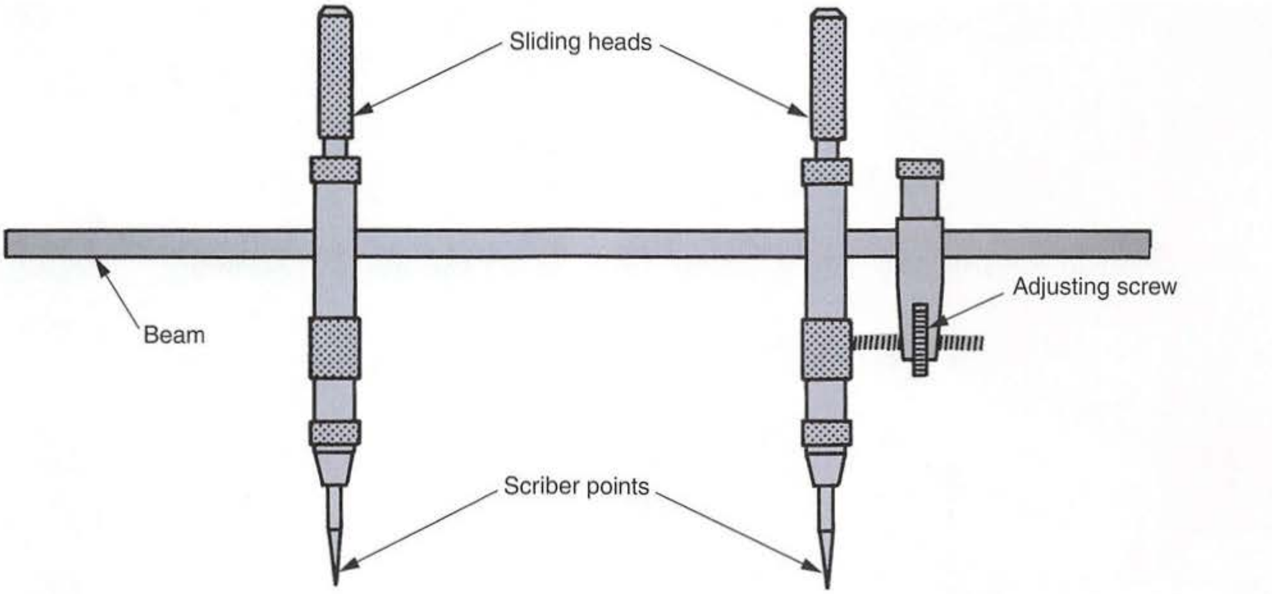
Goodheart-Willcox Publisher

Figure 6-5. Equal spaces can be laid out by "walking" the divider.



Goodheart-Willcox Publisher

Figure 6-6. To set a divider to a desired dimension, place it on a rule as shown.

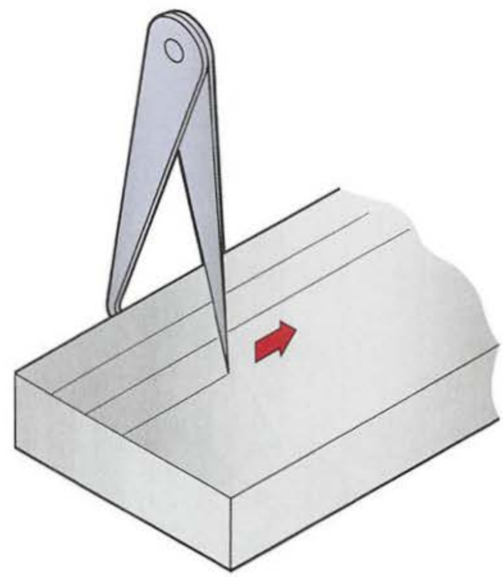


Goodheart-Willcox Publisher

Figure 6-7. Large circles and arcs are drawn with a trammel.

L. S. Starrett Co.

Figure 6-8. A hermaphrodite caliper has a blunt end for the sliding surface and a point for scribing.



Goodheart-Willcox Publisher

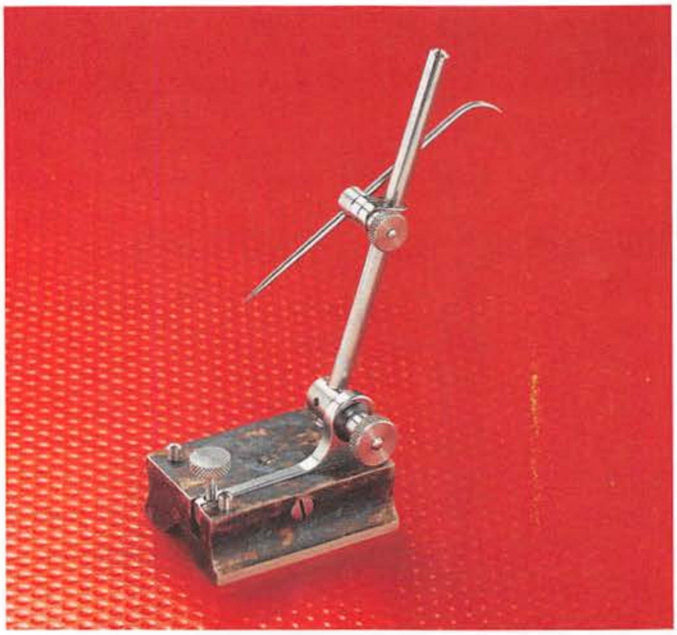
Figure 6-9. Scribing lines parallel to an edge with a hermaphrodite caliper.

SAFETY NOTE

- Never carry an open scriber, divider, trammel, or hermaphrodite caliper in your pocket. The sharp points of these tools can puncture the skin easily.
- Always cover sharp points with a cork when the tool is not being used.
- Wear goggles when grinding scriber points.

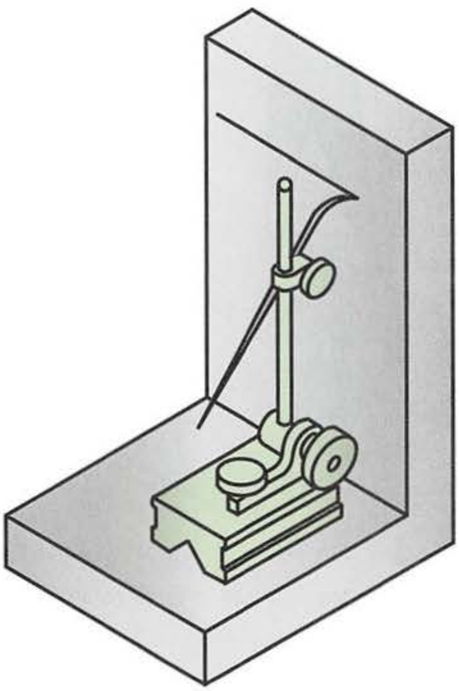
6.1.4 Surface Gage

A ***surface gage*** has many uses, but is most frequently used for layout work, **Figure 6-10**. It consists of a base, spindle, and scriber. An adjusting screw allows fine adjustments. The scriber is mounted so that it can be pivoted into any position. A surface gage can be used to scribe lines at a given height and parallel to the surface, **Figure 6-11**. A V-slot in the base permits the tool to be used on curved surfaces.



L. S. Starrett Co.

Figure 6-10. This small surface gage is designed for light work.



Goodheart-Willcox Publisher

Figure 6-11. Carefully slide the surface gage to scribe lines parallel to the base. Handle the gage carefully because sharp points can cause injury.

As an alternative to a surface gage, a scribe can be mounted in a digital height gage and used to scribe lines at given heights. Although it requires more space than a surface gage, using the height gage eliminates the need to set the tip of the scribe in the surface gage. See **Figure 6-12.** This eliminates a step in which human error can occur. However, in close quarters or when curved surfaces are involved, the smaller surface gage is the tool of choice.

Tibor Machine Products

Figure 6-12. A digital vernier height gage with a dial indicator being used to check whether the V-blocks are parallel to the surface plate.

To check whether a part is parallel to a given surface, fit the surface gage with a dial indicator. Use gage blocks to set the indicator to the required dimension. Then move the tool back and forth along the work.

6.1.5 Surface Plate

Every linear measurement depends on 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:

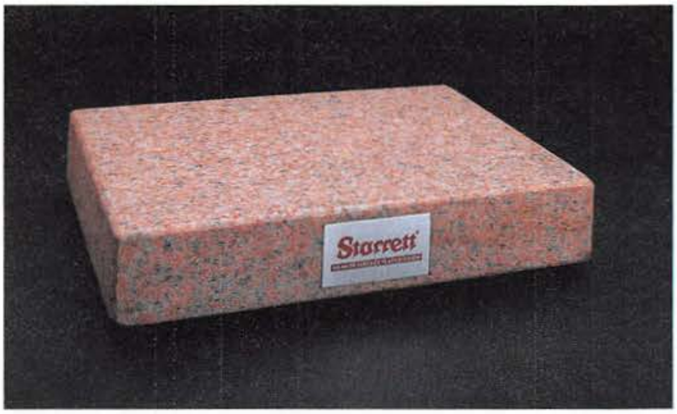
- Grade AA for laboratories.
- Grade A for inspection.
- Grade B for toolroom and layout applications.

Most surface plates made today are granite, **Figure 6-13,** but some are cast iron. Granite is more stable. Cast iron surface plates are affected by temperature changes.

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.

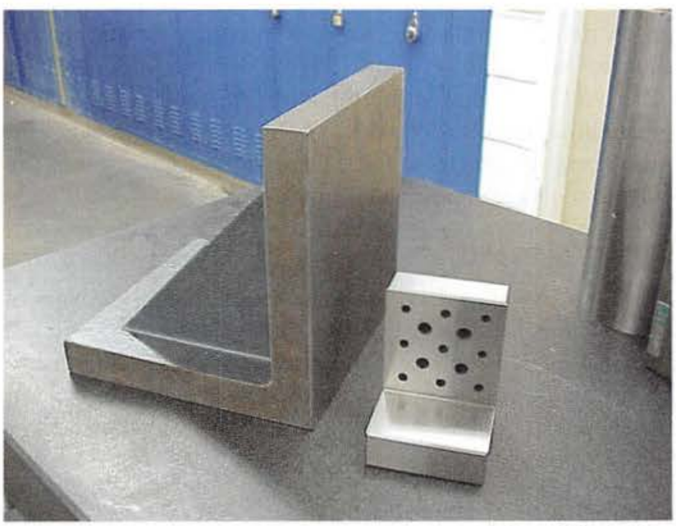
When square reference surfaces are needed, a right-angle plate is used, **Figure 6-14**. The plates can be placed in any position. The work is then clamped to the face for layout, measurement, or inspection.

An accurate surface parallel to the surface plate can be obtained using box parallels, **Figure 6-15**. All surfaces are precision-ground to close tolerances.



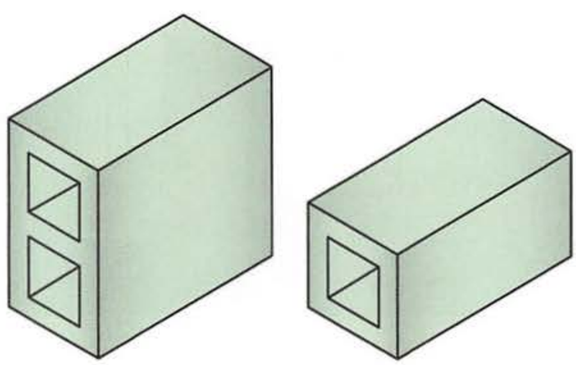
L. S. Starrett Co.

Figure 6-13. Surface plates are available in various grades and materials. Granite surface plates are the most common.



Goodheart-Willcox Publisher

Figure 6-14. Right-angle plates are often used to check perpendicular surfaces.



Goodheart-Willcox Publisher

Figure 6-15. Box parallels are available in a number of sizes.

6.1.6 V-Blocks

V-blocks support round work for layout and inspection, **Figure 6-16**. 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.

SAFETY NOTE

Get help when you must move heavy items, such as angle plates or V-blocks.

6.1.7 Straightedge

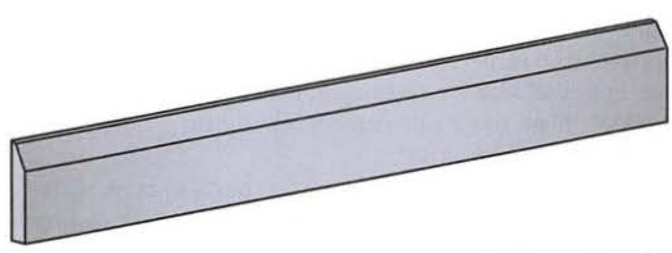
A ***straightedge***, **Figure 6-17**, is used to check the accuracy of long, flat surfaces. This tool is also used for laying out long, straight lines. Straightedges can be made from steel or granite, although steel is more common.

6.1.8 Sine Bar

The ***sine bar***, **Figure 6-18A**, is a tool that can be used on a surface plate with gage blocks to create an accurate right

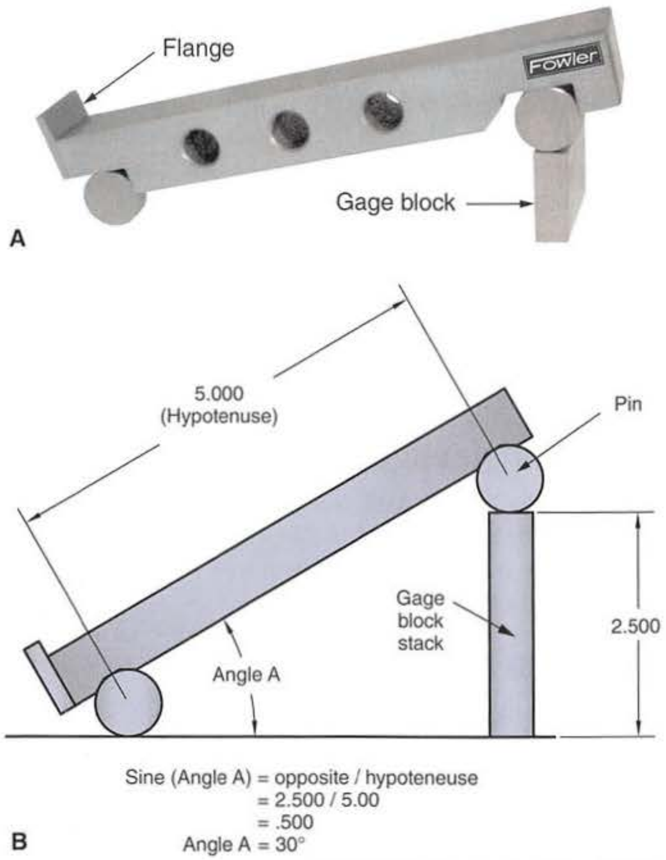
Goodheart-Willcox Publisher

Figure 6-16. V-blocks can be used to hold round stock for layout and measurement work.



Goodheart-Willcox Publisher

Figure 6-17. Steel straightedges are very common. They are also available in granite.



Fowler High Precision; Goodheart-Willcox Publisher

Figure 6-18. A—A sine bar can be used to create accurate right triangles. B—Using a sine bar to calculate an angle.

triangle in order to lay out angles very precisely. Sine bars are manufactured so that the distance between the centers of the cylindrical pins is a precise value, such as 5.0000″ or 6.0000″, so that right-angle trigonometry can be used to calculate the angle of the bar once the stacked gage blocks have been placed under one of the pins.

The sine bar in **Figure 6-18B** is a 5.0000″ sine bar, which means that the distance between the centers of the pins is 5.0000″. A stack of gage blocks totaling 2.5000″ in height has been placed under one of the pins. You can calculate the angle using the following formula:

$$\text{Sine (Angle A)} = \frac{\text{length of opposite side}}{\text{hypotenuse}}$$

Stack the gage blocks directly opposite the angle you want to find (Angle A). The hypotenuse is the distance between the centers of the sine bar pins. When you divide the opposite side by the hypotenuse, you get the answer 0.5000″. Then use a calculator to obtain the inverse sine of 0.5000; the answer is 30°.

When you have only one or a few parts to mark, using a protractor head and rule as described in the next section is usually the fastest method. However, if you have many parts that need to be marked, using a sine bar will usually be the fastest way to finish the job.

6.1.9 Squares

The ***square*** is used to check 90° (square) angles. It is also used to lay 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.

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

Squares and Double Squares

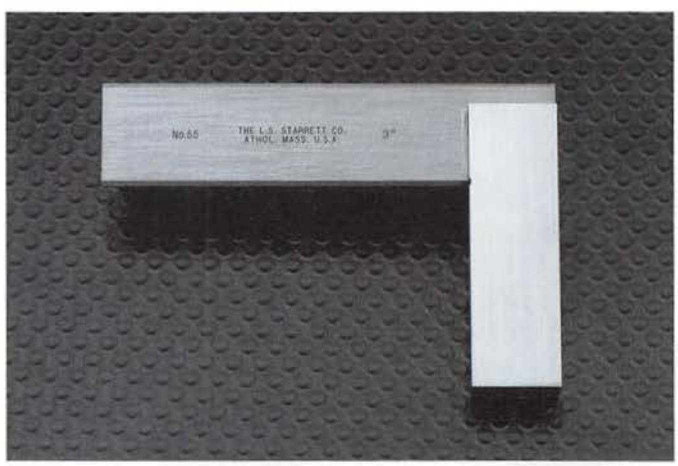
A hardened steel square, commonly called a *machinist's square*, is recommended when extreme accuracy is required. See **Figure 6-19**. The square has true right angles, both inside and outside. It is accurately ground and lapped for straightness and parallelism. The tool comes in sizes up to 36″ (910 mm).

A double square is more practical than a steel square for many jobs because the sliding blade is adjustable and interchangeable with other blades, **Figure 6-20**. 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).

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.

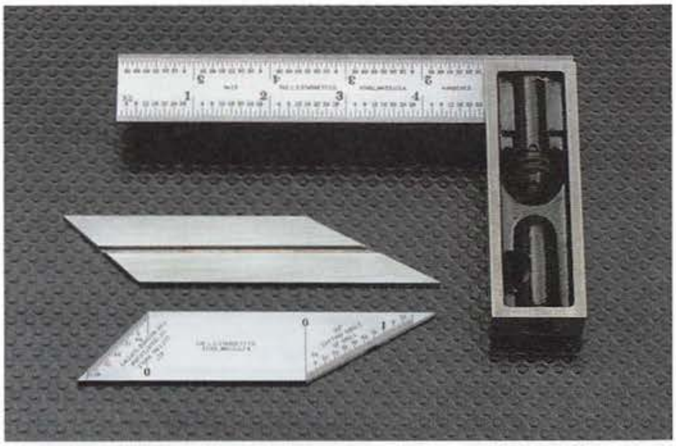
Combination Sets

The combination set consists of a hardened blade, square head, center head, and bevel protractor. The blade fits all three heads, **Figure 6-21**. Combination sets are adaptable to a variety of operations, making them especially valuable in the shop.



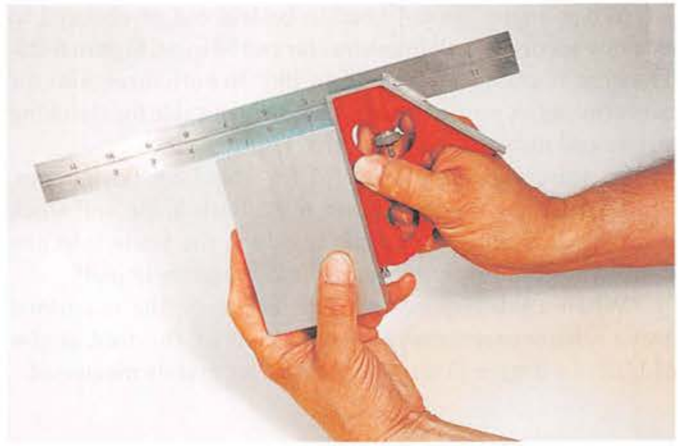
L. S. Starrett Co.

Figure 6-19. The hardened steel square is handy during layout work.



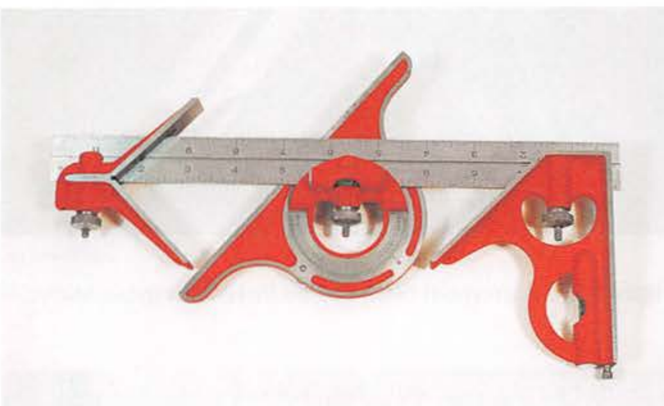
L. S. Starrett Co.

Figure 6-20. This double square has a graduated blade, a beveled blade, and a drill grinding blade.



Goodheart-Willcox Publisher

Figure 6-22. The combination set can be used to check squareness and to measure like a depth gage.



Goodheart-Willcox Publisher

Figure 6-21. A combination set will perform various layout tasks.

Goodheart-Willcox Publisher

Figure 6-23. Using a center head and rule to locate the center of a piece of round stock.

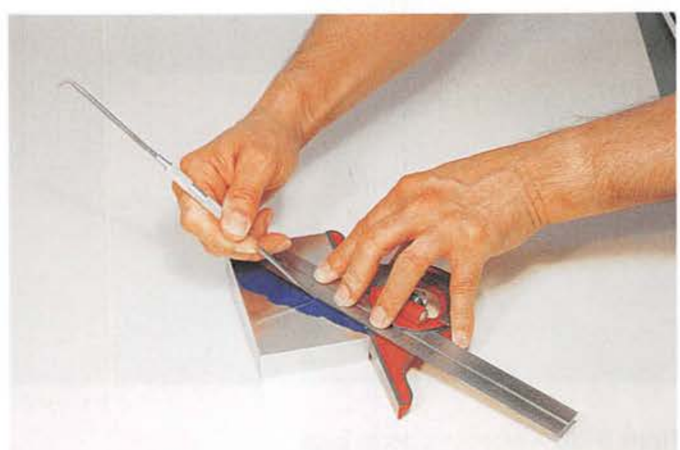
The square head, which has one 45° edge, allows the tool to serve as a miter square. When the blade is projected a specified distance below the edge, it can also serve as a depth gage, **Figure 6-22**. The spirit level, fitted in one edge, allows it to be used as a simple level.

With the rule properly inserted, the center head can be used to quickly find the center of round stock. This is illustrated in **Figure 6-23**.

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 6-24**. The head also has a built-in level for positioning angles for inspection, layout, or machining.

6.1.10 Tools for Measuring Angles

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



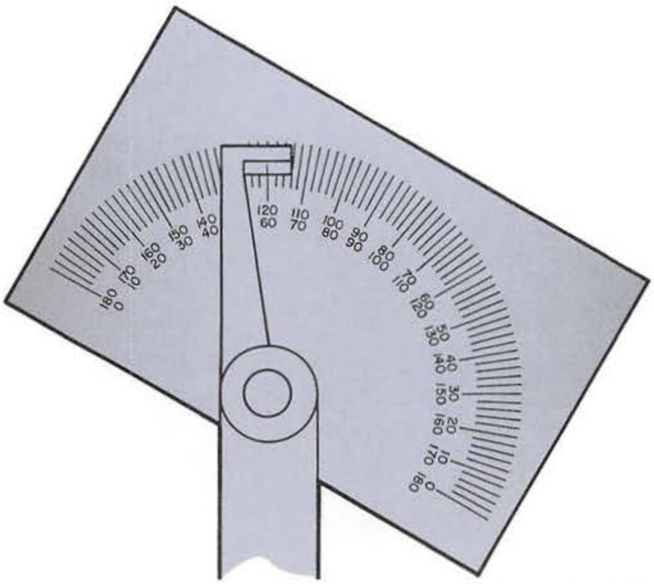
Goodheart-Willcox Publisher

Figure 6-24. Angular settings on layouts can be made with the protractor head and rule of the combination set.

When angles do not need to be laid out or checked to extreme accuracy, a plain ***protractor*** can be used, **Figure 6-25**. 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 6-26**.

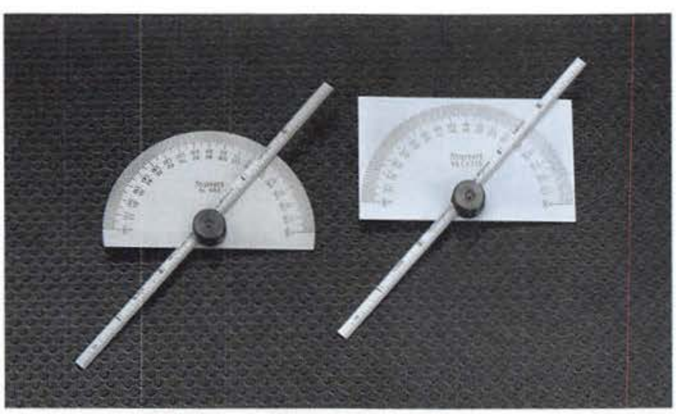
A universal bevel is useful for checking, laying out, and transferring angles, **Figure 6-27**. Both blade and stock are slotted, making it possible to adjust the blade into any desired position. A thumbscrew locks it tightly in place.

When a job requires extreme accuracy, the machinist uses a ***vernier protractor***, **Figure 6-28**. With this tool, angles of 1/12 of a degree (5 minutes) can be accurately measured.



Goodheart-Willcox Publisher

Figure 6-25. A plain steel protractor measures angles with moderate precision.



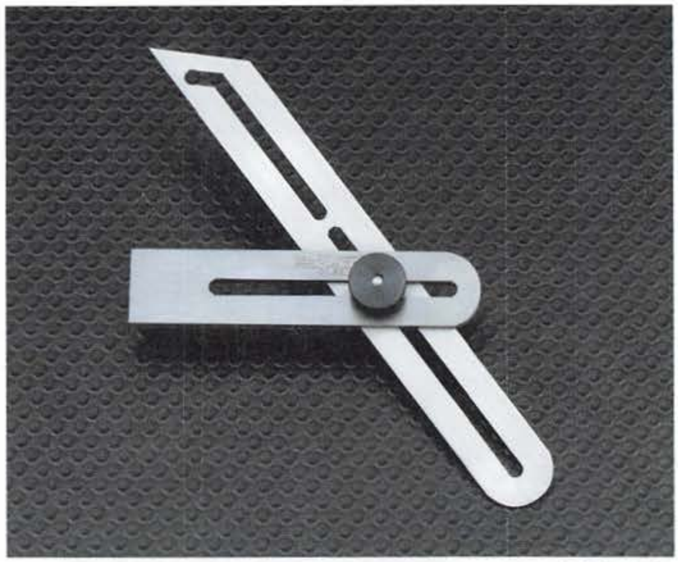
L. S. Starrett Co.

Figure 6-26. A protractor depth gage.

6.2 Layout Steps

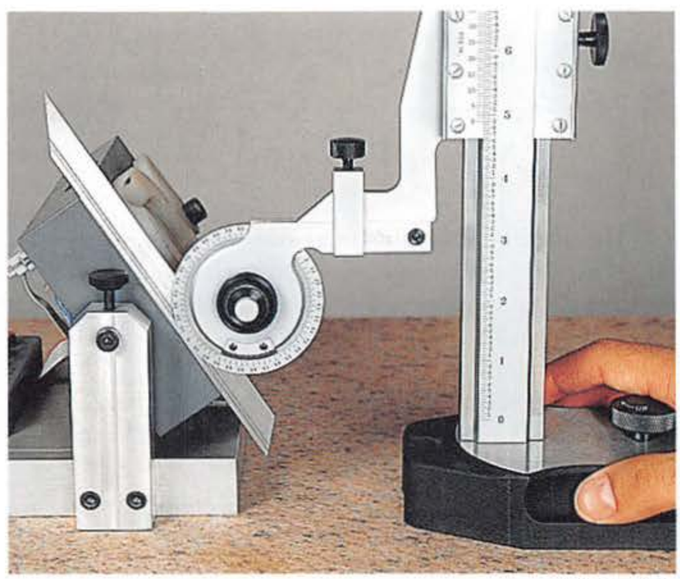
Each layout job requires planning before the operation can be started. **Figure 6-29** shows a typical job. Use the following planning procedure:

1. Carefully study the drawings.
2. Cut the stock to size.



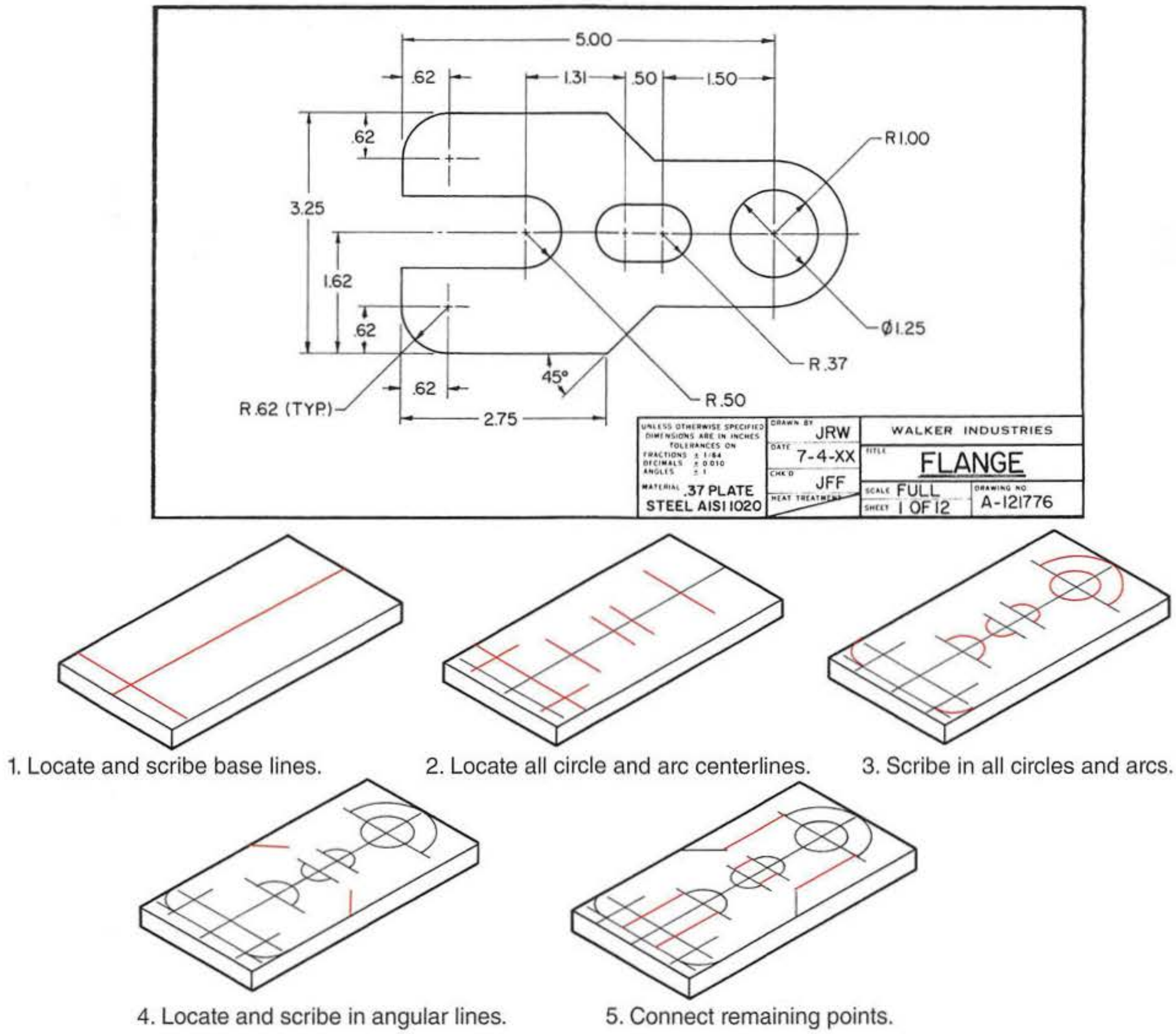
L. S. Starrett Co.

Figure 6-27. A universal bevel can be locked at various angles.



L. S. Starrett Co.

Figure 6-28. Precise angular measurements are made with a vernier protractor. In this view, the protractor is mounted on a height gage.



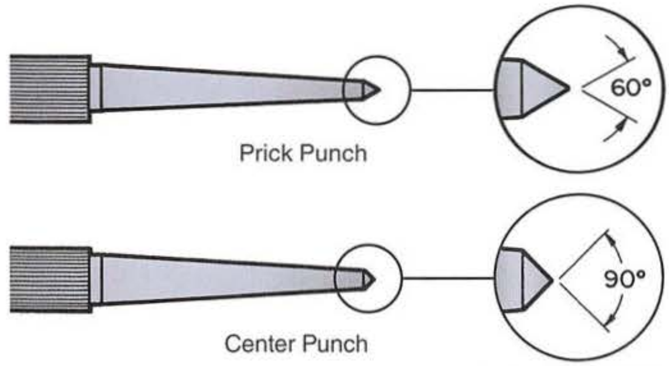
Goodheart-Willcox Publisher

Figure 6-29. Compare the part drawing with steps involved in laying out the job.

SAFETY NOTE

Remove all burrs and sharp edges from stock before starting layout work.

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 instead of the base line.
5. Locate the 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. Check the position of the prick punch mark, then enlarge it slightly with a center punch, **Figure 6-30**.



Goodheart-Willcox Publisher

Figure 6-30. 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.

WORKPLACE SKILLS

Applying for a Job

Job postings can be found through a variety of different media. Want ads in a newspaper contain useful information on jobs in your area. You can also find job leads through networking and talking to friends and family. The Internet may be one of the best sources to search for job openings. There are job-search engines and government agencies that can assist you in finding and applying for jobs.

Once you find a job opening, you will generally be asked to fill out a job application. This can be done in person or online. You should also have a résumé and a portfolio prepared, as many employers will ask to see these along with the application. Employers often use these forms to screen applicants for the skills needed on the job. The appearance of the application form can give an employer the first impression about you. Fill out the form accurately, completely, and neatly. How well you accomplish this can determine whether you get the job.

Many employers now request electronic applications, either through the employer's website or an independent job-search website. When filling out an online application, include key terms for which the employer may search. This will help you stand out from the many other applicants.

When preparing your application, be sure to save it in the appropriate file format. If a preferred format is not given, it is best to save the application in Microsoft Word file format (.doc or .docx) or save it as a PDF file. This will enable the employer to find specific search terms in your document. Be sure to complete all the fields of the application. Many job-search sites have sample forms on which you can practice before attempting to complete a real application.

Chapter Review

Summary

- Layout work is an important first step to the machining process.
- Machinists use a variety of layout tools, including layout dye, scribers, dividers, surface gages, surface plates, V-blocks, and straightedges.
- A variety of squares and protractors are used to measure angles.
- Basic layouts can be created following 10 steps.

Review Questions

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

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

CHAPTER 7
Hand Tools

Chapter Outline

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.
- Set up and cut a keyway using a keyway broach and an arbor press.

Technical Terms

abrasive
American National Thread System
arbor press
blind hole
broach
broaching
classes of fit
foot-pound (ft·lb)
keyway
newton-meter (N·m)
reamer
safe edge
set
torque
Unified System

Selecting and using hand tools correctly will help you do a job safely and quickly. When a hand tool is used incorrectly, it can be damaged or someone may be injured. It is to your advantage to learn to work properly with hand tools.

7.1 Clamping Devices

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

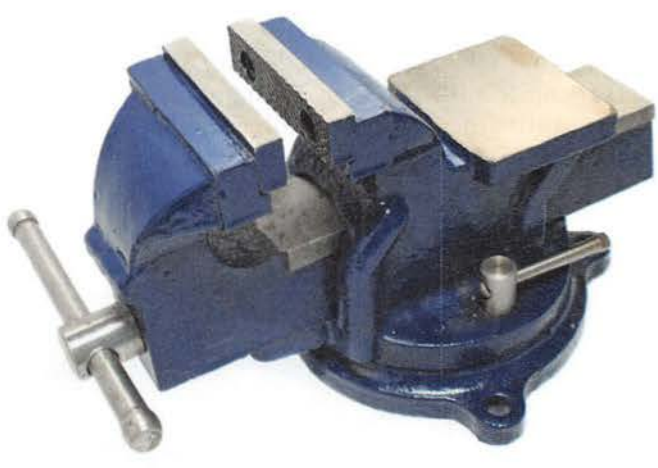
7.1.1 Vises

The bench vise is used for many holding tasks. It should be mounted on the edge of the bench, far enough out to permit clamping long work in a vertical position. A vise may have a solid base or a swivel base, which allows the vise to be rotated. See **Figure 7-1**.

Small precision parts may be held in a small bench vise or toolmaker's vise, **Figure 7-2**. This type of vise can be rotated and tilted to any desired position. Vise size is determined by the width of the jaws, **Figure 7-3**.

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. When clamping work that could be damaged or marred by the jaw serrations, cover the jaws with soft copper, brass, or aluminum caps, **Figure 7-4**.

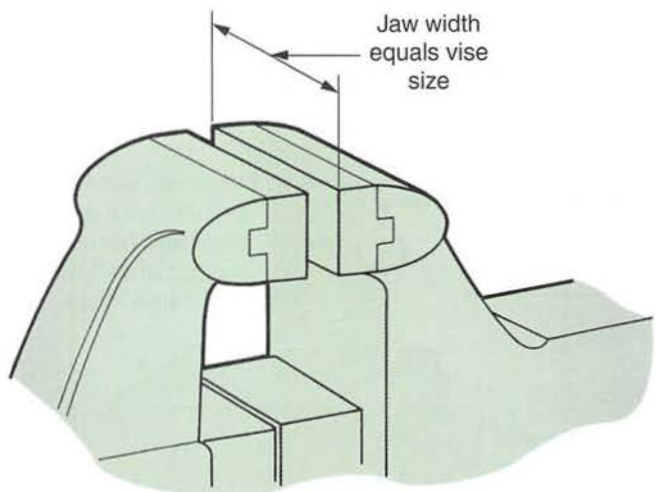


ekipaj/Shutterstock.com

Figure 7-1. A swivel-base bench vise. The base is made in two parts so that the body can be rotated to any desired position.

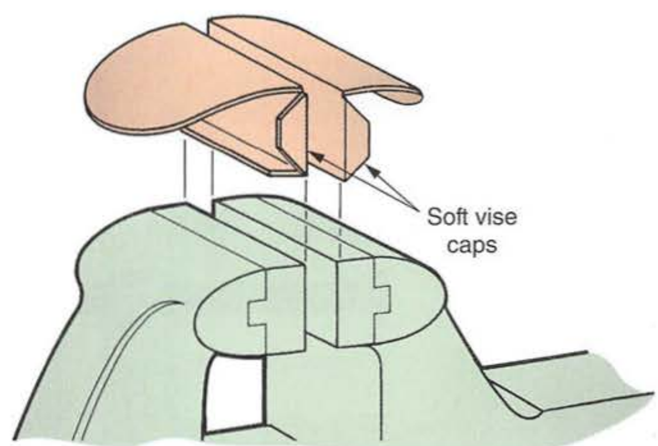
Birgit Reitz-Hofmann/Shutterstock.com

Figure 7-2. A small vise used by a toolmaker. It can be rotated and pivoted to desired working position.



Goodheart-Willcox Publisher

Figure 7-3. The size of a vise is determined by the width of its jaws.



Goodheart-Willcox Publisher

Figure 7-4. Caps made of copper, lead, or aluminum are slipped over hardened vise jaws to protect work from becoming marred or damaged by jaw serrations.

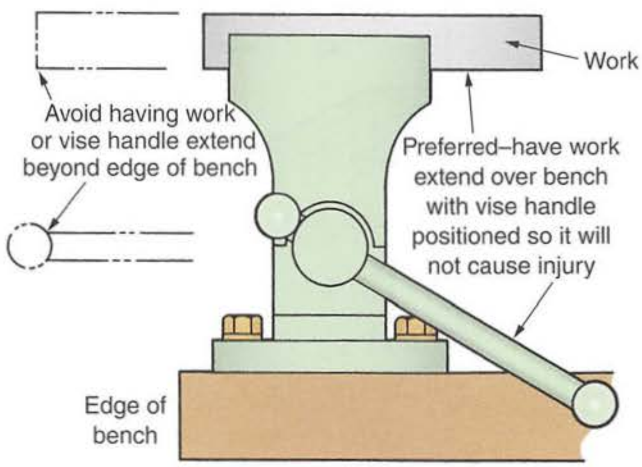
SAFETY NOTE

When clamping a job in a vise, do not allow the vise handle or work to project beyond the edge of the bench, **Figure 7-5**.

7.1.2 Clamps

The C-clamp and the parallel clamp hold parts together while they are worked on. The C-clamp, **Figure 7-6**, 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** 7-7. Placing strips of paper the width of the clamp jaw between the work and the jaws will improve clamping action.

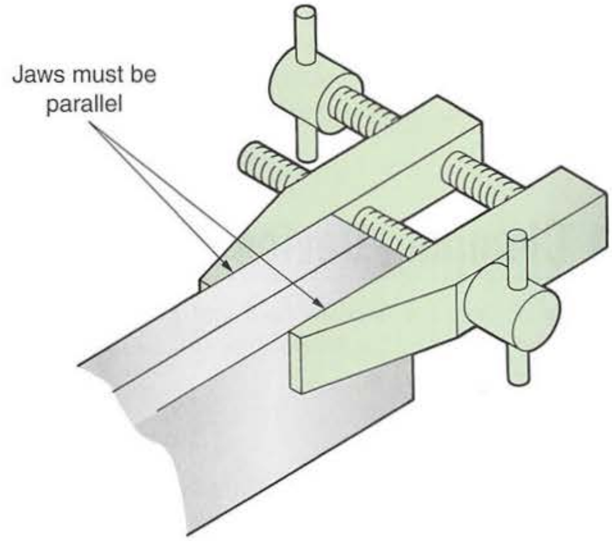


Goodheart-Willcox Publisher

Figure 7-5. To prevent injury, avoid letting the vise handle or work project beyond the edge of the bench.

Goodheart-Willcox Publisher

Figure 7-6. C-clamps are available in a range of sizes.



Goodheart-Willcox Publisher

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

7.2 Pliers

Pliers of various types are among the most useful hand tools in the shop. In general, pliers are used to grip and hold the workpiece. Some types are also used to cut or bend wire.

7.2.1 Types of Pliers

Slip-joint pliers are widely used for holding tasks. See **Figure 7-8**. 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 plier size is measured by the overall length of the tool.

Some slip-joint 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 also widely used for light cutting tasks, **Figure 7-9**. 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.

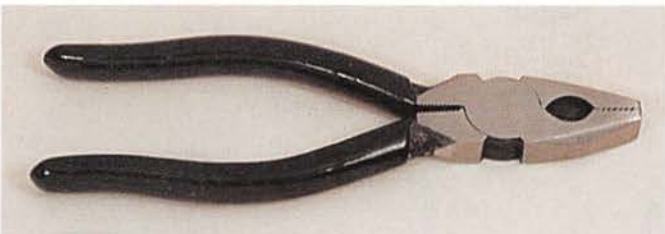
Side-cutting pliers are capable of cutting heavier wire and pins, **Figure 7-10**. Some of these pliers have a wire-stripping

Goodheart-Willcox Publisher

Figure 7-8. Slip-joint pliers are used for holding tasks.

Goodheart-Willcox Publisher

Figure 7-9. Diagonal pliers can cut flush with a surface.



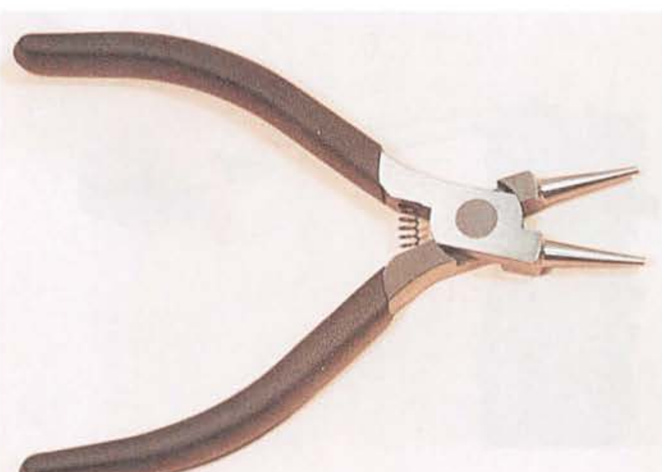
Goodheart-Willcox Publisher

Figure 7-10. Side-cutting pliers have square jaws for holding and cutting tasks.

groove and insulated handles. They are made in 6″, 7″, and 8″ lengths.

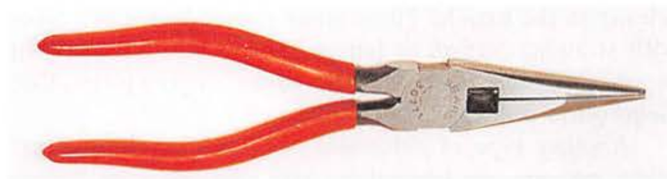
Round-nose pliers, **Figure 7-11**, are helpful for forming wire and light metal. Their jaws are smooth and will not mar the metal being grasped. Round-nose pliers are available in 4″, 4 1/2″, 5″, and 6″ sizes.

Needle-nose pliers are available in both straight and curved-nose types. They are handy for holding small work. They are also used for other holding tasks when workspace is limited because they can reach into cramped places. See **Figure 7-12**.



Goodheart-Willcox Publisher

Figure 7-11. Round-nose pliers have smooth jaws.



A

B

Goodheart-Willcox Publisher

Figure 7-12. 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.

Tongue-and-groove pliers have aligned teeth for flexibility in gripping different size work, **Figure 7-13**. 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; the larger 16″ size has eleven adjustments.

Locking pliers have jaw openings that can be adjusted through a range of sizes using a threaded mechanism on one handle. See **Figure 7-14**. 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 using the quick

Goodheart-Willcox Publisher

Figure 7-13. The jaws of tongue-and-groove pliers expand to hold large objects.

Goodheart-Willcox Publisher

Figure 7-14. Adjustable clamping pliers can be locked on work of different sizes.

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 adjustable clamping pliers, Vise Grip® pliers, and Tag-L-Lock® pliers.

Another type of adjustable pliers, called Robo-Grip® pliers, permits one-handed jaw-size adjustment by merely squeezing the handles. See **Figure 7-15**. This type of adjustable pliers does not have a locking feature.

7.2.2 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 too much leverage is applied to the handles.
- Clean and oil pliers occasionally 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 the right size for the job.

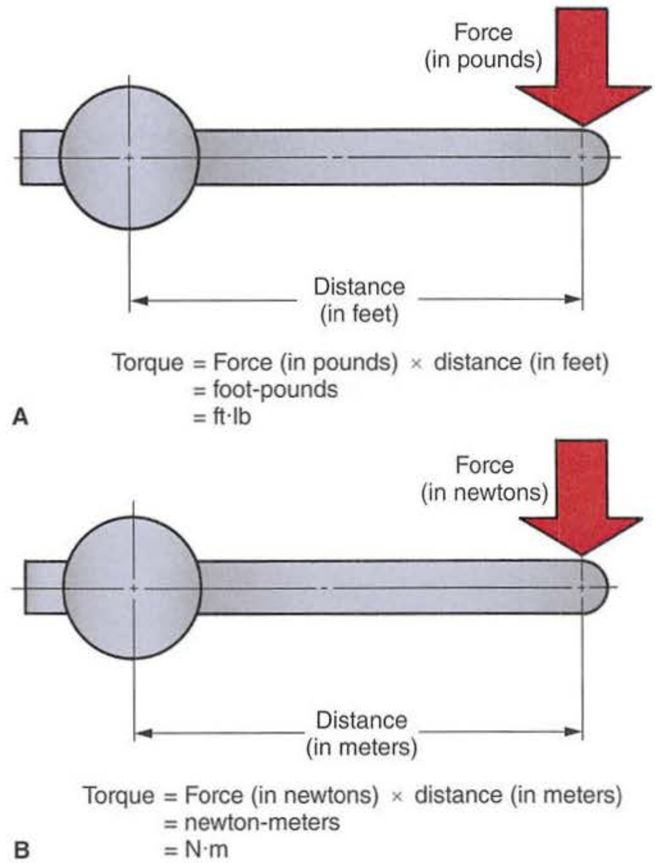
7.3 Wrenches

Wrenches are 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 are described in this chapter.

7.3.1 Torque 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·lb***) or the metric equivalent, ***newton-meters*** (***N·m***). Torque is the product of the force applied times the length of the lever arm. See **Figure 7-16**.

A torque 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 overtightening, which could cause the fastener or part to fail or cause the work to warp or spring out of shape. See **Figure 7-17**.

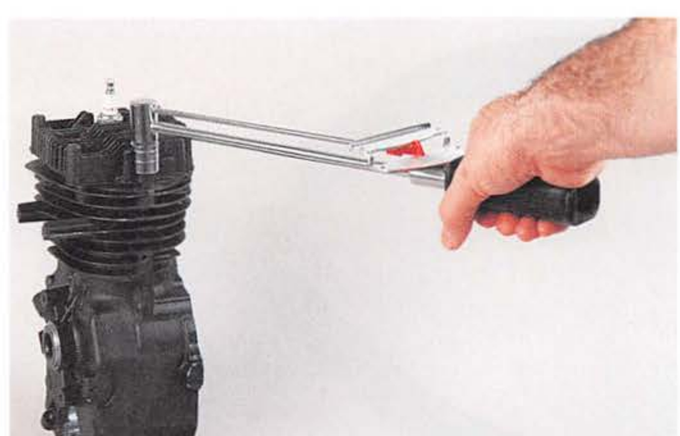


Goodheart-Willcox Publisher

Figure 7-16. 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.

Goodheart-Willcox Publisher

Figure 7-15. Some types of adjustable pliers offer one-handed operation.



Goodheart-Willcox Publisher

Figure 7-17. Torque 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 wrenches, **Figure 7-18**. Some torque wrenches are direct reading. Others 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 7-19**. 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 wrenches provide accurate measurements whether they are pushed or pulled. However, to prevent hand injury, the preferred method is to pull on the wrench handle.

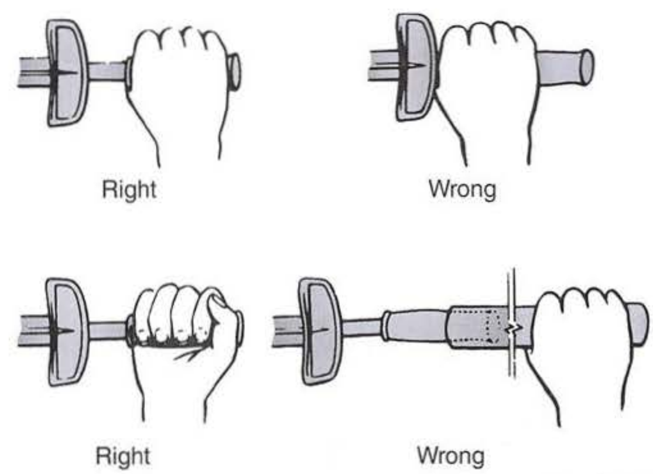
7.3.2 Adjustable Wrenches

The term *adjustable wrench* could be used to describe several different types of wrenches, including the "monkey wrench" and pipe wrench. 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 7-20**.

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.

When using the adjustable wrench, keep the following tips in mind:

- Place the wrench on the bolt head or nut so that the movable jaw faces the direction the fastener is to be rotated, **Figure 7-21**.
- Adjust the thumbscrew so the jaws fit the bolt head or nut snugly, **Figure 7-22**.
- Do not place an extension on the wrench handle for additional leverage.
- Never hammer 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.

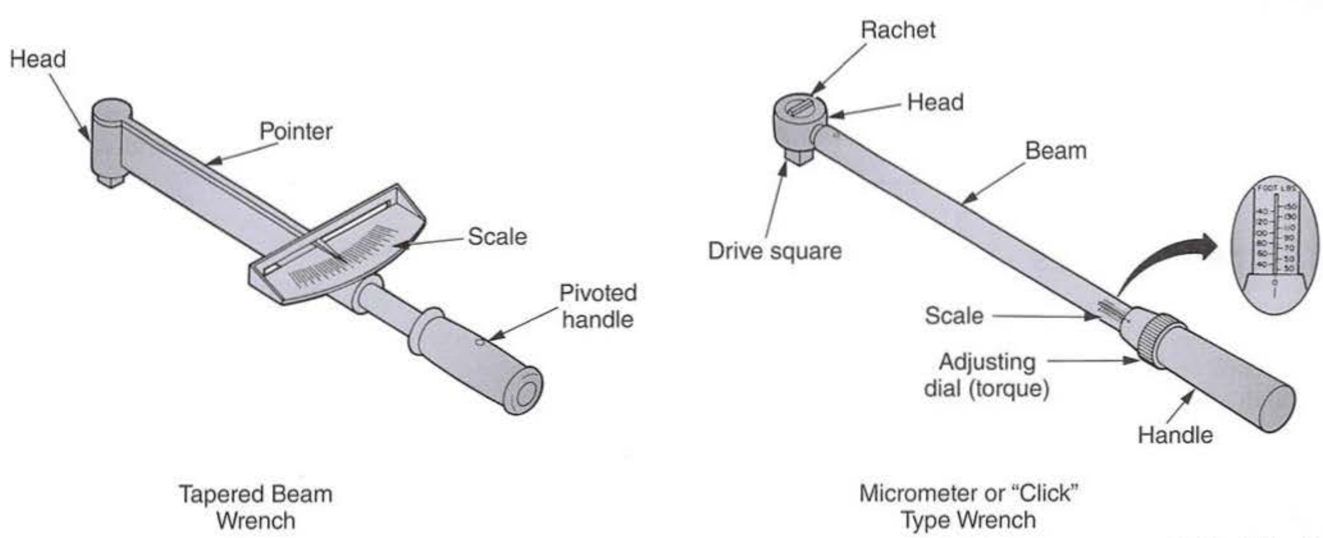


Goodheart-Willcox Publisher

Figure 7-19. The right and wrong ways to apply pressure to a torque wrench handle.

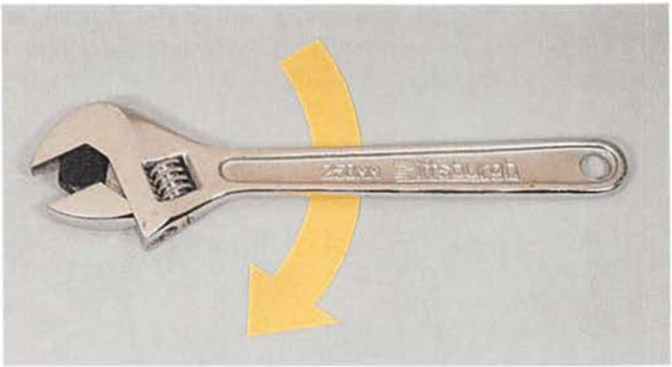
Goodheart-Willcox Publisher

Figure 7-20. An adjustable wrench is handy when a full wrench set is not available.



Goodheart-Willcox Publisher

Figure 7-18. Two types of torque wrenches.



Goodheart-Willcox Publisher

Figure 7-21. The movable jaw of the wrench should always face the direction of rotation.

Goodheart-Willcox Publisher

Figure 7-22. A wrench must fit the nut or bolt snugly.

SAFETY NOTE

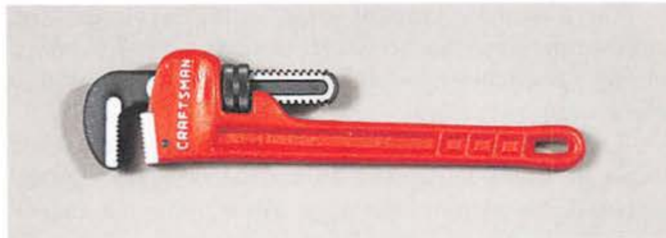
It is dangerous to push on, rather than pull, any wrench. If the fastener fails or loosens unexpectedly, you will almost always strike and injure your knuckles on the work. This operation is commonly known as *knuckle dusting.*

7.3.3 Pipe Wrenches

The pipe wrench is designed to grip round stock, **Figure 7-23**. However, the jaws will 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.

7.3.4 Open-End Wrenches

Open-end wrenches are usually double-ended, with two different size openings, **Figure 7-24**. They are made about 0.005″ (0.13 mm) oversize to permit them to slip easily onto 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.



Goodheart-Willcox Publisher

Figure 7-23. The pipe wrench has jaws that will grasp round objects.

Goodheart-Willcox Publisher

Figure 7-24. An open-end wrench is acceptable when the torque applied is low.

7.3.5 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 7-25**. A properly fitted box wrench will not normally slip. Box wrenches are available in the same sizes as open-end wrenches and with straight and offset handles.

Goodheart-Willcox Publisher

Figure 7-25. A box wrench can handle more torque than an open-end wrench.

7.3.6 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. Both ends are the same size. These wrenches are made in standard and metric sizes, **Figure 7-26**.

7.3.7 Socket Wrenches

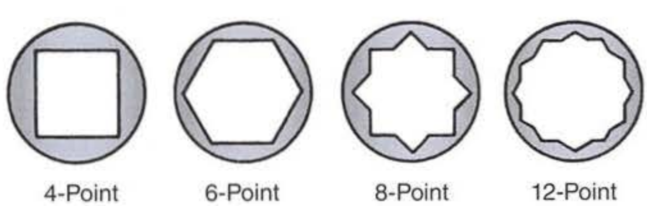
Socket wrenches are like box wrenches and are made with a tool head-socket (opening) that fits many types of handles (either solid bar or ratchet-type). A typical socket wrench set contains various handles and a wide range of socket sizes, **Figure 7-27**. Many sets include both standard and metric sockets. Various types of socket openings are shown in **Figure 7-28**.

Goodheart-Willcox Publisher

Figure 7-26. Combination wrenches have an open-end wrench at one end and a box wrench at the other.

Goodheart-Willcox Publisher

Figure 7-27. A typical socket wrench and sockets. The wrench has a right- and left-hand ratchet mechanism.



Goodheart-Willcox Publisher

Figure 7-28. Types of socket openings available. The 12-point socket can be used with both square and hex head fasteners.

7.3.8 Spanner Wrenches

Spanner wrenches have drive lugs and are designed to turn flush and recessed threaded fittings. The fittings have slots or holes to receive the wrench end. They are usually furnished with machine tools and attachments. See **Figure 7-29**.

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 by pins that fit into holes on the fitting, rather than into notches.

7.3.9 Allen Wrenches

The wrench that is used with socket-head fasteners is commonly known as an *Allen wrench*, **Figure 7-30**. It is manufactured in many sizes to fit fasteners of various standard and metric dimensions.

7.3.10 Wrench Safety

When using a wrench, keep the following safety precautions in mind:

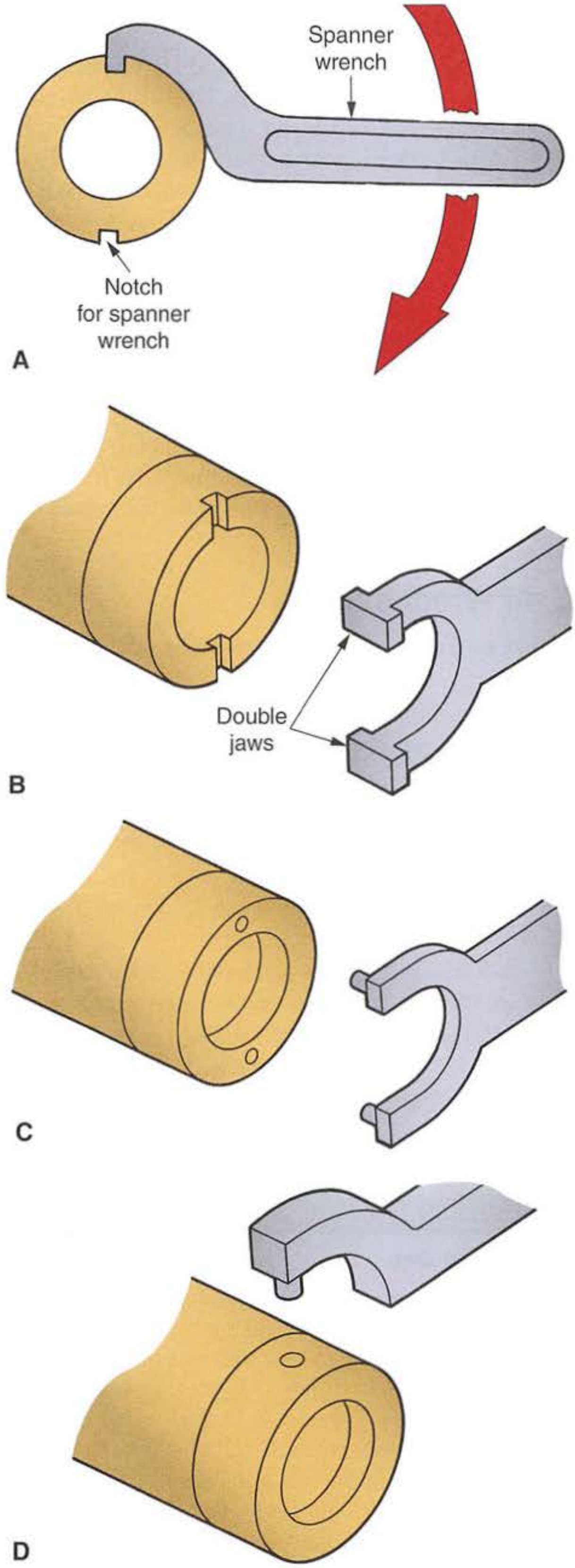
- Always pull on a wrench; never push. You have more control over the tool and there is less chance of injury.
- Select a wrench that fits properly. A loose-fitting wrench, or one with worn jaws, may slip and cause injury. It can also round off and ruin the bolt or nut on which it is being used.
- Never hammer on a wrench to loosen a stubborn fastener.
- Do not lengthen a wrench handle for additional leverage; this is a dangerous practice. Use a larger wrench.
- Clean any grease or oil off the handle and the floor in the work area before using a wrench. This will reduce the possibility of your hands or feet slipping.
- Never try to use a wrench on moving machinery.

GREEN MACHINING

Recycling Steel

Steel is the most recycled product in the world. The rate of recycling is determined by comparing how much steel is produced each year with how much steel is recycled. The US rate of recycling steel is around 85%. In fact, more steel is recycled in the United States each year than all other products combined! In addition to its high recycling rate, steel also offers eco-friendly benefits throughout its life cycle. For example, wastewater and waste material from steel manufacturing sites can be recovered and reused in other applications.

You can minimize your own environmental footprint by contributing to steel recycling programs. When your tools or other steel products wear out, collect and recycle them.



Goodheart-Willcox Publisher

Figure 7-29. 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.

Goodheart-Willcox Publisher

Figure 7-30. Allen wrenches are used with socket-head fasteners. They are made in both inch and metric sizes.

7.4 Screwdrivers

Screwdrivers are manufactured with many different tip shapes, **Figure 7-31**. Each shape is designed for a particular type of fastener. Most shop workers are familiar with the standard and Phillips screwdrivers. The other shapes may not be as well-known.

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 to the length. Screwdriver length is measured from the tip of the blade 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 7-32**. The conventional straight-shank screwdriver is used for a variety of work. The electrician's screwdriver has an insulated handle and a long, thin blade that can reach into tight areas. A heavy-duty screwdriver has a thick, square shank that permits a wrench to be applied to drive or remove large or stubborn screws. The stubby or close-quarters screwdriver is designed for use where workspace 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. The double-end offset screwdriver can be used where there is not enough space for a conventional straight-shank tool.

The Phillips screwdriver has a +-shaped tip for use with Phillips recessed head screws. They are manufactured in the same general styles as the standard screwdriver and are available in four sizes: #1, #2, #3, and #4.

The Pozidriv® screwdriver tip is similar in appearance to the Phillips tip but has a slightly different shape. This type has been designed for the Pozidriv® screws that are used extensively in the aircraft, automotive, electronic, and appliance industries. The tip of this screwdriver has a black oxide finish to distinguish it from the Phillips tool. Using a Phillips

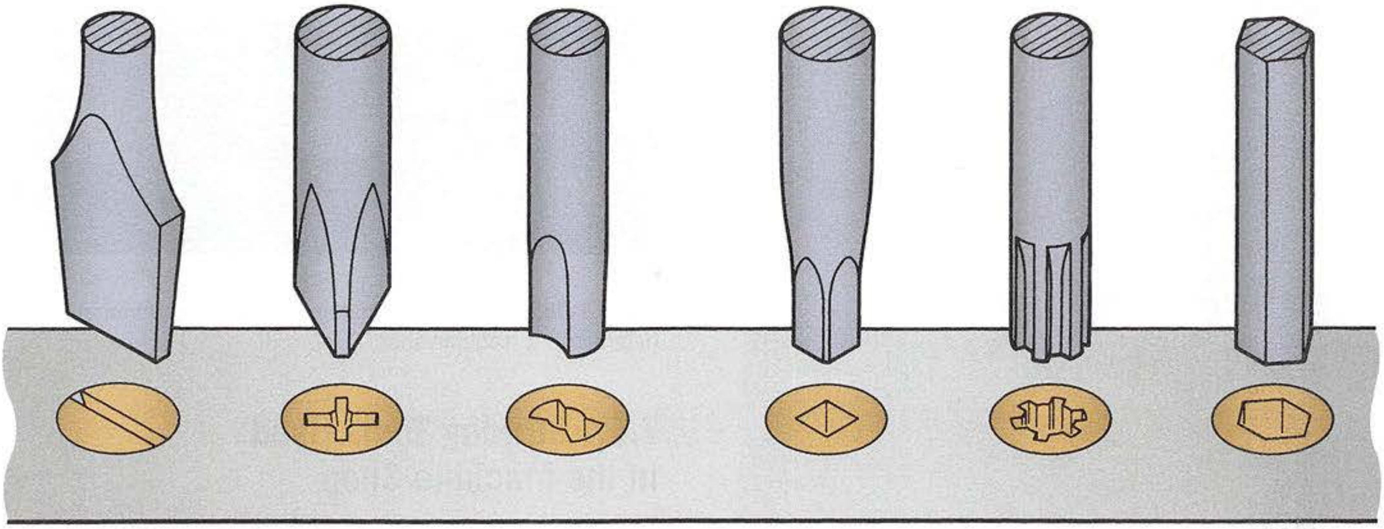


Figure 7-31. Types of screwdriver tips.

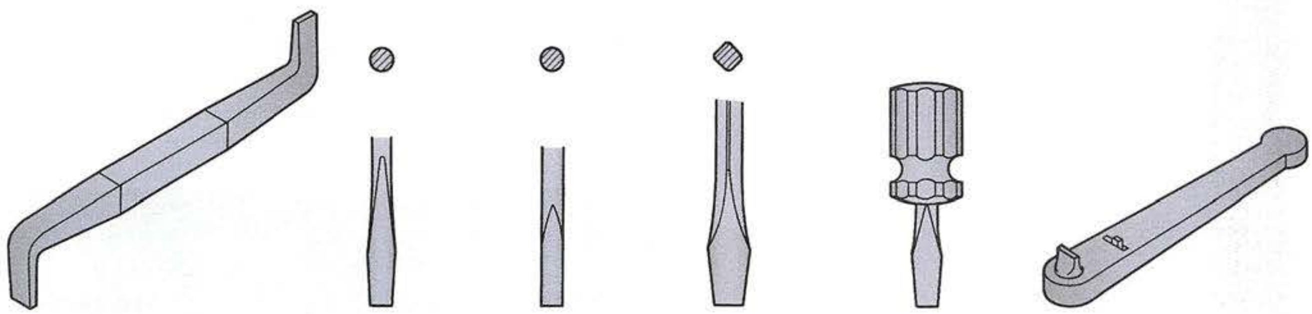


Figure 7-32. Styles and types of standard screwdrivers.

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.

7.4.1 Using a Screwdriver

Always select the correct size screwdriver for the screw to be driven, **Figure 7-33**. A poor fit can damage the screw slot and often damages the tool's tip. Damaged screw heads are dangerous, and they are often difficult to drive or remove. They should be replaced.

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 those shown on the right in **Figure 7-34**, must be reground using a fine grinding wheel and light pressure. 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.

7.4.2 Screwdriver Safety

When using a screwdriver, keep the following safety precautions in mind:

- Use a screwdriver only for its intended purpose. 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.
- Replace screws with burred heads, or remove the burrs with a file or abrasive cloth. Burred screws are dangerous.
- For electrical work, use a screwdriver with an insulated handle specifically designed for this purpose. Always turn electric power off before working on electrical equipment.

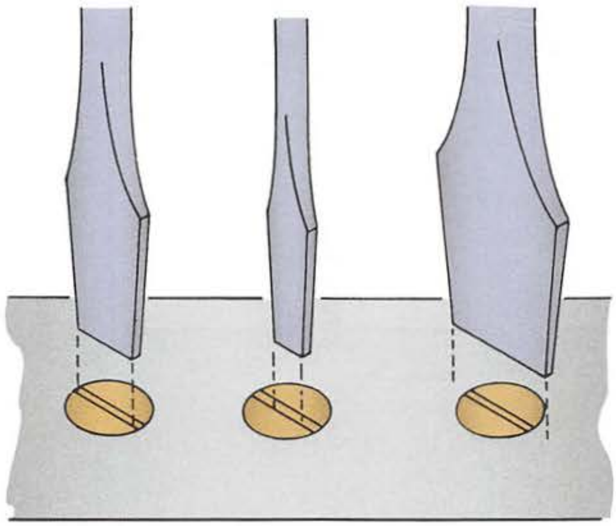


Figure 7-33. Use the correct screwdriver tip for the job.

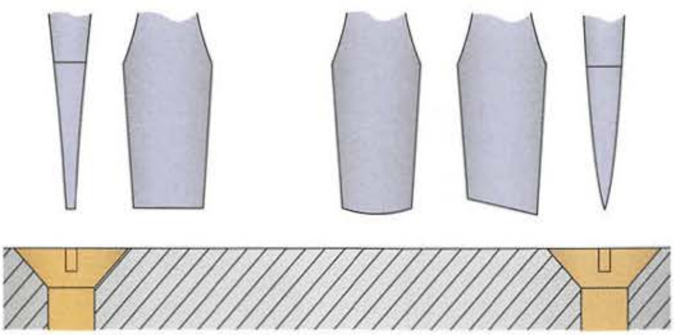


Figure 7-34. Avoid using screwdrivers with tips like those on the right. They are worn or improperly sharpened. The tip at left is ground correctly. The sides are concave to help hold the tip in the slot when pressure is applied.

- Avoid carrying a screwdriver in your pocket. It is a dangerous practice that can cause injury to you or to someone else.

7.5 Striking Tools

The machinist's ball-peen hammer, **Figure 7-35**, 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 7-35. The ball-peen hammer is the most commonly used hammer in a machine shop.

7.5.1 Striking Tools Used in the Machine Shop

Ball-peen hammers 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 allows 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 7-36.**

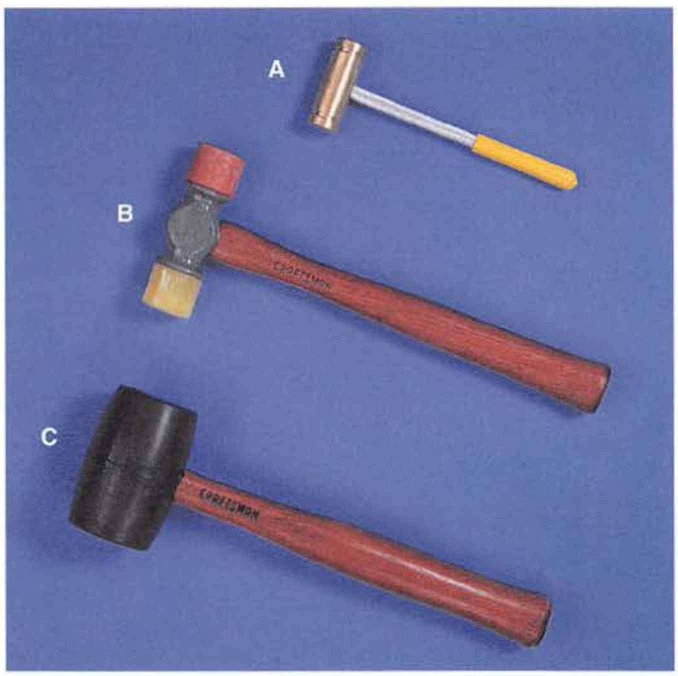


Figure 7-36. Soft-face hammers and mallets. A—Brass hammer. B—Plastic-face hammer. C—Rubber mallet.

7.5.2 Striking Tool Safety

When using a hammer, keep the following safety precautions in mind:

- 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 hold the handle too close to the head 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.

7.6 Chisels

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

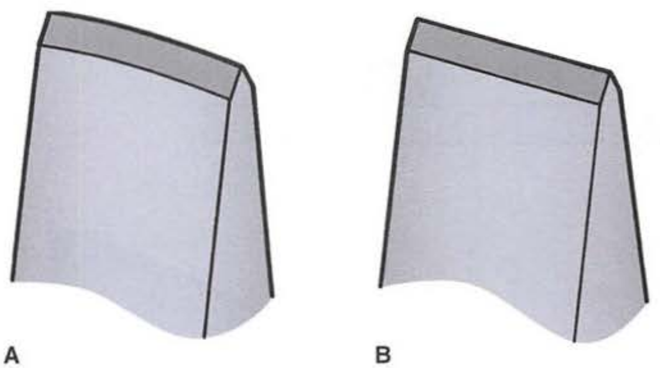
7.6.1 Cutting

The chisel is used mostly to cut cold metal, hence the term *cold chisel*. The four chisels illustrated in **Figure 7-37** are the most common types. Other chisels in this category are variations or combinations of these chisels.

The work to be cut determines how the chisel should be sharpened, **Figure 7-38**. A chisel with a slightly curved cutting edge works better for cutting on a flat plate. The curved edge helps prevent the chisel from cutting unwanted grooves in the surrounding metal, as when shearing rivet heads. If the chisel will be used to shear metal held in a vise, the cutting edge should be straight.

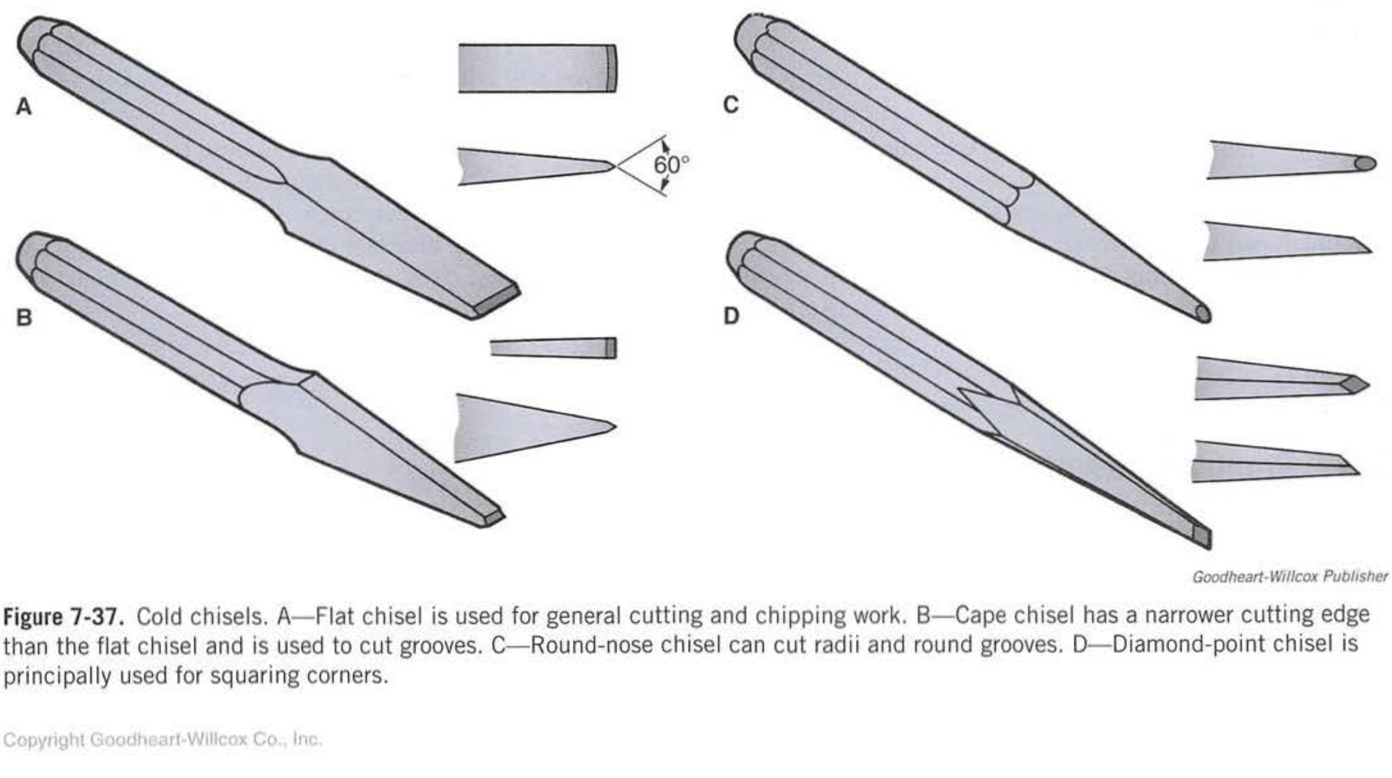
The chisel is frequently used to chip surplus metal from castings. Chipping is started by holding the chisel at an angle, as shown in **Figure 7-39A**. The angle must be great enough to cause the cutting edge to enter the metal.

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, **Figure 7-39B**. Cut depth can be reduced by decreasing the chisel angle. However, if the cutting angle is decreased too much, the



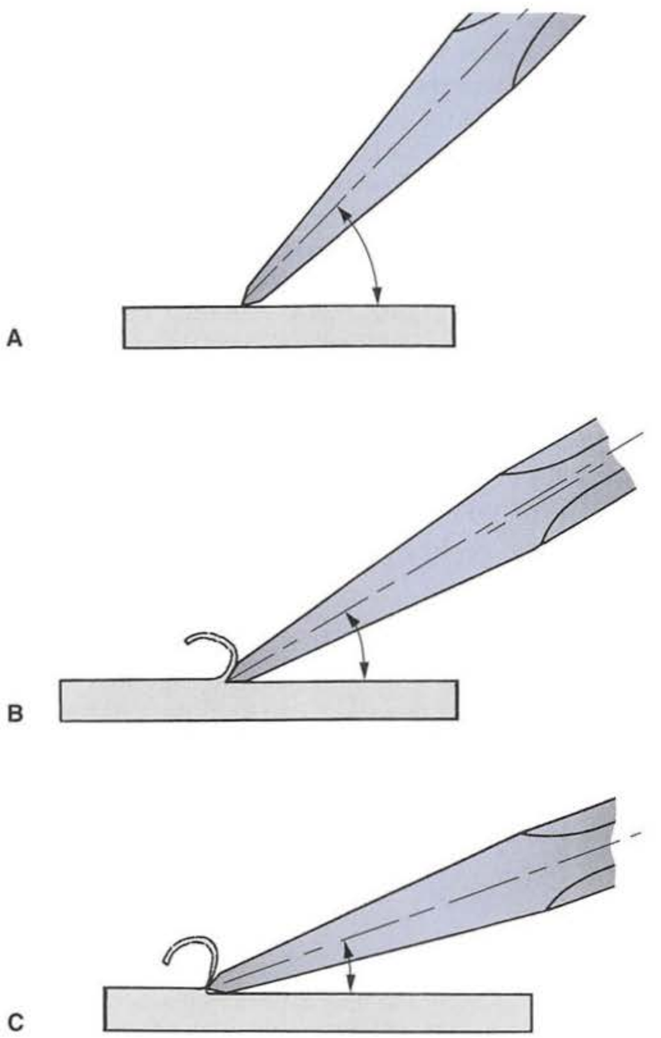
Goodheart-Willcox Publisher

Figure 7-38. The work to be done determines how a chisel should be sharpened. A—Slightly rounded edge for cutting on flat plate. B—Straight edge for shearing.



Goodheart-Willcox Publisher

Figure 7-37. 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.



Goodheart-Willcox Publisher

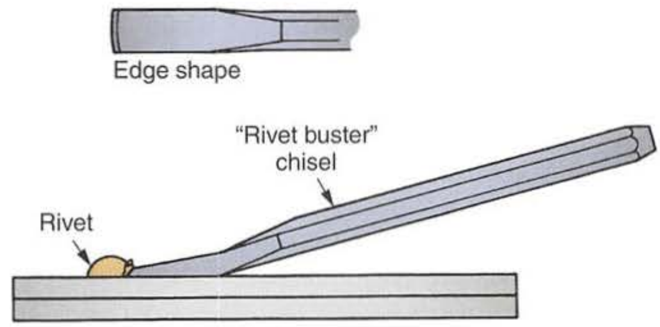
Figure 7-39. Proper chisel angles for various cutting operations. A—Starting the cut. B—Maintaining the cut at the desired depth. C—Reducing the cutting angle too much causes the chisel to lift out of the cut.

chisel will ride on the heel of the cutting edge and lift out of the cut, **Figure 7-39C**.

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.

7.6.2 Removing Rivets

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 7-40**.



Goodheart-Willcox Publisher

Figure 7-40. This variation of the flat chisel is often referred to as a rivet buster. The 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 7-41A**. 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 7-41B** shows how this is done. Rivets can also be removed by drilling and using the narrow cape chisel, **Figure 7-41C**.

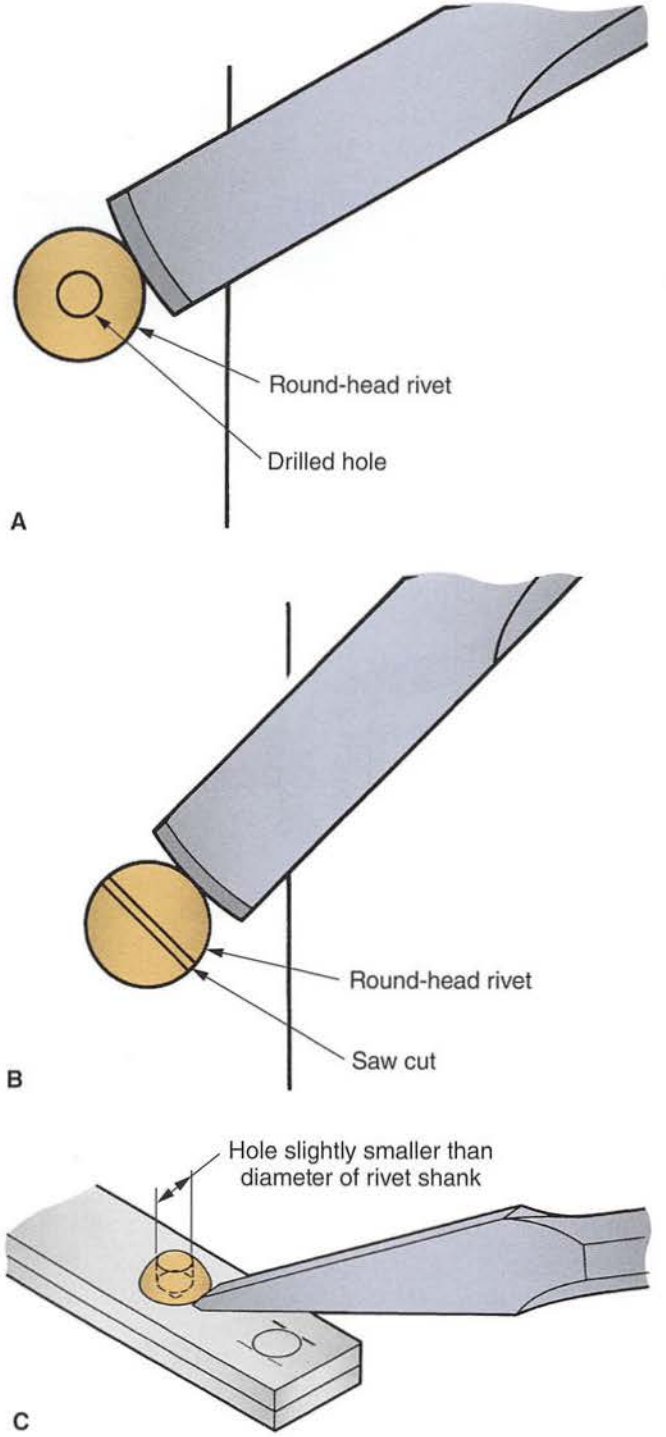
7.6.3 Chisel Safety

When using a chisel, keep the following safety precautions in mind:

- 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. Flying chips are dangerous.
- Hold a chisel so that if you miss it with the hammer, you will not strike and injure your hand. Use a chisel holder if one is available.
- Remove a mushroomed chisel head by grinding away the excess metal. See **Figure 7-42**. A mushroomed chisel head is extremely dangerous, since jagged metal can be knocked or chipped off and cause serious injury. This mushroom effect can occur after repeated use with a hammer.
- Edges on metal cut with a chisel are sharp and can cause bad cuts. Remove them by grinding or filing.

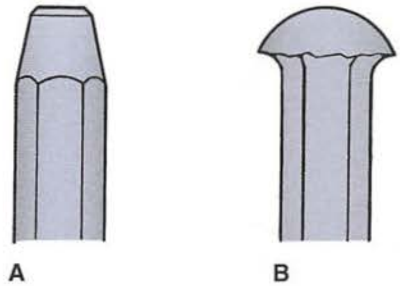
7.7 Hacksaw

The typical hacksaw is composed of a frame with a handle and a replaceable blade, **Figure 7-43**. Almost all hacksaws made today are adjustable to accommodate several different blade



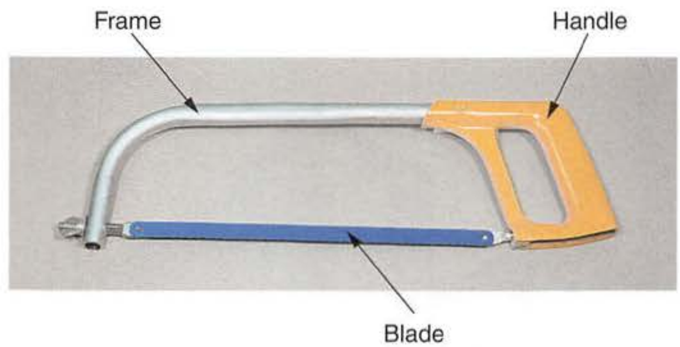
Goodheart-Willcox Publisher

Figure 7-41. Alternate methods for removing rivet heads. A—When there is not enough room to swing the hammer with sufficient force. B—When the rivet head is too large to be removed as one piece. C—A cape chisel may also be used to remove rivets.



Goodheart-Willcox Publisher

Figure 7-42. Chisel safety. A—Chisel head ground to a safe condition. B—A dangerous mushroomed head.



Goodheart-Willcox Publisher

Figure 7-43. A typical hacksaw.

lengths. They are also made so the blade can be installed in either a vertical or horizontal position, **Figure 7-44.**

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.

The hacksaw blade must be positioned with the teeth pointing away from the handle, **Figure 7-45.** This will make it cut on the forward (push) stroke.

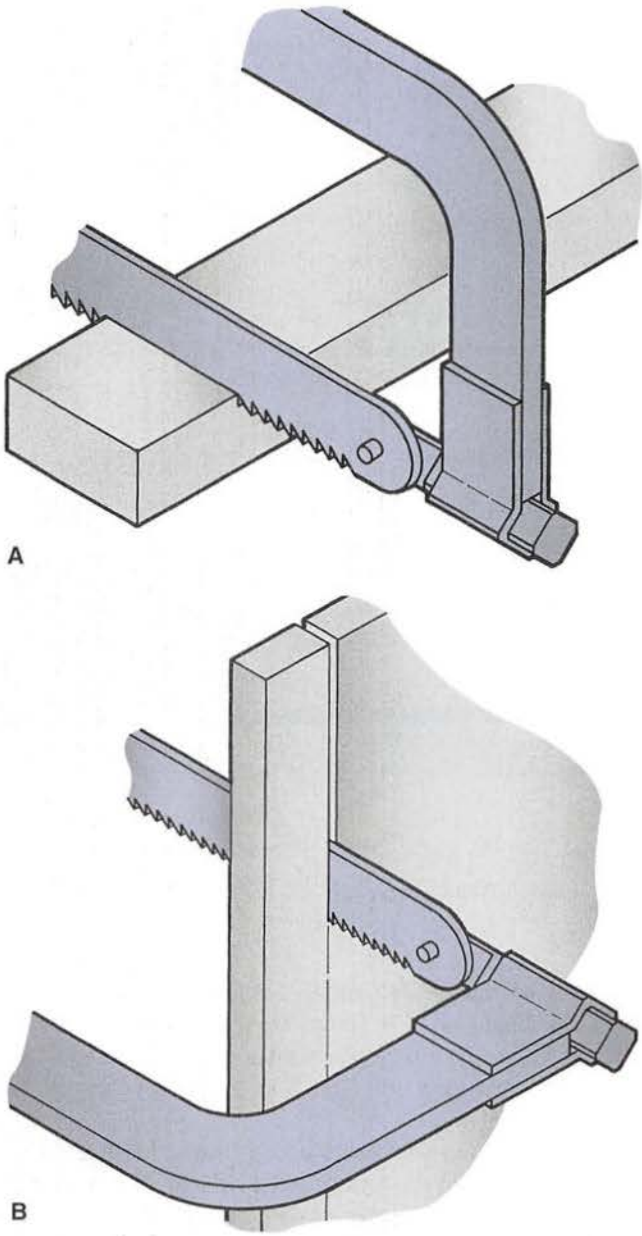
7.7.1 Holding Work for Sawing

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

Figure 7-46 shows some preferred methods for holding work that is irregular in 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.

7.7.2 Starting a Cut

Before starting a cut to a marked line, it is best to notch the work with a file, **Figure 7-47.** You can also use the thumb of your left hand to guide the blade until it starts the cut, **Figure 7-48.** Work carefully to avoid injury. Some hacksaw blades are manufactured with very fine teeth at the front

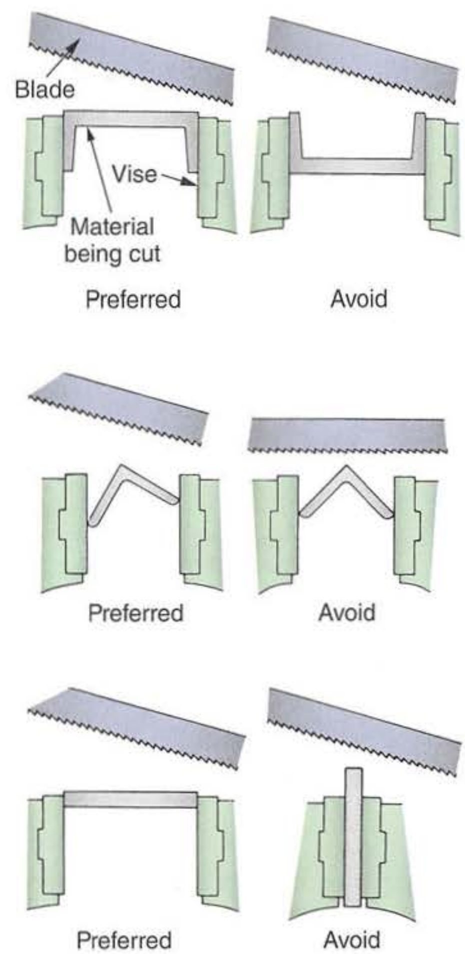


Goodheart-Willcox Publisher

Figure 7-44. Blade positions. A—The blade is set to cut in a conventional vertical position. B—The blade has been pivoted 90° to cut a long, narrow strip of stock.

Goodheart-Willcox Publisher

Figure 7-45. 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.



Goodheart-Willcox Publisher

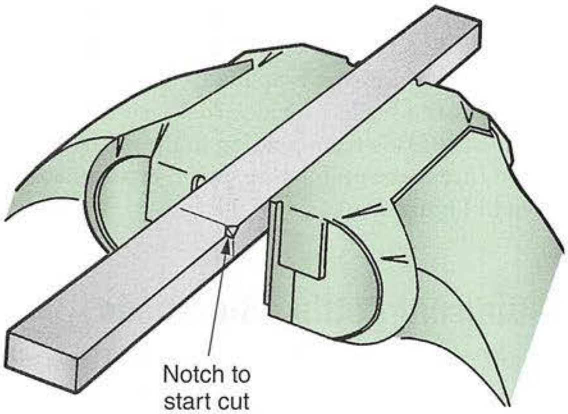
Figure 7-46. Preferred methods of holding irregular stock for sawing.

to make starting a cut easier. Use enough pressure that the blade begins to cut immediately.

7.7.3 Making the Cut

Grasp the hacksaw firmly by the handle and the front of the frame. Apply enough pressure on the forward stroke to make the teeth cut. Insufficient pressure permits the teeth to slide over the material, dulling the teeth. Lift the saw slightly on the return stroke to avoid dragging the teeth across the material.

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 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.



Goodheart-Willcox Publisher

Figure 7-47. Using a file to notch or nick the edge of a piece to be cut permits easier starting of the hacksaw cut.

7.7.4 Dull or Broken 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 becomes dull, the kerf (the slot made by the 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.

Goodheart-Willcox Publisher

Figure 7-48. Using your thumb to guide the hacksaw blade as the cut is started.

7.7.5 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 the stock from dropping when the cut is completed.

CAREER CONNECTION

Assembler/Fabricator

What does an assembler/fabricator do?

Assemblers and fabricators assemble parts and finished products in many different manufacturing industries. There are more than 1,800,000 assembly jobs in the United States, and these professionals assemble almost everything from massive aircraft to delicate electronic devices. They also conduct quality-control checks and may work closely with designers and engineers to refine and finalize designs as product moves from start to finish.

What education and skills are needed to be an assembler/fabricator?

For most assembly jobs, only a high school diploma is required, but qualifications vary by employer. Skilled assembly work often requires an associate's degree. More formal training in a technical school program or an apprenticeship is needed for industries such as aircraft or motor vehicle production.

Assembly/fabrication can also be physically demanding work with specific physical requirements. Assemblers/fabricators must have excellent color vision, dexterity, and physical strength and stamina.

What is it like to be an assembler/fabricator?

Many assemblers and fabricators work as team assemblers in manufacturing. Other professionals may specialize in assembling electrical and electronic equipment or structural metals. Wages vary by industry and range from $27,000 to $48,000 according to the *Occupational Outlook Handbook*.

In most industry areas, this work can be strenuous. Workers are at risk of exposure to hazardous noise levels, temperatures, machinery, and chemicals and fumes. Following safety regulations is critical for reducing workplace dangers.

7.7.6 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. Choose a blade that has an appropriate number of teeth per inch for the material and shape of the work, **Figure 7-49.** At least 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.

On a flexible back blade, only the teeth are hardened. An 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.

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 7-50.**

7.7.7 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 problem can be eliminated by inserting a wood dowel of the

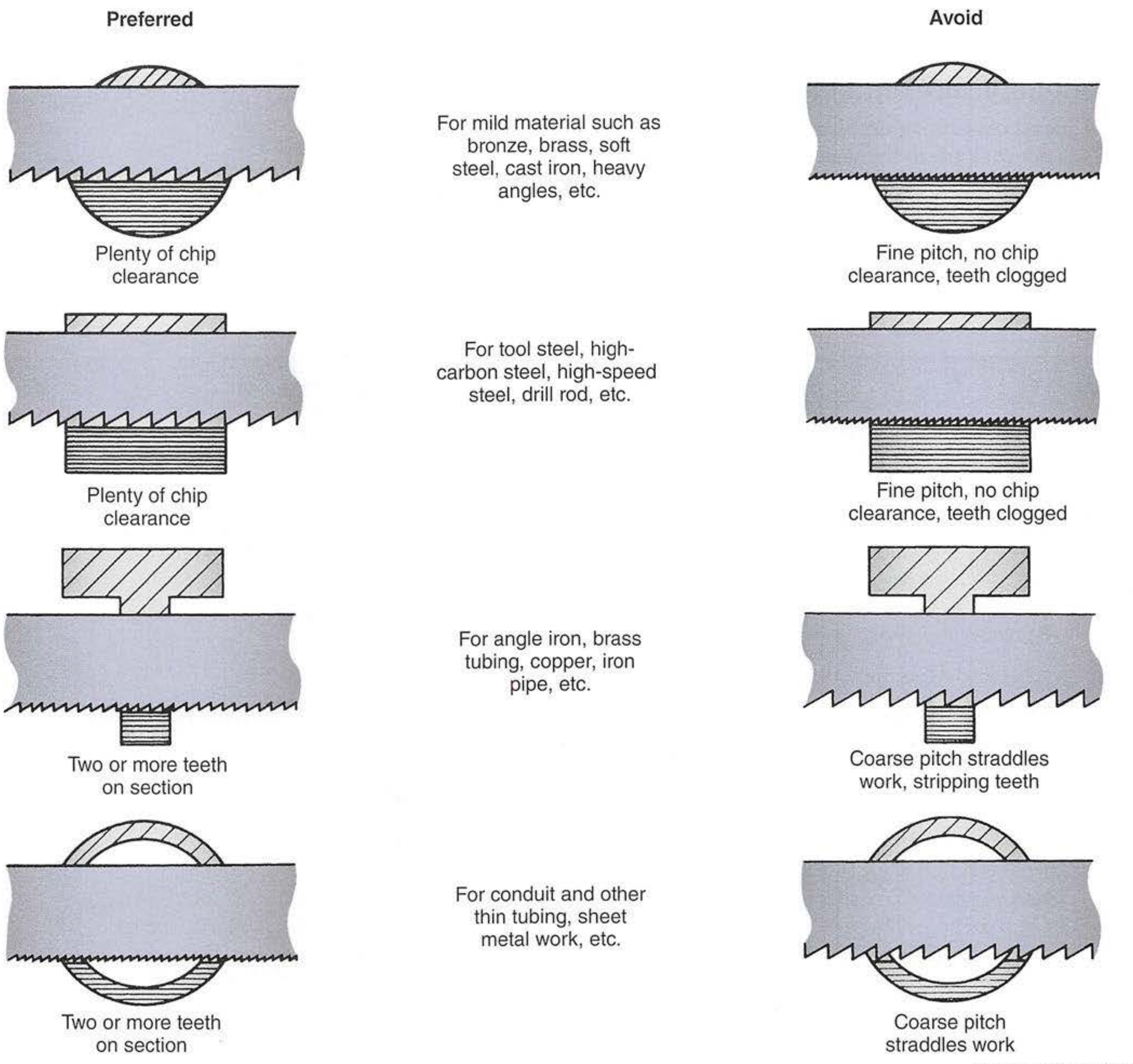


Figure 7-49. Use the proper hacksaw blade for each job to ensure long blade life and rapid cutting action.

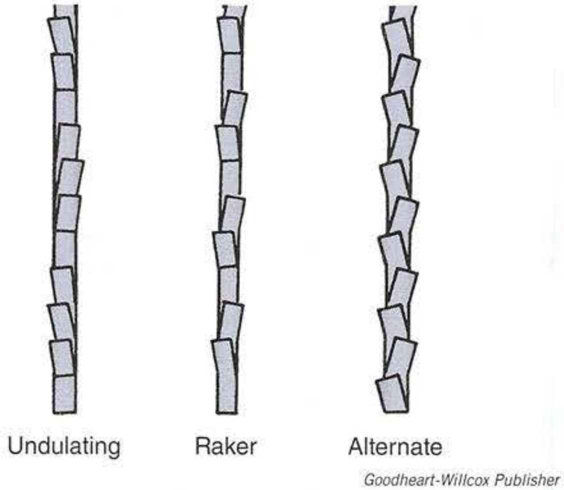


Figure 7-50. Types of sets in hacksaw teeth.

proper size into the tubing. Then cut through both tubing and dowel. See **Figure 7-51**.

To cut a long, narrow strip from thin metal, set the blade at right angles to the frame. Make the cut in the usual way, as shown in **Figure 7-52**. Strips of any width, up to the capacity of the saw frame, can be made in this manner. Thin metal can also be cut more easily and precisely by putting it between two pieces of wood, and cutting through both wood and metal, **Figure 7-53**.

7.7.8 Hacksaw Safety

When using a hacksaw, keep the following safety precautions in mind:

- 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.

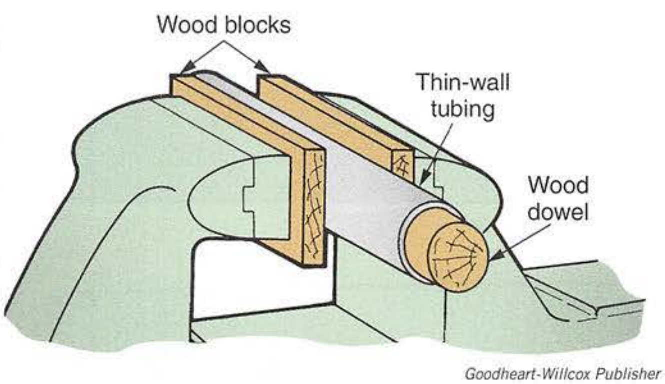


Figure 7-51. A snug-fitting dowel slid into thin-wall tubing makes cutting the tubing easier. If the tubing is to be held in a vise for cutting, place soft wood blocks between the vise jaws and the work to prevent marring the exterior surface of the tubing.

- Take care when handling cut metal pieces. Burrs formed on the cut edge 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.

7.8 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. The number of different kinds, shapes, and cuts of files

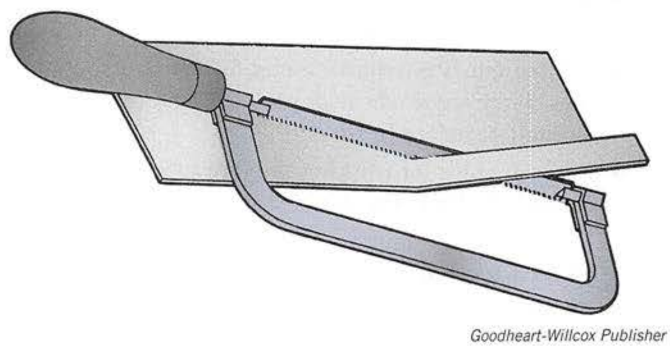


Figure 7-52. The hacksaw blade can be pivoted to a horizontal position for cutting long, narrow strips. The best results are obtained if the strip is bent up slightly, as shown, during the sawing operation.

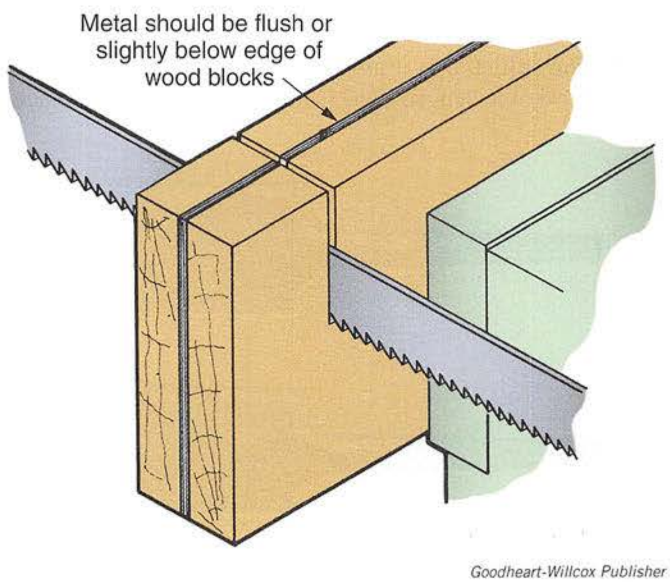


Figure 7-53. Sandwiching thin metal between two pieces of wood makes sawing easier and more precise.

available is almost unlimited. For this reason, only the general classifications of files are covered in this chapter.

7.8.1 File Classifications

Files can be classified according to their shape, type, and length. The shape is the general outline and cross section. The outline is either tapered or blunt, **Figure 7-54**. Of the many file shapes available, the most commonly used are flat, pillar, square, 3-square, knife, half-round, crossing, and round, **Figure 7-55**. Each shape is available in many sizes and degrees of coarseness: rough, coarse, bastard, second-cut, smooth, and dead smooth. Because file coarseness is related to length, **Figure 7-56**, a small (4″) rough-cut file may be as fine as a large (16″) second-cut file.

File length is always measured from the heel to the point, **Figure 7-57**. The tang is not included in the measurement.

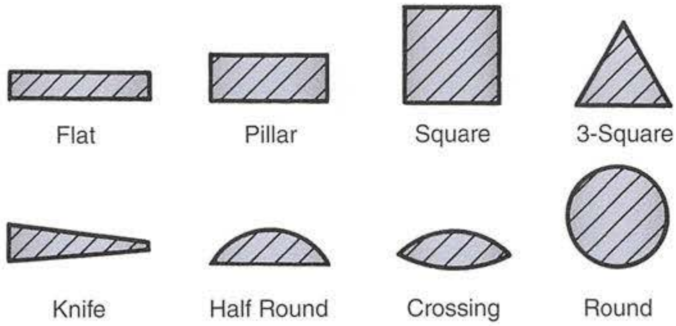
The file cut indicates the relative coarseness of the teeth (**Figure 7-58**):

- Single-cut files are usually used to produce a smooth surface finish. They require only light pressure to cut.
- Double-cut files remove metal faster than single-cut files. They require heavier pressure, and they produce a rougher surface finish.
- Rasps are best for working wood or other soft materials where a large amount of stock must be removed in a hurry.
- Curved-tooth files are used to file flat surfaces of aluminum and sheet steel.

7.8.2 Types of Files

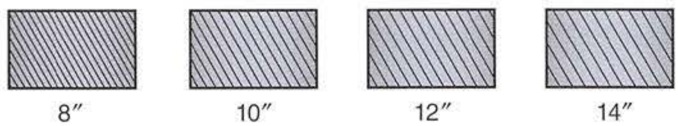
Files can be divided into five general groups:

- The machinist's 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. 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.
- Special purpose files, including those specifically designed to cut one type of metal, are for one kind of operation. An example is the long-angle lathe file, which does an efficient filing job on the lathe.



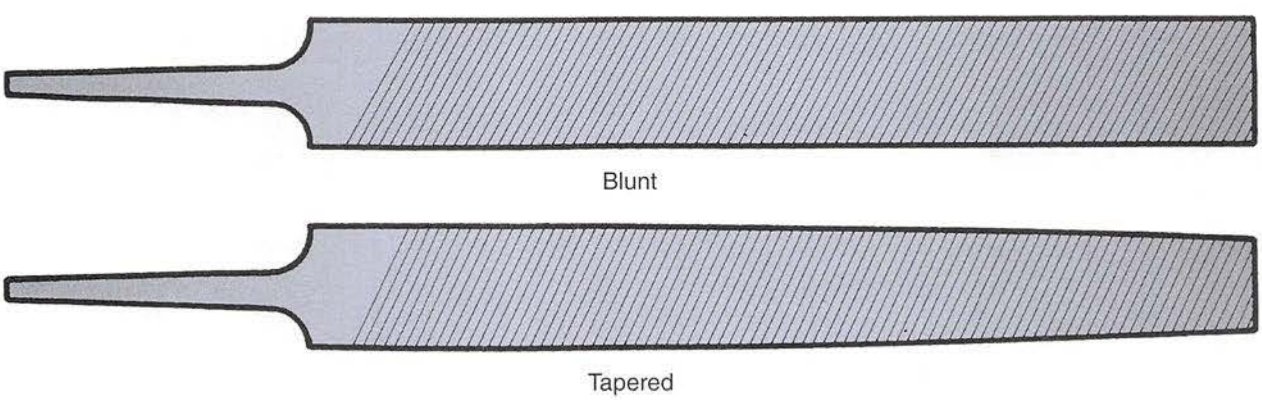
Goodheart-Willcox Publisher

Figure 7-55. Cross-sectional views of the most widely used file shapes.



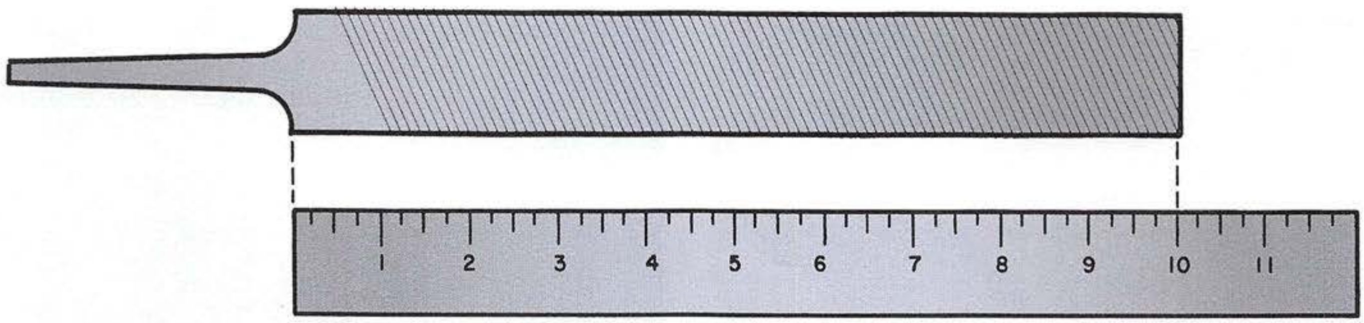
Goodheart-Willcox Publisher

Figure 7-56. File coarseness is directly related to file length. The longer the file, the coarser the teeth.



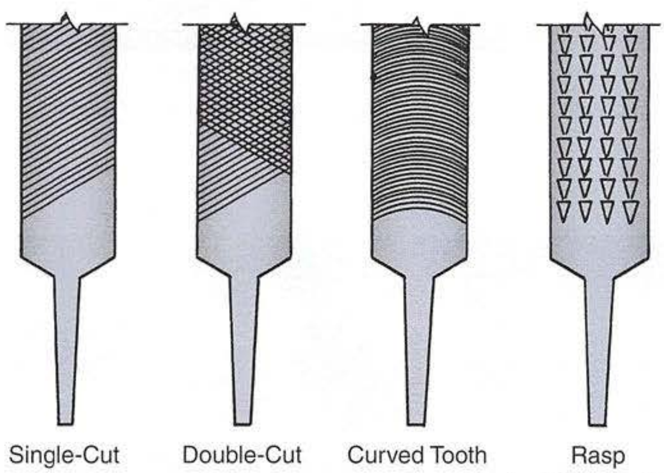
Goodheart-Willcox Publisher

Figure 7-54. File shape classifications.



Goodheart-Willcox Publisher

Figure 7-57. How a file is measured.



Goodheart-Willcox Publisher

Figure 7-58. File cut classifications.

7.8.3 File Care

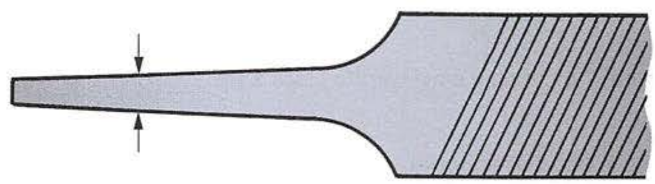
SAFETY NOTE

Never use a file without a handle. It is too easy to drive the unprotected tang into your hand.

A handle can be fit to the file by drilling a hole in the handle. The hole should be equal in diameter to the width of the tang at its midpoint, **Figure 7-59**. Mate the file and handle by inserting the tang in the hole, then sharply striking the handle on a solid surface.

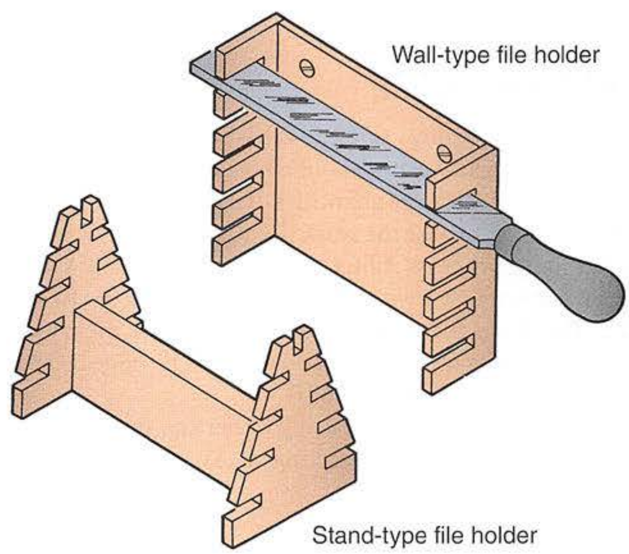
Store files so that they are always separated, **Figure 7-60**. Never throw files together in a drawer or store them in a damp place. Clean files frequently with a file card or brush, **Figure 7-61**.

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.



Goodheart-Willcox Publisher

Figure 7-59. The hole in the file handle should be equal in diameter to the width of the file tang at the point indicated.



Goodheart-Willcox Publisher

Figure 7-60. Storing files properly, in holders like these, will greatly extend their useful life.

7.8.4 Selecting a File

Choosing the correct file for the job helps ensure maximum cutting efficiency. Several factors must be considered in order to select an appropriate file:

- The nature of the work (flat, concave, convex, notched).

Goodheart-Willcox Publisher

Figure 7-61. Using a file card to clean a file.

Goodheart-Willcox Publisher

Figure 7-63. Mount work at elbow height for general filing.

- Kind of material.
- Amount of material to be removed.
- Surface finish and accuracy required.

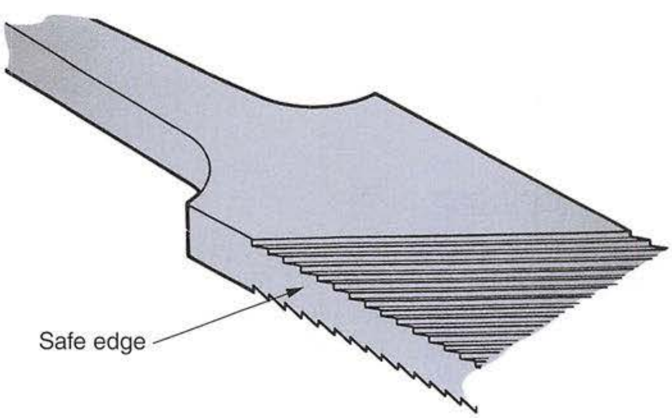
Some files have ***safe edges***. This denotes that the file has one or both edges without teeth, **Figure 7-62**. This permits filing corners without danger to the portion of the work that is not to be filed.

7.8.5 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 7-63**. If large quantities of metal must be removed by heavy filing, mount the work slightly lower.

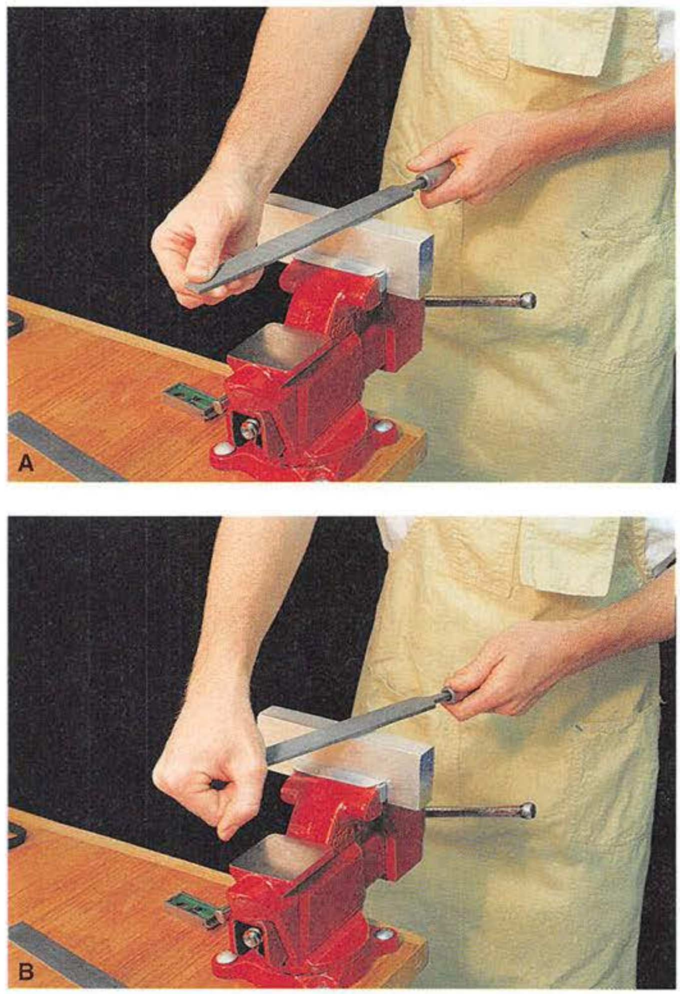
Straight or cross filing consists of pushing the file lengthwise across the work, either straight ahead or at a slight angle. Grasp the file as shown in **Figure 7-64A**. Heavy-duty filing requires heavy pressure and can best be done if the file is held as shown in **Figure 7-64B**.

A file can be ruined by using either too much pressure or too little pressure on the cutting stroke. Apply just enough



Goodheart-Willcox Publisher

Figure 7-62. The safe edge of a file does not have teeth.



Goodheart-Willcox Publisher

Figure 7-64. 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.

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.

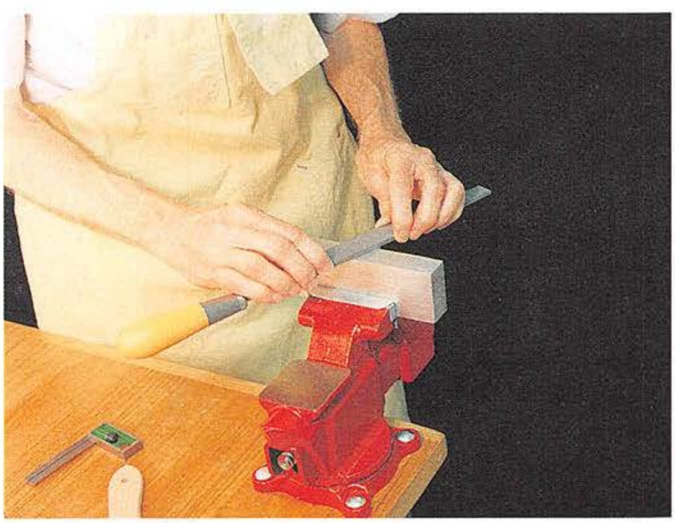
Lift the file from the work on the reverse stroke, except when filing soft metal. The pressure on the return stroke when filing soft metal should be no more than the weight of the file itself.

Draw filing, when properly done, produces a finer finish than straight filing. Hold the file as shown in **Figure 7-65.** 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 use a single-cut file to produce the final finish.

7.8.6 File Safety

When using a file, keep the following safety precautions in mind:

- 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 an infection.
- Do not clean a file by slapping it on the bench; this may cause the file to 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.



Goodheart-Willcox Publisher

Figure 7-65. Draw filing, when done properly, improves the surface finish.

7.9 Reamers

A drill does not produce a smooth or accurate enough hole for a precision fit. A ***reamer*** is used to enlarge, smooth, and size a drilled hole by removing a small amount of metal. It produces better smoothness and accuracy than a drill. Ordinarily, hand reaming is used only for final sizing of a hole.

7.9.1 Hand Reamer

A hand reamer has a square shank end so that it can be held in a tap wrench. 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. See **Figure 7-66A.**

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. 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.

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.

An expansion hand reamer is used when a hole must be cut a few thousandths inch over nominal size for fitting purposes, **Figure 7-66B.** 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.

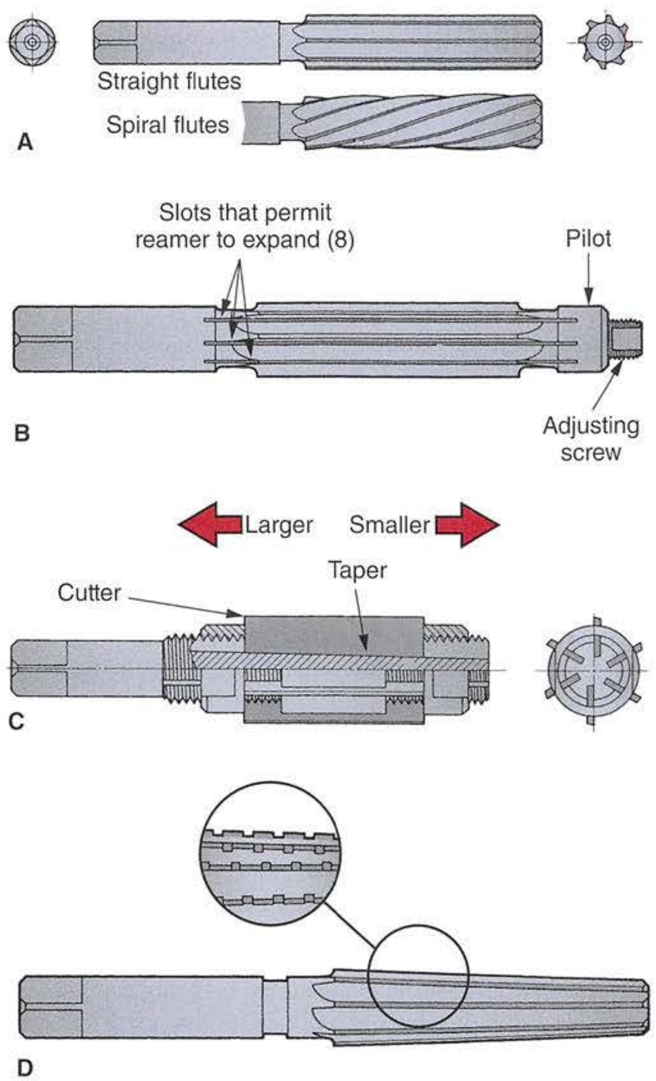
The entire length of an adjustable hand reamer is threaded, and it is fitted with tapered slots to receive the adjustable blades, **Figure 7-66C.** The blades are tapered along one edge to correspond with the taper slots in the reamer body so that the cutting edges of the blades remain parallel.

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, **Figure 7-66D,** can finish a tapered hole accurately and with a smooth finish for taper pins. However, because of their long cutting edges, taper reamers are somewhat difficult to operate.

7.9.2 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

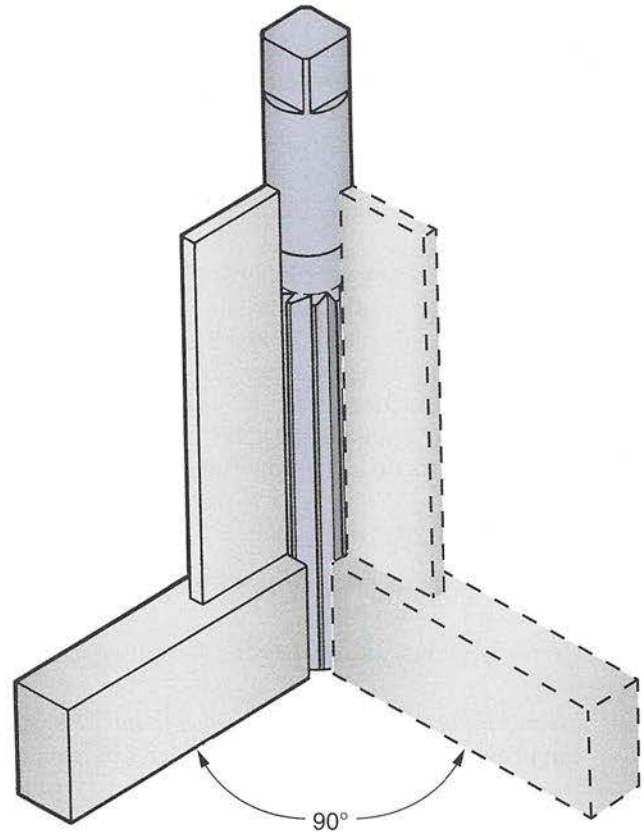


Goodheart-Willcox Publisher

Figure 7-66. Hand reamers. A—Straight flute and spiral flute solid reamers are used for different applications. B—Adjustments for an expansion hand reamer. 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.

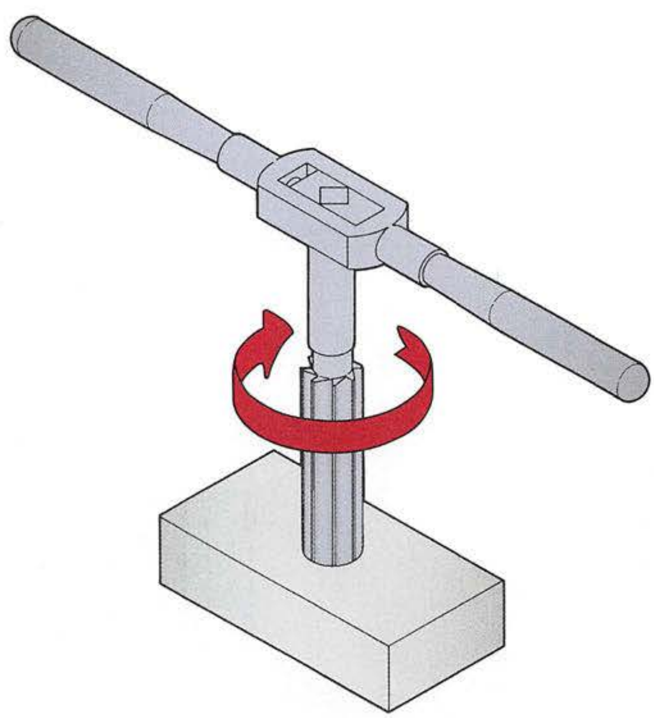
virtually impossible to secure a satisfactory hole using an adjustable wrench to turn the reamer. Follow these guidelines to use a hand reamer:

- To start reaming, rotate the tool slowly to allow it to align with the hole. Check several points around the reamer's circumference to make sure that the reamer has started square with the work, **Figure 7-67**.
- 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. Rotation should be steady and rapid.
- Apply turning pressure evenly with both hands, always in a clockwise direction, **Figure 7-68**. Never turn a



Goodheart-Willcox Publisher

Figure 7-67. Always make sure that the reamer is square with the work.



Goodheart-Willcox Publisher

Figure 7-68. Always turn a hand reamer in a clockwise direction.

reamer in a counterclockwise direction, because 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 depends on the metal being reamed.

7.9.3 Reaming Safety

When using a hand reamer, keep the following safety precautions in mind:

- 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.
- 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.

7.10 Hand Threading

Threaded sections have many applications in 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 as shown in **Figure 7-69**. They are specified by diameter and number of threads per inch. Metric threads are specified by diameter, and thread pitch is given in millimeters.

The ***American National Thread System*** was adopted in 1911. It is the most 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, NATO (the North Atlantic Treaty Organization) adopted a standard thread form. It is referred to as the ***Unified System***, **Figure 7-70**. 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 (Unified National Fine) and UNC (Unified National Coarse), **Figure 7-71**. Fasteners using this thread series are interchangeable with fasteners using the American National thread.

Several other thread groups exist in addition to those just described. 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″.

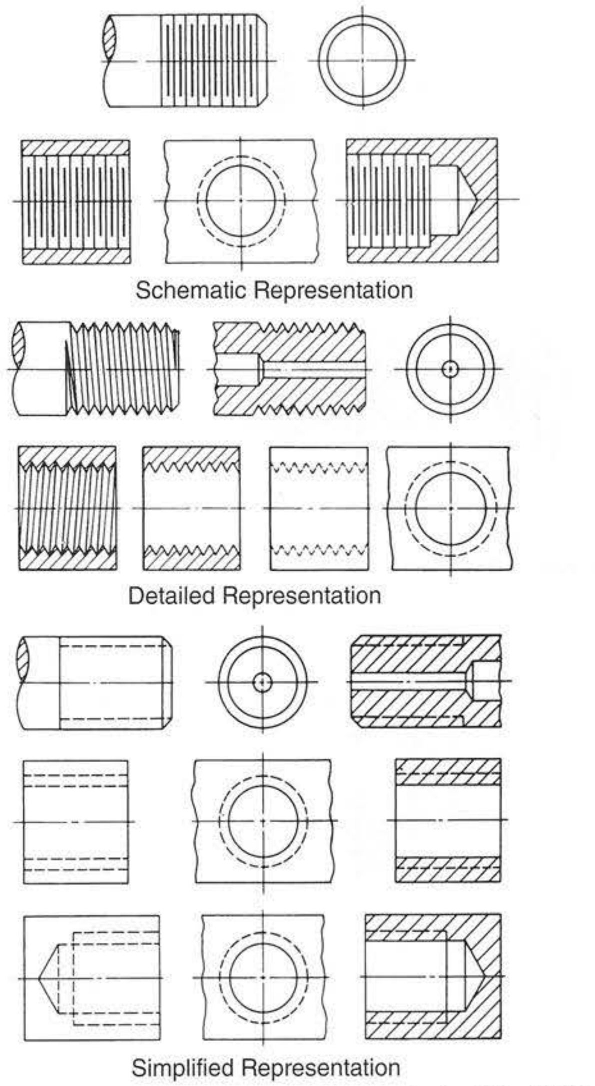


Figure 7-69. Methods used to depict threads on drawings. Only one type is used on a given drawing.

Metric unit threads have the same shape as Unified threads, but are specified in a different manner, **Figure 7-72**. Metric threads and Unified threads are not interchangeable. See **Figure 7-73**.

7.10.1 Thread Size

Threads of the Unified 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.

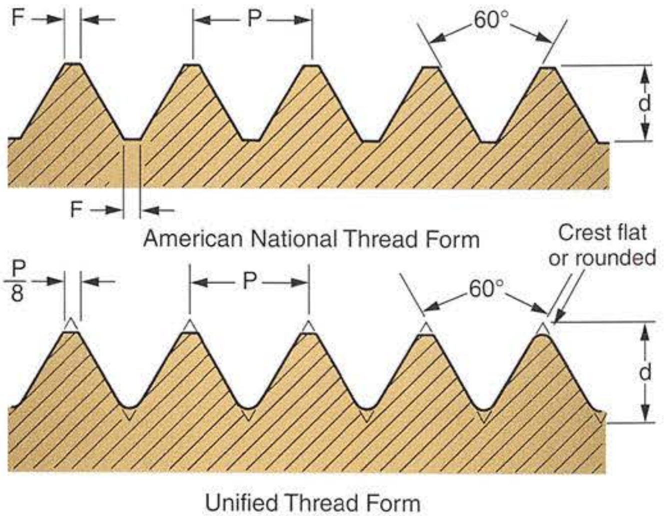


Figure 7-70. Similarities and differences of the American National thread form and the Unified thread form.

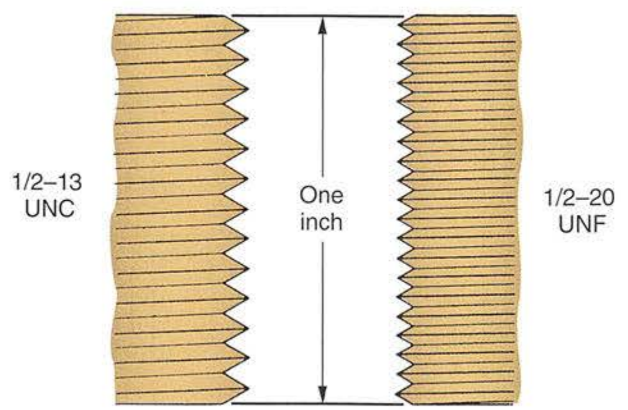


Figure 7-71. Comparison of Unified National Coarse (UNC) and Unified National Fine (UNF) threads. Both have the same geometric shape.

Be careful not to mistake the number denoting the thread diameter and the number of threads for a fraction. For example: a #8-32 UNC thread has a #8 (0.164″) diameter and 32 threads per inch; it is not an 8/32 (1/4″) diameter fastener with a UNC series thread.

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 fit***, which are indicated by the last number on the thread description.

Fits for inch-based threads are as follows:

- Class 1—Loose fit.
- Class 2—Free fit.
- Class 3—Medium fit.
- Class 4—Close fit.

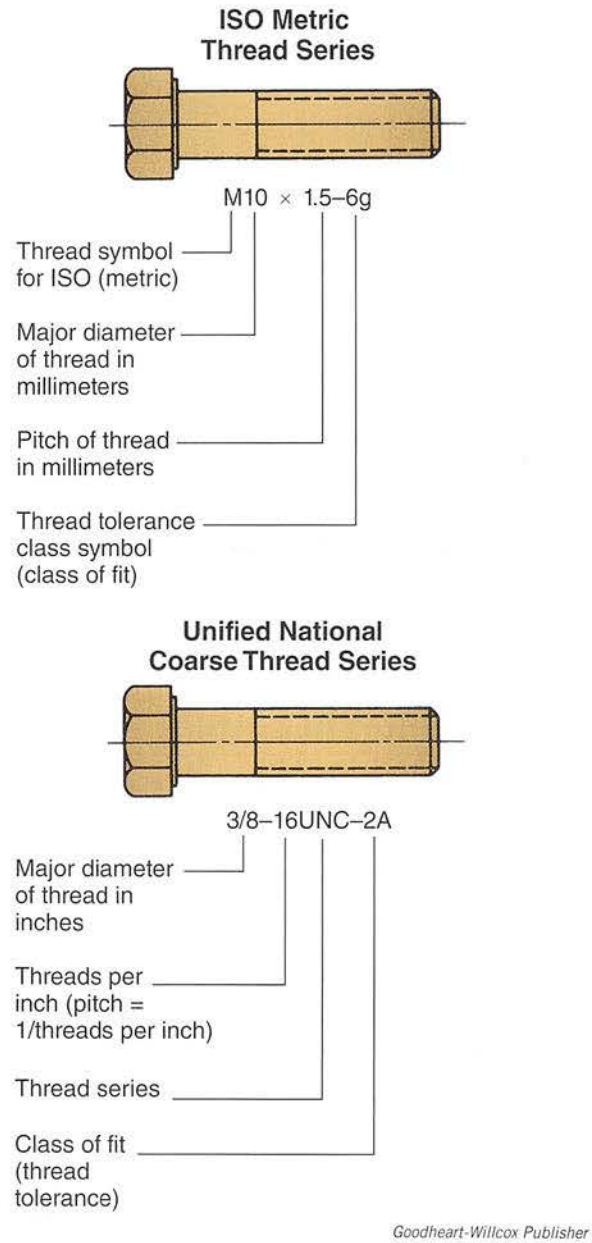
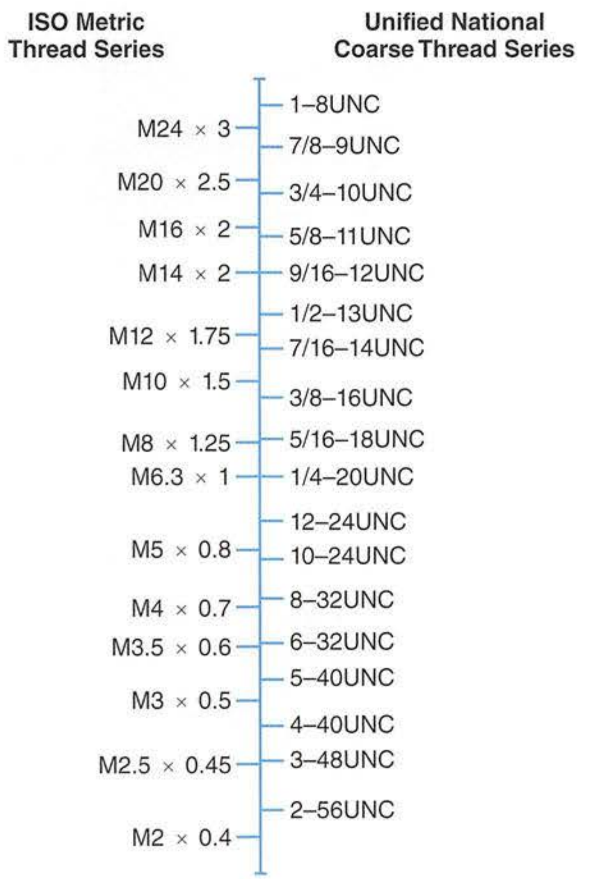


Figure 7-72. The meaning of thread specifications for ISO metric and Unified thread series.

For example, in the thread description 1/2-13 UNC-2, the "2" indicates a class 2 or free fit.

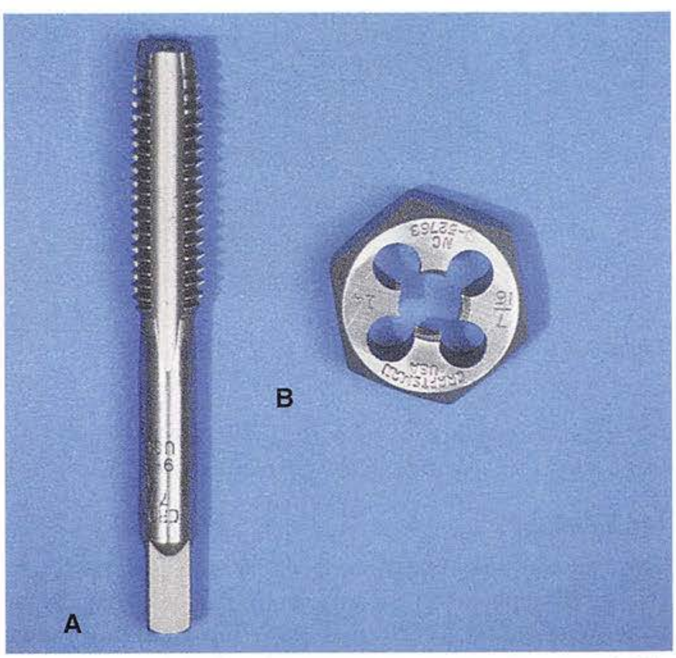
Under revised ISO standards, there are two classes of thread tolerances for external threads: 6g for general-purpose threads and 5g6g for close tolerance threads. There is only one tolerance class for internal threads: 6H. A lowercase letter indicates the tolerance on a bolt and a capital letter is used for the nut.

Because thread dimensions have been standardized, taps and dies are universally used to cut internal and external threads whenever threads are to be cut by hand. Taps are used to cut internal threads, and dies are used to cut external threads. See **Figure 7-74**.



Goodheart-Willcox Publisher

Figure 7-73. While ISO metric threads may appear to be similar in diameter to the Unified thread series, the two are not interchangeable.



Goodheart-Willcox Publisher

Figure 7-74. Thread cutting. A—Internal threads are cut with a tap. B—External threads are cut with a die.

7.10.2 Internal Threads

Internal threads are made with a tap, **Figure 7-75**. 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.

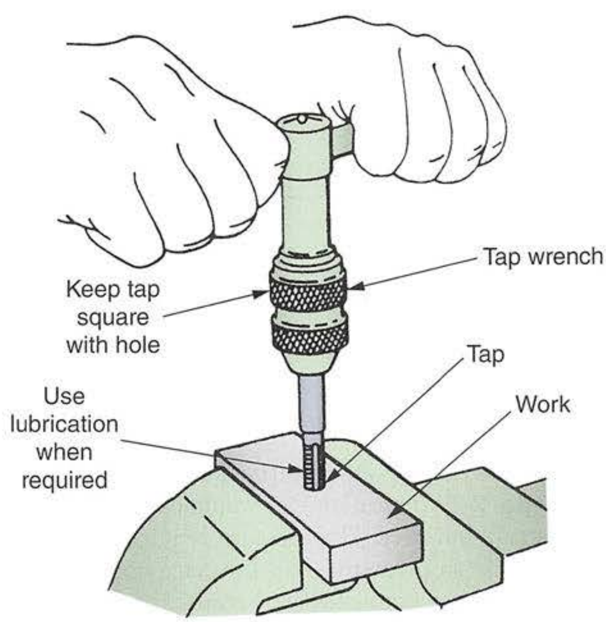
Standard hand taps are made in sets of three. They are known as *taper, plug,* and *bottoming* taps, **Figure 7-76**:

- 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.
- 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 2 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 7-77**.

Another tap used in the shop is a pipe tap. A pipe tap cuts a tapered thread, which sets up a "wedging" action to make a leak-tight joint. The fraction that indicates pipe tap size may be confusing at first because it indicates pipe size, rather than the thread diameter. See **Figure 7-78**. 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.

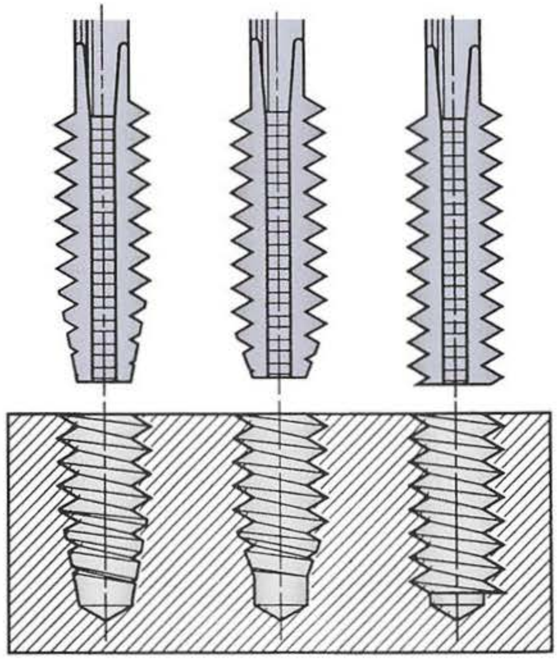
Tap Drill

The drill used to make the hole prior to tapping is called a *tap drill*. Theoretically, it should be equal in diameter to the



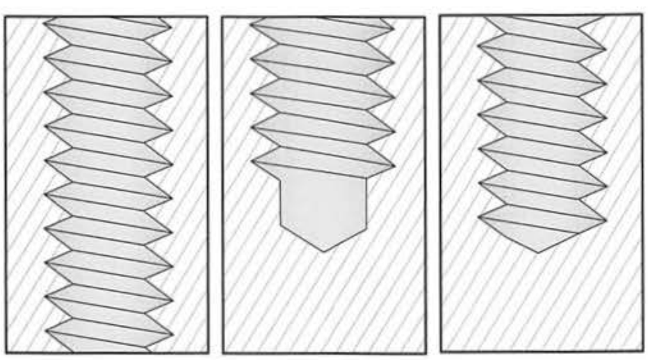
Goodheart-Willcox Publisher

Figure 7-75. Cutting internal threads with a tap.



Goodheart-Willcox Publisher

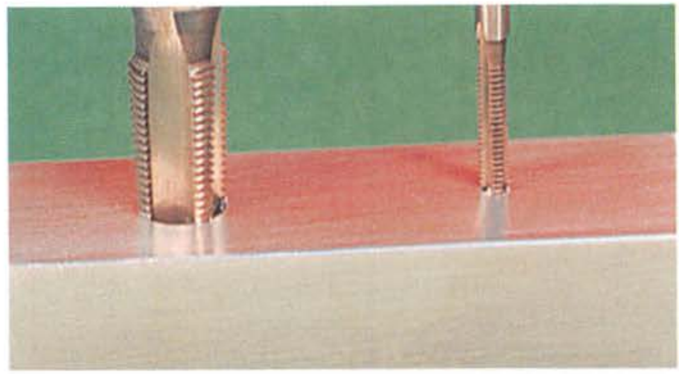
Figure 7-76. Standard hand taps. A—Taper tap for starting the thread. B—Plug tap for continuing the thread after the taper tap has cut as far into hole as it can. C—Bottoming tap for continuing the threads to the bottom of a blind hole.



Goodheart-Willcox Publisher

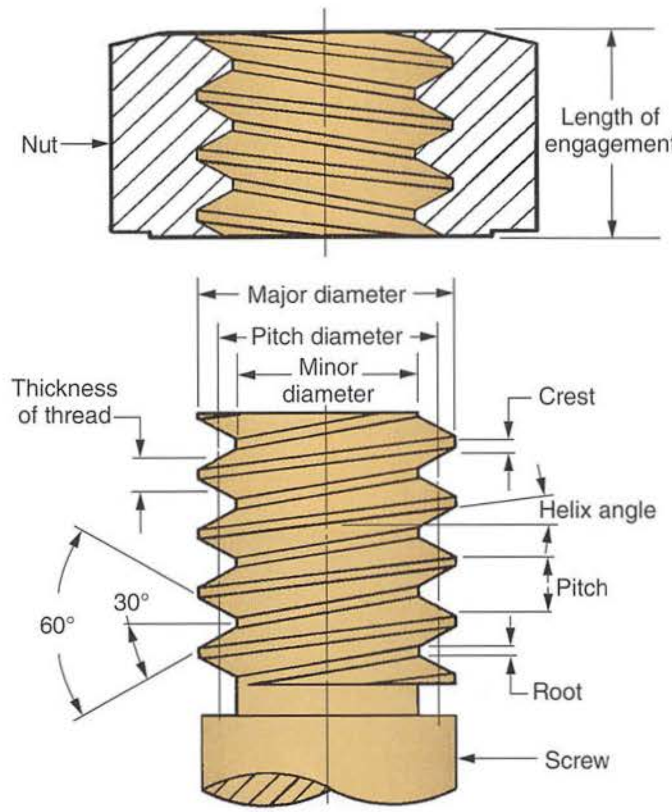
Figure 7-77. Cutaway of a metal block illustrates three types of threaded holes. A—Open or through hole. B—Blind hole that has been drilled deeper than desired threads. C—Blind hole with threads tapped to the bottom of the hole.

minor diameter of the screw that will be fitted into the tapped hole. See **Figure 7-79**. This diameter 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



Goodheart-Willcox Publisher

Figure 7-78. There is considerable difference between a 1/8" standard thread tap and a 1/8" pipe thread tap.



Goodheart-Willcox Publisher

Figure 7-79. Nomenclature of a fastener thread.

breaks before the threads strip. Drill sizes can be determined using a tap drill chart, **Figure 7-80** and **Figure 7-81**.

Tap Wrenches

Two types of tap wrenches are available, **Figure 7-82**. The type used depends on the size of the tap. 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.

National Coarse and National Fine Threads and Tap Drills								
Size	Threads per Inch	Major Dia.	Minor Dia.	Pitch Dia.	Tap Drill 75% Thread	Decimal Equivalent	Clearance Drill	Decimal Equivalent
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

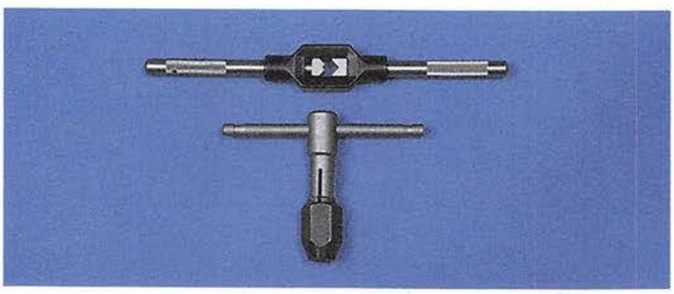
Goodheart-Willcox Publisher

Figure 7-80. Thread and tap drill chart for Unified 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

Goodheart-Willcox Publisher

Figure 7-81. Thread and tap drill chart for metric threads.

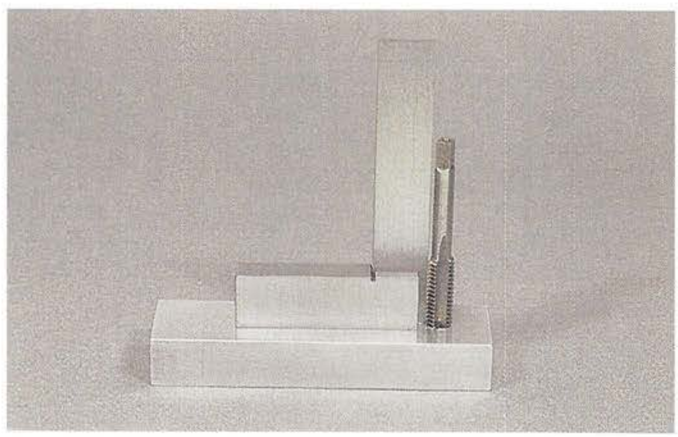


Goodheart-Willcox Publisher

Figure 7-82. Tap wrenches. Top—Hand tap wrench. Bottom—T-handle tap wrench.

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 7-83.** The tap must be backed off (reversed in rotation) one-half turn for every one or two cutting turns. This breaks the chips free and allows them to drop through the tap flutes. Backing off prevents chips from jamming the tap and damaging the threads. Use a cutting fluid designed for the particular metal you are tapping.

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.



Goodheart-Willcox Publisher

Figure 7-83. A tap must be started squarely with the hole. A quick way to check this is to use a machinist's square.

Power Tapper

Tapping can also be done using an articulating arm power tapper, which can be fitted with an auto reverse (the tap is run out of the hole automatically when the threads are at the proper depth), automatic tap lubrication, and digital depth readout. Holes can be tapped quickly with little chance of tap breakage.

Care in Tapping

Considerable care must be exercised when tapping. Follow these guidelines:

- Use the correct size tap drill. You can find this information in 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.

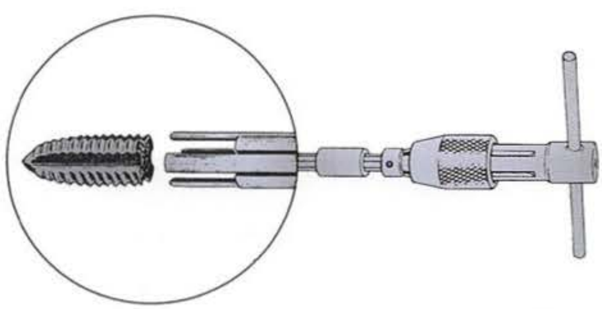
Dealing with Broken Taps

Taps sometimes break off in a hole. Several tools and techniques have been developed to remove them without damaging the threads already cut. However, 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 7-84**. 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.



Goodheart-Willcox Publisher

Figure 7-84. A tap extractor may help remove a broken tap. The close-up shows the fingers of the extractor and how they fit into the flutes of the broken tap. This method does not always work.

Slip the collar on the extractor down flush with the work surface and fit a tap wrench on the extractor. Then carefully twist the tap extractor back and forth to loosen the tap segments. After the broken parts have been loosened, it is a simple matter to remove them.

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.

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 7-85**. 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.

7.10.3 External Threads

External threads are cut with a die, **Figure 7-86**. Solid dies are not adjustable and for that reason are 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.

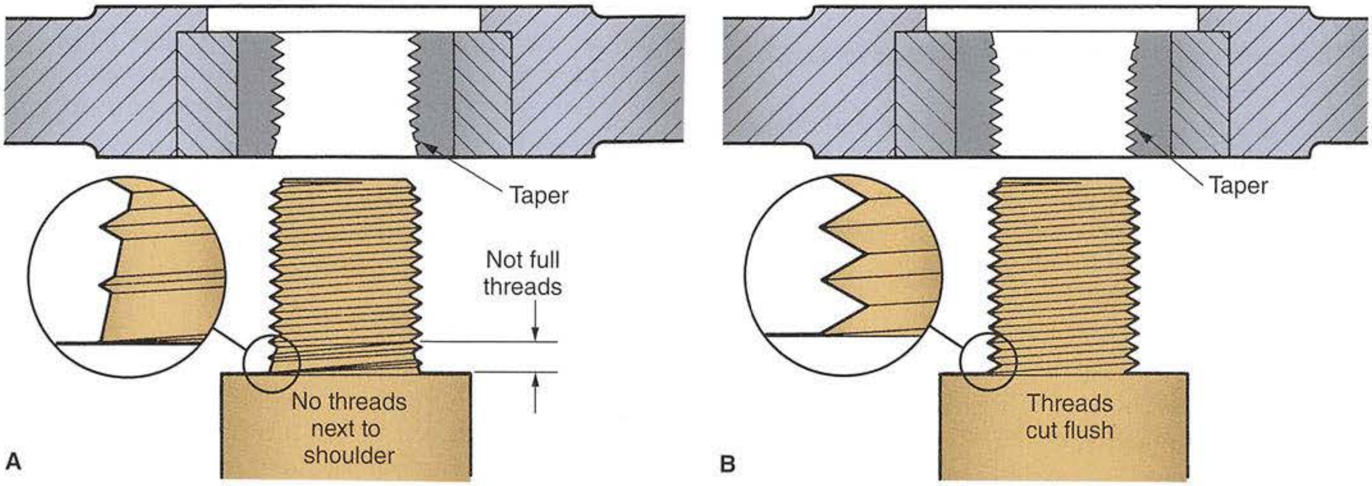
Die Stocks

A die stock holds the die and provides leverage for turning the die on the work. See **Figure 7-87**. When cutting external threads, it is necessary to remember the following:

- 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.
- Mount the work solidly in a vise.
- Set the die to the proper size. Make trial cuts on a piece of scrap until you find the proper adjustment.
- Grind a small chamfer on the shaft end, as shown in **Figure 7-88**. This permits the die to start easily.
- Start the cut with the tapered end of the die.
- Back off the die after every one or two turns to break the chips.
- Use cutting oil. Place a paper towel 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.

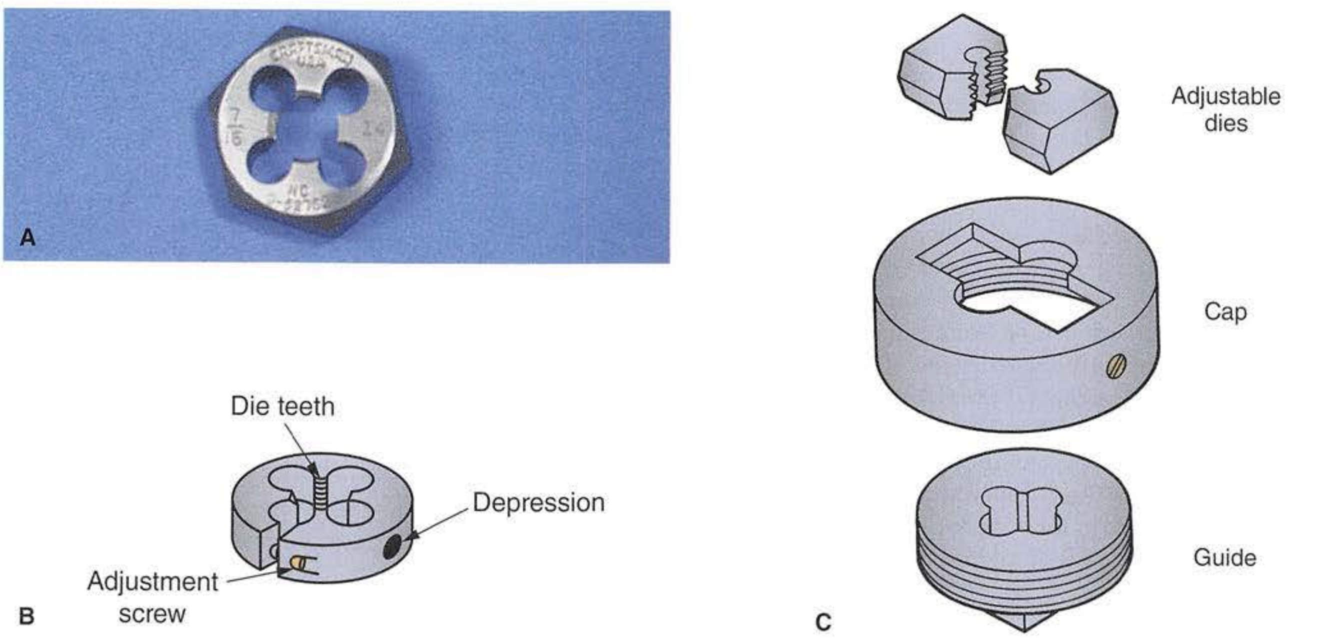
Problems in Cutting External Threads

The most common problem encountered when cutting external threads with a die is ragged threads. Ragged threads can be caused by any of the following:



Goodheart-Willcox Publisher

Figure 7-85. How to cut threads to a shoulder. After the 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 the die down normally. B—Reversing the die to cut flush.



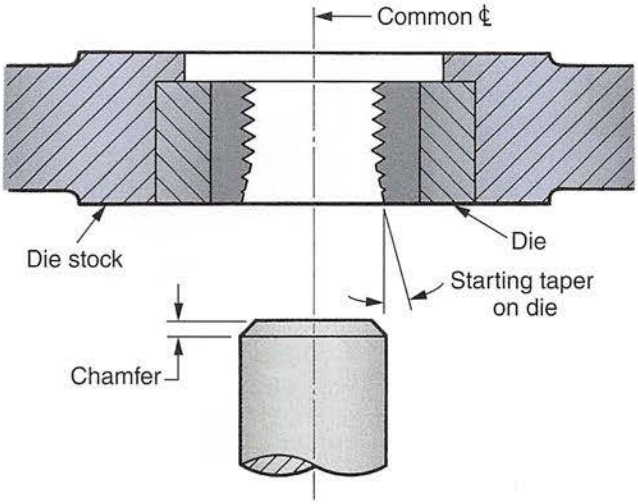
Goodheart-Willcox Publisher

Figure 7-86. 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.

- 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.

Goodheart-Willcox Publisher

Figure 7-87. The die is held in a die stock.



Goodheart-Willcox Publisher

Figure 7-88. 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.

7.10.4 Hand Threading Safety

When cutting threads, keep the following safety precautions in mind:

- 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 others 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.

7.11 Hand Polishing

An ***abrasive*** is any hard substance that will wear away another material. The substance, grain size, backing material, and manner in which the substance is bonded to the backing material determine the performance and efficiency of an abrasive. The table in **Figure 7-89** 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	
80	0	Coarse	
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		

Coated Abrasive Manufacturers Institute

Figure 7-89. A comparative grading chart for abrasives.

7.11.1 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 one of 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 such as 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.

7.11.2 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*.

7.11.3 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 until it is completely used up. Used cloth is excellent for polishing.

Avoid using abrasives on machined surfaces. 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 obtain the proper support, wrap the cloth around a block of wood or file, **Figure 7-90**. Apply pressure and rub the abrasive back and forth in a straight line, parallel to the long side of the work.

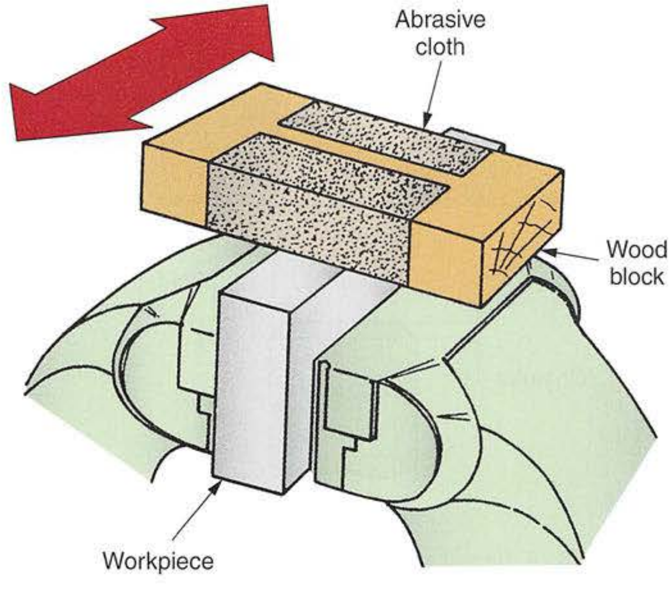


Figure 7-90. Abrasive cloth should be supported with a block of wood or a file. Do not support it with your fingers alone.

7.11.4 Abrasives Safety

When using abrasives, keep the following safety precautions in mind:

- 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 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.

7.12 Broaching Operations

Broaching is a manufacturing process for machining flat, round, and contoured surfaces. Both internal and external surfaces can be shaped by this process. Broaching is ideal for producing keyways, splines, and irregularly shaped openings.

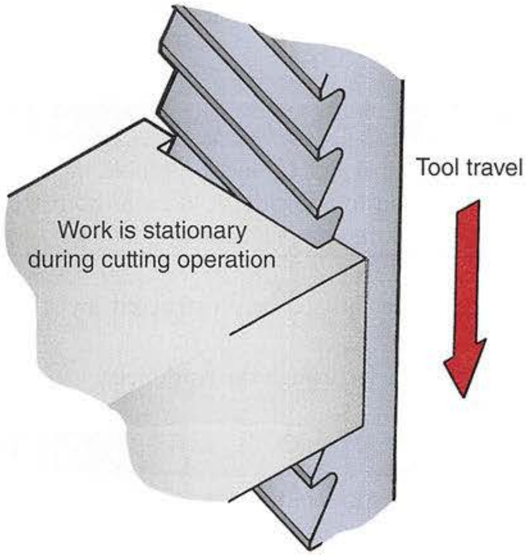
7.12.1 Broaches and Broaching Machines

In broaching, a multitoothed cutting tool is pushed or pulled across the work, **Figure 7-91**. Each tooth on the ***broach*** (the cutting tool) removes only a small portion of the material being machined, **Figure 7-92**. Broaching is usually performed with a cutting fluid.

A broach has three kinds of teeth: roughing teeth, semifinishing teeth, and finishing teeth. The roughing teeth remove the most material, and the semifinishing and finishing teeth produce a finer finish. See **Figure 7-93**.

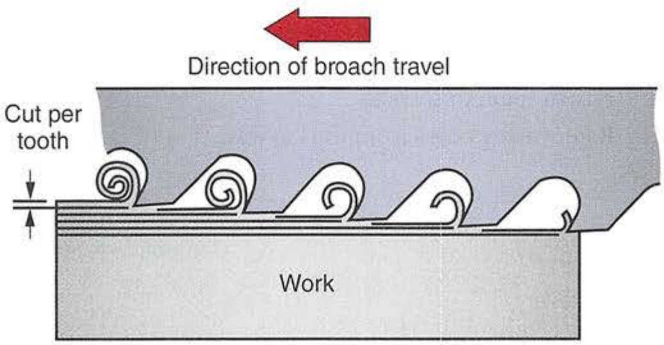
7.12.2 Keyway Broaching

Cutting a ***keyway*** (the slot in a shaft that holds a key) in a gear, pulley, or similar component is a basic broaching operation that can be done in the average machine shop. See **Figure 7-94**. A typical keyway broach set, **Figure 7-95**,



Goodheart-Willcox Publisher

Figure 7-91. Broaching involves the use of a multitooth cutting tool (the broach) that moves against the stationary work. The operation may be on a vertical or horizontal plane, and may involve making internal or external cuts.



Goodheart-Willcox Publisher

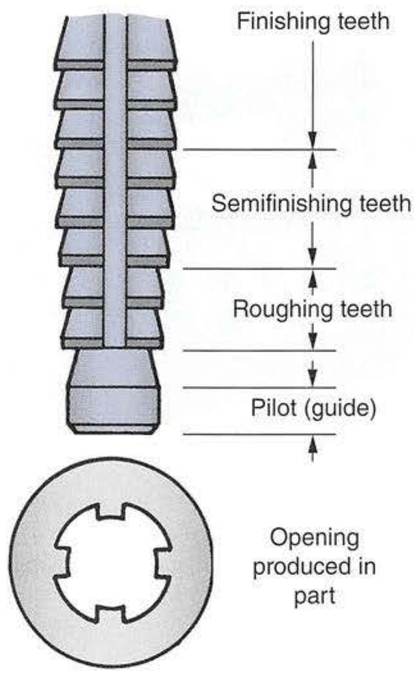
Figure 7-92. Each tooth on a broach removes only a small portion of the material being machined.

contains an assortment of precision broaches, slotted bushings, the necessary shims, instructions, and a lubrication guide.

Small arbor presses, like those shown in **Figure 7-96**, can also be used to progressively remove material from inside diameters to create keyways using the broaching process. An ***arbor press*** is a manually operated press that can have a leverage of several tons. It is used on smaller work to cut keyways in gears, pulleys, and similar components.

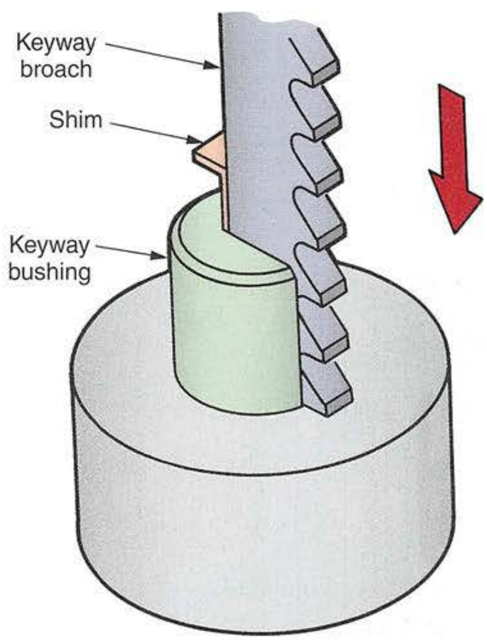
SAFETY NOTE

Handle the broach with care. Its sharp teeth can cause a serious injury to the hand.



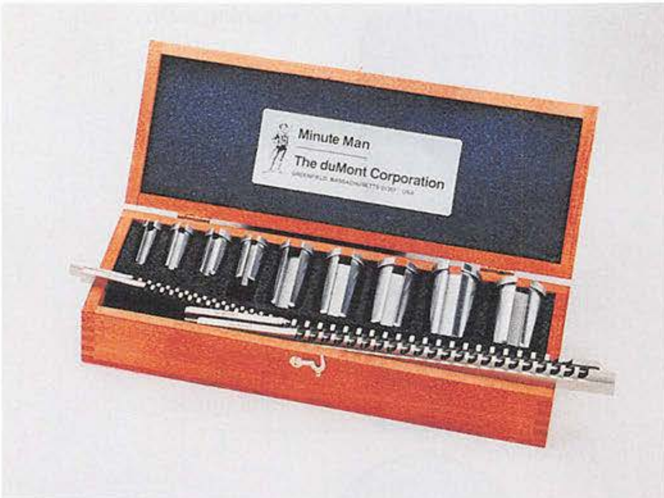
Goodheart-Willcox Publisher

Figure 7-93. This drawing shows a greatly shortened section of an internal broaching tool and the splines it cuts into a part. The pilot guides the cutter into a cut or hole previously made in the work. Each tooth of the broach increases slightly in size until the specified size is attained.



Goodheart-Willcox Publisher

Figure 7-94. Cutting a keyway using an arbor press to push a broach through the work. Several passes may be required. Shims of different sizes move the cutting tool into the work.



duMont Corp.

Figure 7-95. A typical keyway broach set. It contains precision broaches, slotted bushings, and necessary shims. Ample lubrication is necessary.

First, measure the bore into which the keyway is to be cut. Then complete the following steps:

1. Select the bushing that fits the hole and the required broach.
2. Place the bushing into the hole and insert the broach.
3. Set the assembly into position on the arbor press, making sure there is ample clearance for the broach to pass through the work. Be sure that the broach is centered on the arbor press ram.

SAFETY NOTE

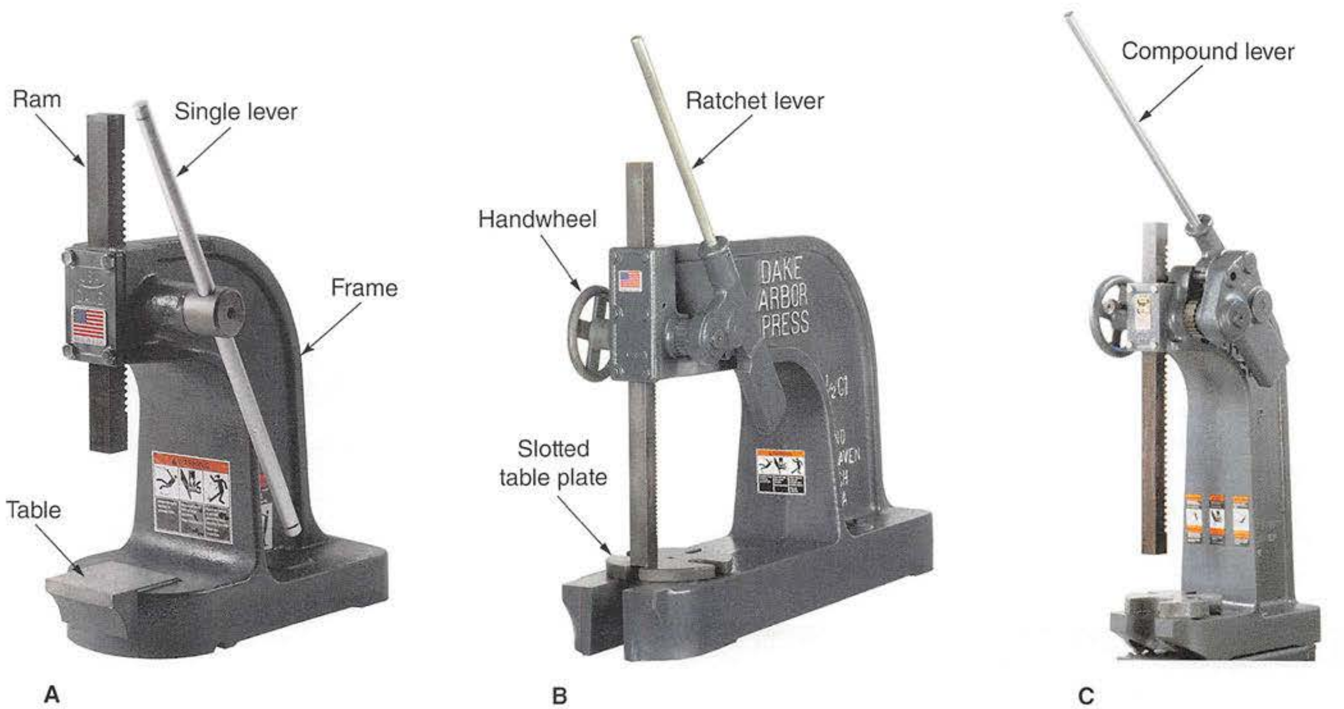
If the broach is not centered, it could be damaged by being pushed to one side. A loose or worn arbor press ram can also damage a broach by pushing it to one side.

4. Lubricate the broach as instructed by the broach manufacturer.
5. Push the broach through the workpiece.

SAFETY NOTE

Do not allow the tool to fall to the floor as it is pushed through the workpiece. Dropping the broach could damage the tool.

6. Clean the broach and insert the second pass shim.
7. Lubricate the broach again, and push the broach through.
8. Repeat the sequence until the keyway is the correct depth. Take care to properly align the broach with each previous path.
9. Use a clean cloth to wipe the broach, bushing, and shims clean. Apply a thin coating of oil to prevent rusting and return them to storage.
10. Remove any burrs from the keyway.



Dake Corporation

Figure 7-96. Examples of three different types of arbor presses. A—A single lever arbor press. B—A ratchet lever arbor press with a handwheel, slotted table plate, and counter-weighted lever. C—A compound leverage arbor press can be used for heavier press work that requires greater pressure.

Chapter Review

Summary

- Several different vises and clamps can be used to mount a piece to be worked on.
- Wrenches and screwdrivers come in many different shapes and sizes and serve many different purposes.
- Great care must be taken when working with chisels and hacksaws to avoid injury.
- It is important to know how to select the right tool for the project.
- Threading is a precise process that has different classes of fits.
- Internal threads are made using taps, and external threads are made using dies.
- Various materials can be used as abrasives for polishing and finishing a surface.
- A keyway can be cut in a single pass with a keyway broach set on an arbor press.

Review Questions

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

1. List two variations of the bench vise.
2. How is vise size determined?
3. How can the work held in a vise be protected from damage by the jaw serrations?
4. To prevent injuries, what should be avoided when mounting work in a vise?
5. A(n) _____ clamp is ideal for holding small work.
6. What advantage do slip-joint pliers have over many other types of pliers?
7. Why are the cutting edges on diagonal pliers set at an angle?
8. What are adjustable clamping pliers?
9. List three ways of extending the working life of pliers.
10. For what purpose are torque wrenches used?
11. Do torque wrenches give a more accurate reading when they are pushed or when they are pulled? Explain.
12. Several different wrenches can be classified as adjustable wrenches. Name two.
13. List three guidelines that should be observed when using an adjustable wrench.
14. Round work can be gripped with a(n) _____ wrench.
15. Describe socket wrenches.
16. The wrench that is used with socket-head fasteners is commonly known as a(n) _____.
17. When using a wrench, is it better to push or pull on the handle?
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?
20. List three safety precautions that should be observed when using a screwdriver.
21. How is the size of a ball-peen hammer determined?
22. When are soft-face hammers and mallets used in place of a ball-peen hammer?
23. List three safety precautions that should be observed when using striking tools.
24. List the four general types of cold chisels.
25. The chisel is an ideal tool for removing _____ by shearing the heads off.
26. A dangerous condition that can cause injury if not fixed is a(n) _____ chisel head.
27. Why are almost all hacksaws adjustable?
28. Why should the work be mounted solidly and close to the vise before cutting with a hacksaw?
29. A hacksaw cuts best at about _____ strokes per minute.
30. If a blade breaks or dulls before completing a cut, why should you not continue in the same cut with a new blade?
31. It is important to choose a hacksaw blade that will have at least _____ teeth cutting the workpiece at all times.
32. What is the best way to hold soft metal tubing for hacksawing?
33. What is the best way to hold thin metal for hacksawing?
34. What are the most commonly used file shapes?
35. List the four types of file cuts.
36. Files are cleaned with a(n) _____, never with your hand.
37. List three safety precautions that should be observed when files are used.
38. What is hand reaming used for?
39. How much stock should be left in a hole for hand reaming?
40. How does the UNC thread series differ from the UNF thread series?
41. A(n) _____ is used to cut internal threads.
42. A hole that does not go entirely through the part is called a(n) _____ hole.
43. List the correct sequence of taps used to form threads to the full depth of a blind hole.
44. The drill used to make the hole prior to tapping is called a(n) _____.

45. Which type of tap wrench is used with small taps?
46. External threads are cut with a(n) ______.
47. What is an abrasive?
48. Broaching is a manufacturing process for machining ______ surfaces.
    A. flat
    B. round
    C. contoured
    D. All of the above.
    E. None of the above.
49. What are the three types of teeth on the cutting tool of a broaching machine?

Match each phrase with the correct screwdriver name listed below.

50. Has a flattened wedge-shaped tip.
51. Moves the fastener on the power stroke, but not on the return stroke.
52. Has a square shank to permit additional force to be applied with a wrench.
53. Tip is similar to that of a Phillips head screwdriver.
54. Has an insulated handle.
55. Is short and is used when space is limited.

A. Pozidriv®
B. Standard
C. Electrician
D. Heavy-duty
E. Stubby
F. Ratchet

CHAPTER 8
Fasteners

Chapter Outline

8.1 Threaded Fasteners
- **8.1.1** Machine Screws
- **8.1.2** Machine Bolts
- **8.1.3** Cap Screws
- **8.1.4** Setscrews
- **8.1.5** Stud Bolts
- **8.1.6** Eye Bolts
- **8.1.7** Removing Broken or Sheared Bolts
- **8.1.8** Nuts
- **8.1.9** Inserts
- **8.1.10** Washers
- **8.1.11** Lock Washers
- **8.1.12** Liquid Thread Lock
- **8.1.13** Thread-Forming and Thread-Cutting Screws
- **8.1.14** Drive Screws

8.2 Nonthreaded Fastening Devices
- **8.2.1** Dowel Pins
- **8.2.2** Cotter Pins
- **8.2.3** Retaining Rings
- **8.2.4** Rivets
- **8.2.5** Keys

8.3 Adhesives
- **8.3.1** Types of Adhesives
- **8.3.2** Using Adhesives

Learning Objectives

After studying this chapter, you will be able to:
- Identify several types of fasteners.
- Describe how specific fasteners are used.
- Select the proper fastening technique for a specific job.
- Describe adhesive fastening techniques.

Technical Terms

adhesive
cyanoacrylate quick-setting adhesive
fastener
key
threaded fastener

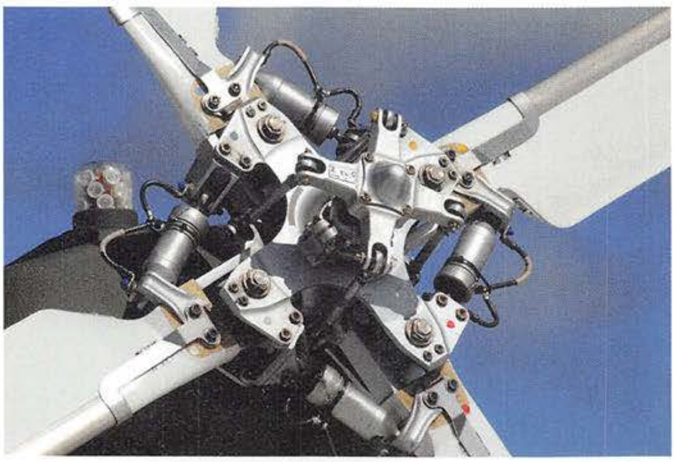
A *fastener* is any device used to hold two objects or parts together. Examples of fasteners include bolts, nuts, screws, pins, keys, rivets, and even chemical bonding agents or adhesives. The most common types of fasteners are explained and illustrated in this chapter.

It is critical to choose the proper fasteners for each job, **Figure 8-1**. A poorly selected fastener can greatly reduce the safety and dependability originally designed into a product. Choosing improper fasteners could increase assembly costs or result in an inferior or faulty product. To improve quality, several different fastening techniques are often used in the same or related assemblies. For example, one auto manufacturer uses more than 11,000 kinds and sizes of fasteners.

8.1 Threaded Fasteners

Threaded fasteners make use of the wedging action of a screw thread to clamp parts together. 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 8-2**. 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 8-3**.

Most threaded fasteners are available in both US Conventional and metric sizes. Many American manufacturers now use metric fasteners in their products, which has led to some problems. Metric threads and the common Unified (inch-based) threads have the same basic profile (shape), but they are *not* interchangeable. See **Figure 8-4**.

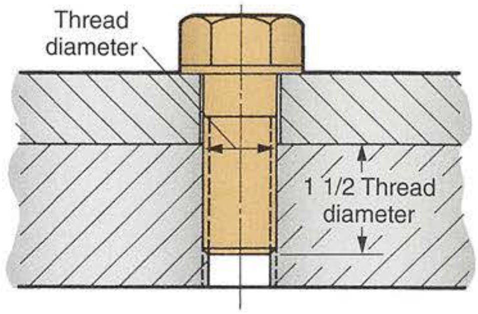


Steve Mann/Shutterstock.com

Figure 8-1. Complex assemblies, such as this helicopter tail rotor, require the use of many types and sizes of fasteners. Reliability of the product, and the safety of people using it, can be greatly reduced if improper fasteners are selected in the design or assembly phases.

8.1.1 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 8-5**. Machine screws are available in body diameters ranging from #0000 (0.021″) to 3/4″ and

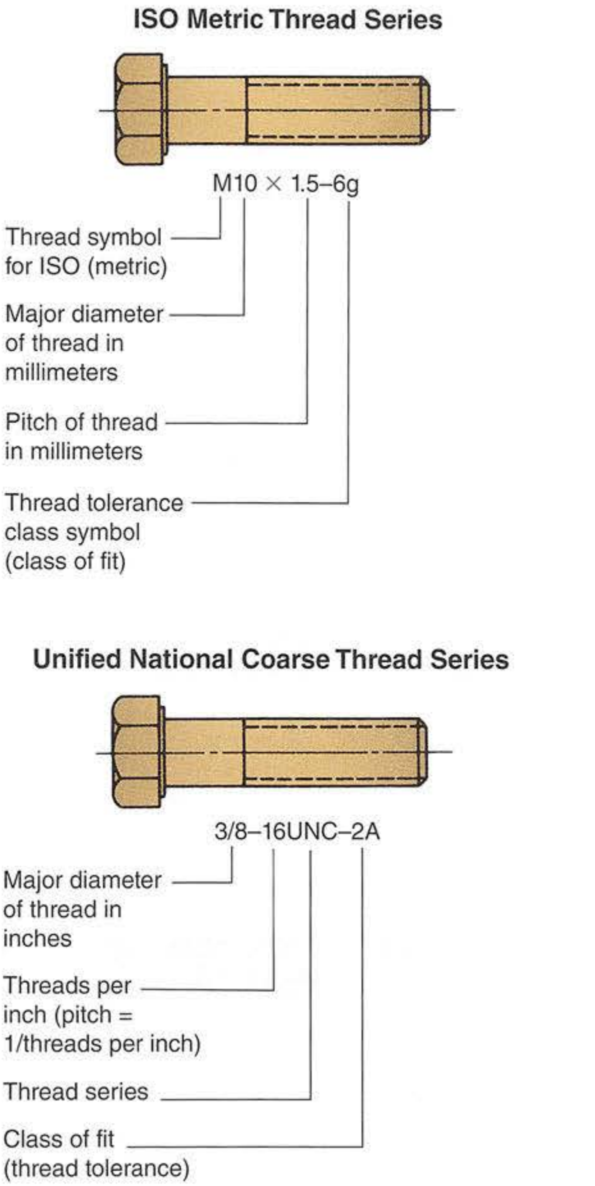


Goodheart-Willcox Publisher

Figure 8-2. 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.

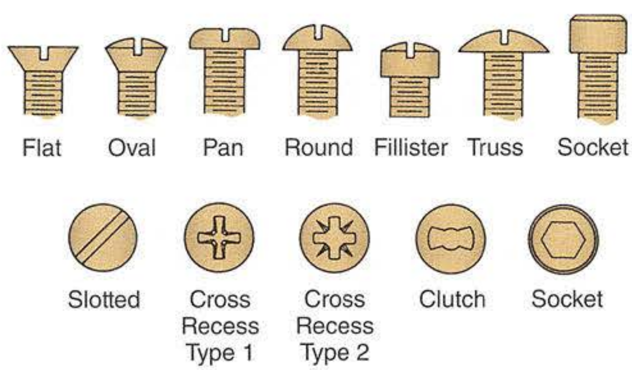
1125089601/Shutterstock.com

Figure 8-3. A wide range of threaded fasteners is available, from the 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.



Goodheart-Willcox Publisher

Figure 8-4. Metric threads have the same basic profile (shape) as the Unified thread series. However, Unified and metric threads are not interchangeable.



Goodheart-Willcox Publisher

Figure 8-5. A sampling of the many types of machine screws available.

in lengths from 1/8″ to 3″. Metric sizes are manufactured in body diameters from 1.6 mm to 12 mm and in lengths from 3 mm to 100 mm. Nuts, in either square or hexagonal shapes, are purchased separately.

8.1.2 Machine Bolts

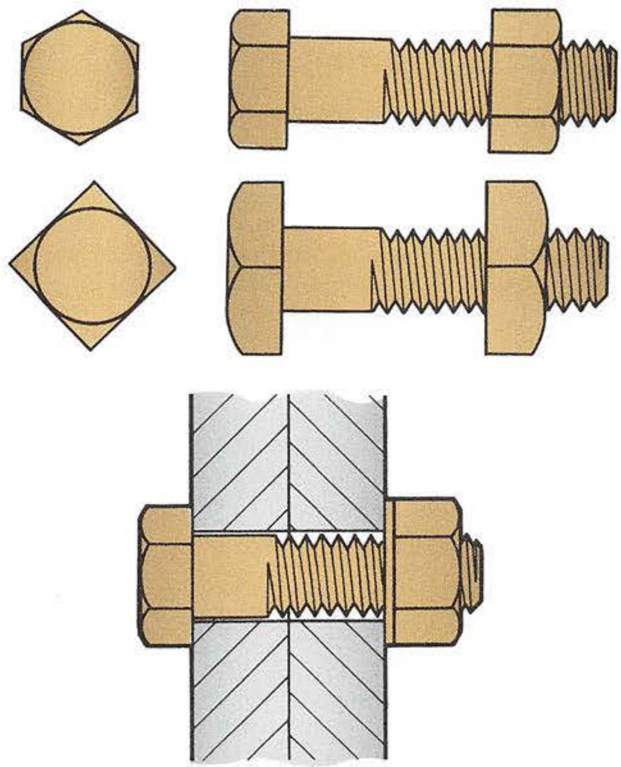
Machine bolts are used to assemble parts that do not require close tolerances. They are manufactured with square and hexagonal heads, in diameters ranging from 1/2″ to 3″. The nuts are similar in shape to the bolt head. They are usually furnished with the machine bolts. Tightening the nut produces a clamping action to hold parts together, **Figure 8-6**.

8.1.3 Cap Screws

Cap screws are used in assemblies requiring a higher quality and a more finished appearance, **Figure 8-7**. 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 have a machined or semifinished bearing surface under the head. Some types of cap screws are heat-treated.

The strength of the cap screw to be used depends on the application. Required strength is indicated on the print.



Goodheart-Willcox Publisher

Figure 8-6. Machine bolts use a nut to produce clamping force.

Since all steel hex-head cap screws are similar in appearance, a series of markings on the head indicate their strength. The machinist can look up these markings in tables and charts to determine the strength of a fastener and the material from which it was made. See **Figure 8-8**. The stronger the cap screw, the more expensive it is.

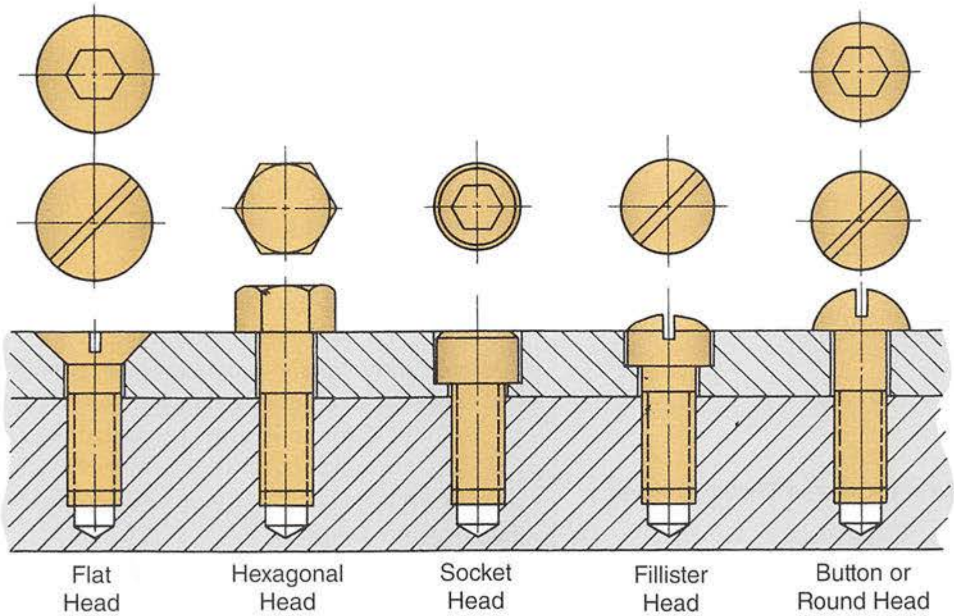


Figure 8-7. Cap screws are tightened into the mating part and do not require nuts to achieve a clamping action.

CAREER CONNECTION

Ironworker

What does an ironworker do?

Ironworkers assemble the metal frames that provide structure and support for buildings, bridges, roadways, and other structures. While called 'ironworkers,' these professionals primarily work with structural steel. While they may work at great heights while constructing high-rises or bridges, they also lay the reinforcing iron that strengthens concrete used in highways and roads. Ironworkers in assembly/fabrication may preassemble metal structures off-site.

What education and skills are needed to be an ironworker?

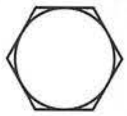
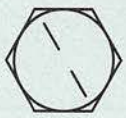
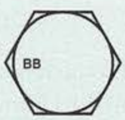
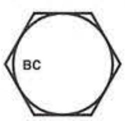
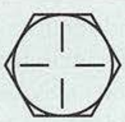
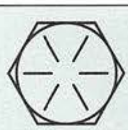
Ironworkers learn their trade on the job through 3- to 4-year apprenticeships. A high school diploma is required for entry into most apprenticeship programs. Interested students should study math and print reading, welding, and other technical subjects.

The physical and mental demands of this career must also be considered carefully. Ironworkers must be mentally and physically able to work comfortably at great heights without anxiety or accidents due to poor balance or stamina.

What is it like to be an ironworker?

Ironworkers must adapt to the jobsite, whether they are working on the ground or fifty stories up on an unfinished skyscraper. Because they often work at great heights, ironworking is weather-dependent. Working in inclement conditions is not worth the risk of injury or death.

The median annual wage for ironworkers, as reported by the *Occupational Outlook Handbook*, is $48,000. The highest wage is more than $91,200.

Grade Marking for Bolts				
Bolt Head Marking		**SAE = Society of Automotive Engineers** **ASTM = American Society for Testing and Materials**	**Bolt Material**	**Minimum Tensile Strength in Pounds per Square Inch (psi)**
No Marks		SAE Grade 1 SAE Grade 2 Indeterminate quality	Low-carbon steel Low-carbon steel	65,000 psi
2 Marks		SAE Grade 3	Medium-carbon steel, cold worked	110,000 psi
3 Marks		SAE Grade 5 ASTM – A 325 Common commercial quality	Medium-carbon steel, quenched and tempered	120,000 psi
Letters BB		ASTM – A 354	Low-alloy steel or medium-carbon steel, quenched and tempered	105,000 psi
Letters BC		ASTM – A 354	Low-alloy steel or medium-carbon steel, quenched and tempered	125,000 psi
4 Marks		SAE Grade 6 Better commercial quality	Medium-carbon steel, quenched and tempered	140,000 psi
5 Marks		SAE Grade 7	Medium-carbon alloy steel, quenched and tempered, roll-threaded after heat treatment	133,000 psi
6 Marks		SAE Grade 8 ASTM – A 345 Best commercial quality	Medium-carbon alloy steel, quenched and tempered	150,000 psi

Goodheart-Willcox Publisher

Figure 8-8. Markings on the heads of bolts and cap screws identify various grades. Each grade has specific material and strength requirements.

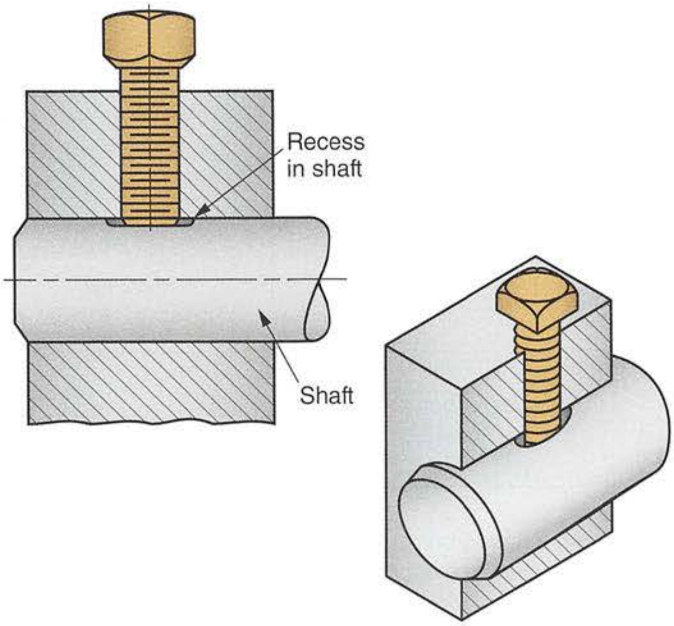
8.1.4 Setscrews

Setscrews are semipermanent fasteners that are used for applications such as preventing pulleys from slipping on shafts, holding collars in place on assemblies, and positioning shafts on assemblies. See **Figure 8-9**. Setscrews are usually made of heat-treated steel. They are classified in two ways: by head style and by point style, **Figure 8-10**.

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 8-11**. Thumbscrews are available with points similar to those on setscrews.

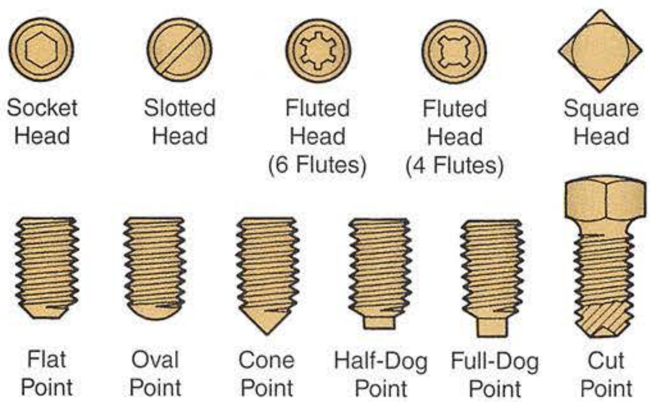
8.1.5 Stud Bolts

Stud bolts are headless bolts that are threaded over their entire length or (more commonly) on both ends, **Figure 8-12**. One end is designed for semipermanent installation in a tapped hole, and the other end is threaded for standard nut assembly to clamp the pieces together.



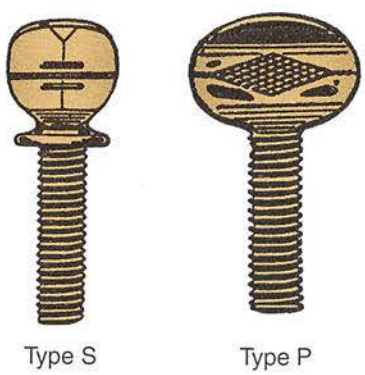
Goodheart-Willcox Publisher

Figure 8-9. Setscrews lock pulleys and gears to shafts to prevent rotation of the shaft in the hole.



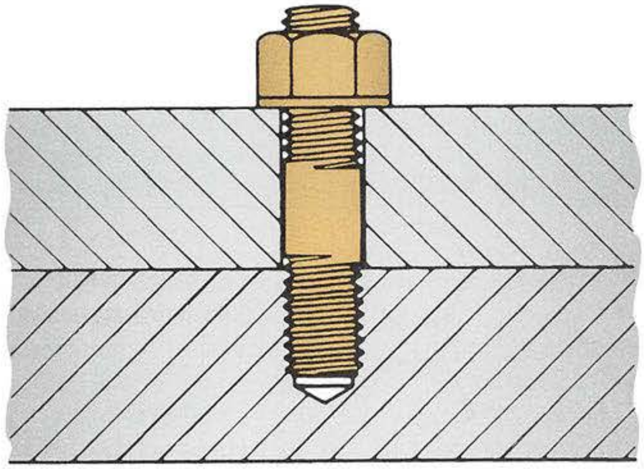
Goodheart-Willcox Publisher

Figure 8-10. Setscrew head and point designs.



Parker-Kalon

Figure 8-11. Thumbscrews can be removed or installed by hand.

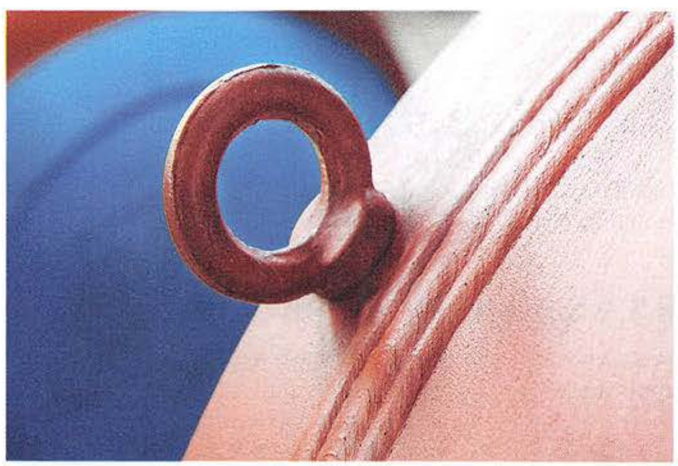


Goodheart-Willcox Publisher

Figure 8-12. One end of a stud bolt usually threads into a part, while the other end accepts a nut.

8.1.6 Eye Bolts

Eye bolts technically are not used to fasten objects together, but it is important for the machinist to know what they are and how they are commonly used in the machine shop. An eye bolt is a threaded bolt with a loop at one end. The purpose of the eye bolt is to create a means to lift and move heavy objects and equipment by providing a loop through which ropes, chains, or cables can be securely fastened. Heavy objects, such as stamping dies and molds, often need to be lifted onto a bench or machine for maintenance or repair. These objects usually have threaded holes specifically for the purpose of threading an eye bolt into the tool so it can be safely lifted and moved, **Figure 8-13**.



Kuraka/Shutterstock.com

Figure 8-13. Threaded holes for eye bolts are often machined into heavy equipment to provide a means of lifting or moving the equipment.

8.1.7 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 8-14**, is available in several sizes. It is used with a 3/8″ 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.

A spiral-fluted bolt extractor, **Figure 8-15**, 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.

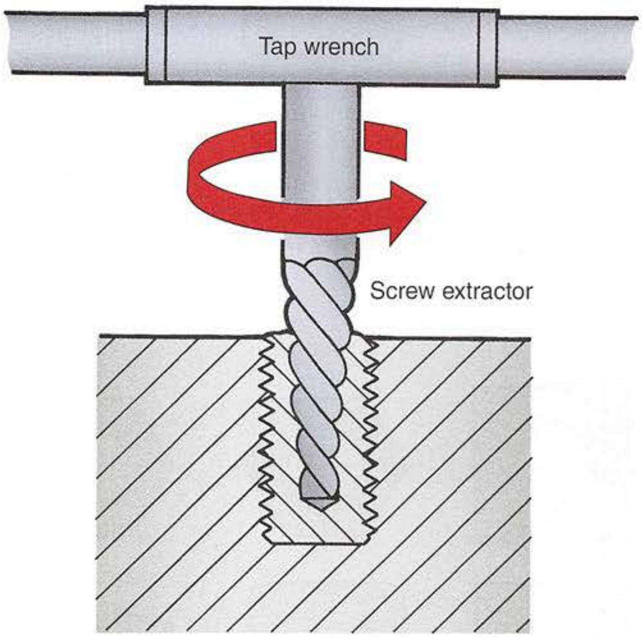
Extractors of the type shown in **Figure 8-16** are designed to remove sheared machine screws. The blade is made of heat-treated tool steel. A hole is drilled in the screw, and then the tapering blade is lightly driven into the hole. The extractor is turned counterclockwise to remove the screw. Treatment with penetrating oil will often make it easier to remove stubborn sheared bolts and machine screws.

8.1.8 Nuts

For most threaded fasteners, nuts with hexagonal or square shapes are used with bolts having a head of the same shape. Nuts are usually manufactured of the same materials as their mating bolts. Nuts are available in various degrees of finish, **Figure 8-17**:

- A regular nut is unfinished (not machined) except on the threads.
- A regular semifinished nut is machined on the bearing face to provide a truer surface for the washer.
- A heavy semifinished nut is identical in finish to the regular semifinished nut. However, the body is thicker for additional strength.

Nuts vary in shape and size depending on their intended function. Plain hexagonal nuts are the most common. Square nuts are no longer common, but can be found in older assemblies. The jam nut is thinner than the standard nut. It is frequently used to lock a full-size nut in place, **Figure 8-18**.

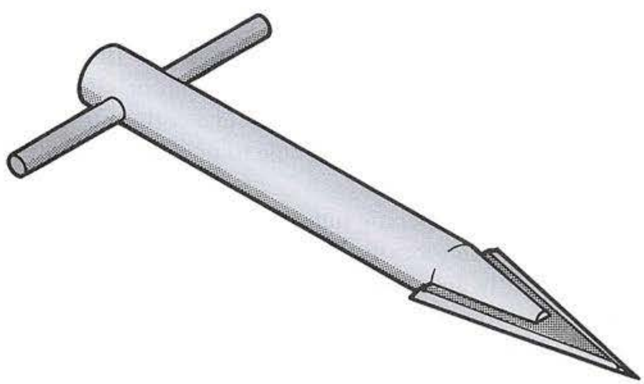


Goodheart-Willcox Publisher

Figure 8-15. Spiral-flute broken bolt extractor. A hole of proper size is drilled, a tap wrench is applied, and the broken bolt is turned out.

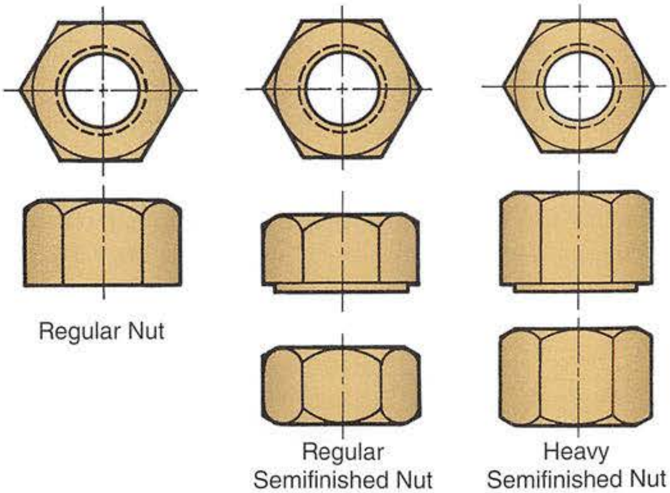
Alden Corp.

Figure 8-14. The Drill-Out Power Extractor™ combines a drill with an adjustable extractor collar. The drill makes the required size hole, and then the power drill is reversed. This causes the extractor to grip the broken bolt and torque it out.



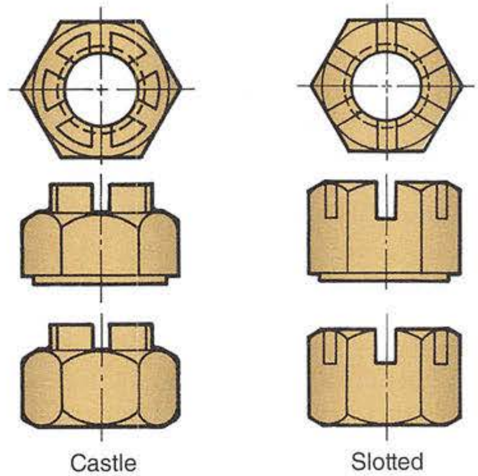
Goodheart-Willcox Publisher

Figure 8-16. 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 the broken screw.



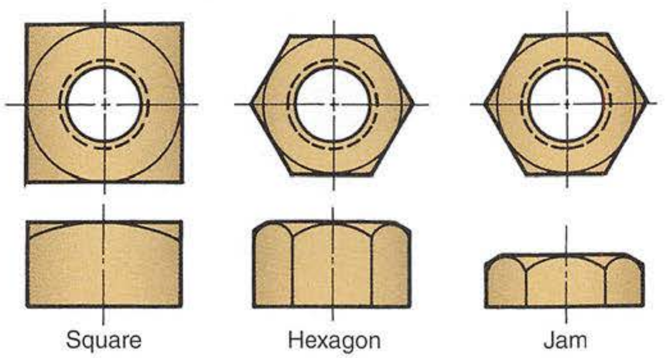
Goodheart-Willcox Publisher

Figure 8-17. Degrees of finished nuts. Only the threads of regular nuts are machined. Regular semifinished nuts have a machined bearing surface. Heavy semifinished nuts are thicker than regular nuts.



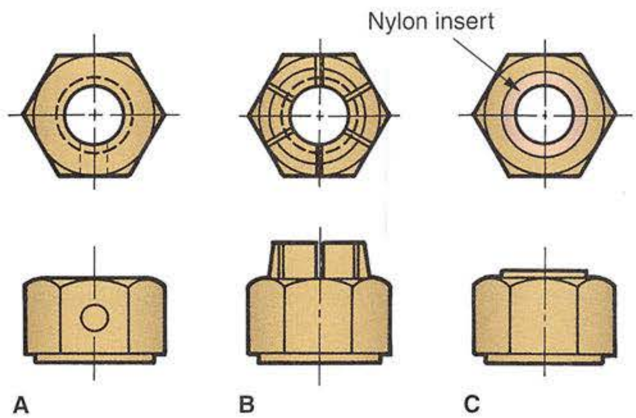
Goodheart-Willcox Publisher

Figure 8-19. Castle and slotted nuts can be locked in place with a cotter pin or wire, but a hole must be drilled through the bolt first.



Goodheart-Willcox Publisher

Figure 8-18. Square nuts are no longer common. A hexagon nut has six vertical sides and can be loosened or tightened with standard sockets and wrenches. A jam nut can lock a full-size nut in place to prevent loosening of the bolt.



Goodheart-Willcox Publisher

Figure 8-20. Self-locking nuts. A—The threads of center-lock nuts are deformed in the middle of the nut. B—Split-beam nuts have slots cut into the top of the nut that are bent slightly inward. C—Lock nut with a nylon insert.

Castle 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, **Figure 8-19**.

These types of locking nuts are being replaced in many applications by self-locking nuts, **Figure 8-20**. Self-locking nuts are slightly deformed to produce a friction fit, or they have a nylon insert so they cannot vibrate loose. No hole through the bolt is required when self-locking nuts are used in an assembly.

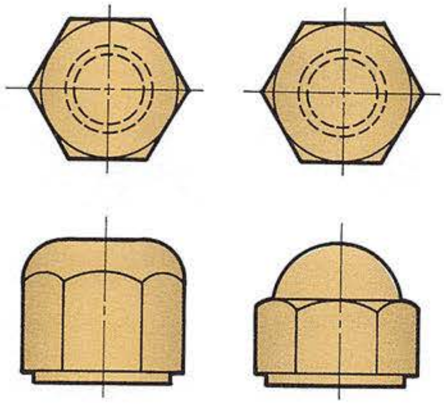
In critical assemblies, always use a new self-locking nut to replace one that has been removed for any reason. The used nut may not have adequate locking action remaining and may loosen in service.

Acorn nuts are used when appearance is of primary importance, or when projecting threads must be protected. They are available in high- or low-crown styles. See **Figure 8-21**.

A wing nut is used when frequent adjustment or removal is necessary. It can be loosened and tightened rapidly without a wrench. Refer to **Figure 8-22**.

8.1.9 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 8-23** 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 of standard size and form. For optimum results, inserts must be installed according to the manufacturer's instructions.



Goodheart-Willcox Publisher

Figure 8-21. Acorn or cap nuts look good and protect threads.

Parker-Kalon

Figure 8-22. A wing nut has "wings" to help the machinist grip and turn the nut. Like the thumbscrew, it is turned by hand.

8.1.10 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 8-24**.

8.1.11 Lock Washers

A lock washer prevents 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 in most applications. See **Figure 8-25**.

Preassembled lock washer/screw nuts and lock washer/nut units have a washer mounted on the nut. They are used to reduce assembly time and waste in the mass-assembly market.

8.1.12 Liquid Thread Lock

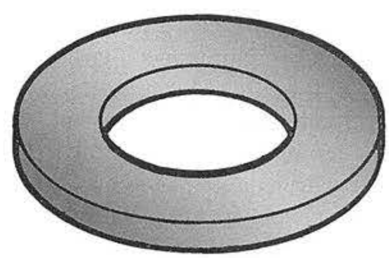
Nuts, bolts, and machine screws can be prevented from loosening due to vibration through use of a liquid thread lock, **Figure 8-26**. Although the thread lock material prevents fasteners from vibrating loose, it allows easy removal of the fastener should disassembly be necessary. When using a liquid thread lock, follow the manufacturer's recommendations for maximum effectiveness.

A

B

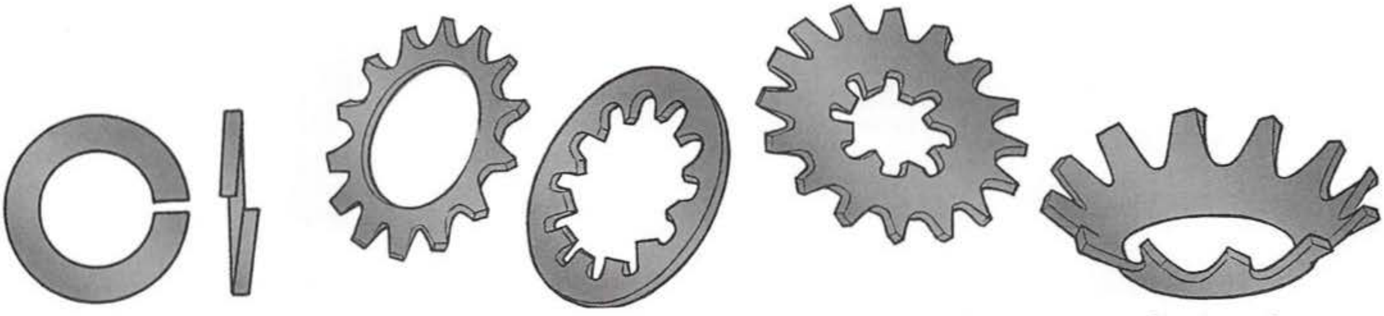
Heli-Coil Corp.; Jergens, Inc.

Figure 8-23. Thread repair inserts. A—An insert is frequently used to replace damaged or stripped threads in a part. B—These key-locking inserts can be easily installed or removed without special tools. They are available in both carbon steel and stainless steel.



Goodheart-Willcox Publisher

Figure 8-24. The standard flat washer provides a bearing surface for a fastener.

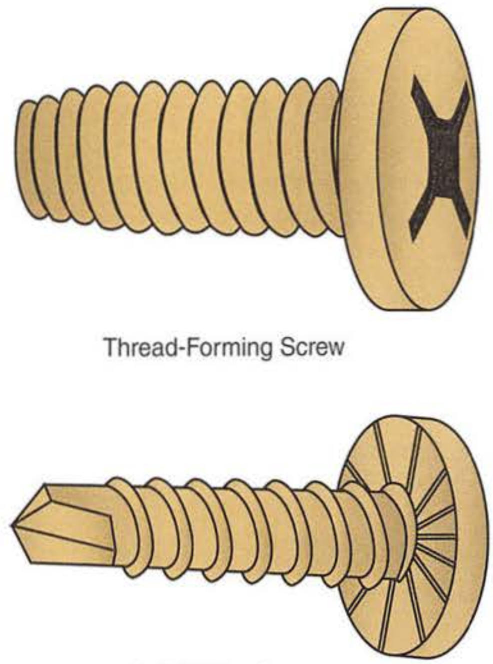


Goodheart-Willcox Publisher

Figure 8-25. Lock washer variations. External lock washers should be used whenever possible, because they provide greatest resistance to turning. The internal lock washer is used with small-head screws and to hide teeth, either for appearance or to prevent snagging. Internal-external lock washers are used when mounting holes are oversize. The countersunk lock washer is used with flat- or oval-head screws.

Goodheart-Willcox Publisher

Figure 8-26. Many brands of liquid thread locks are available. They prevent a bolt, nut, or screw from vibrating loose, but allow easy removal should disassembly be required.



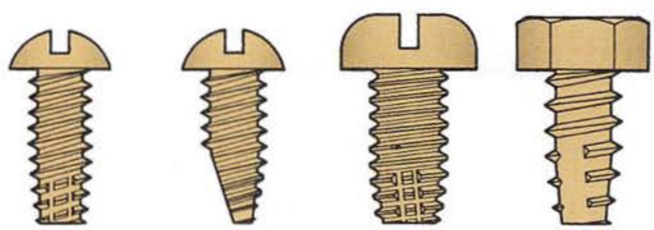
Goodheart-Willcox Publisher

Figure 8-27. Thread-forming screws.

8.1.13 Thread-Forming and Thread-Cutting Screws

Thread-forming screws produce a thread in the part as they are driven. This feature eliminates a costly tapping operation. A variation of the thread-forming screw—the self-drilling 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 8-27**.

Thread-cutting screws differ from thread-forming screws in that they actually cut threads into the material when driven. Refer to **Figure 8-28**. Thread-cutting screws are hardened. They are used to join heavy-gage sheet metal and to thread into nonferrous metal assemblies.



Goodheart-Willcox Publisher

Figure 8-28. Variations among thread-cutting screws.

8.1.14 Drive Screws

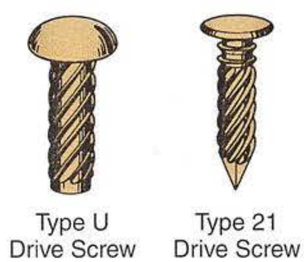
Drive screws are simply hammered into a drilled or punched hole of the proper size. Drive screws are used when an assembly will not have to be taken apart. They result in a permanent assembly. **Figure 8-29** shows two common drive screws.

8.2 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 specific advantages.

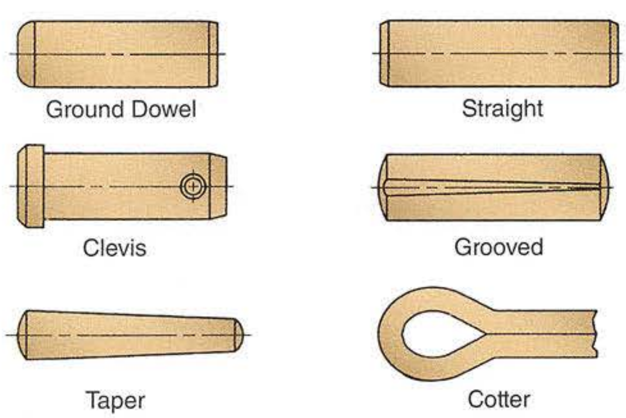
8.2.1 Dowel Pins

Dowel pins are made of heat-treated alloy steel and are found in assemblies in which parts must be accurately positioned. See **Figure 8-30**. They ensure 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.



Parker-Kalon

Figure 8-29. Drive screws are hammered or forced into place in a presized hole.



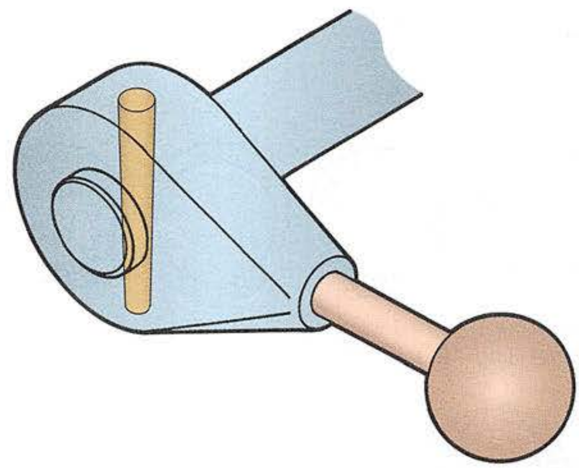
Goodheart-Willcox Publisher

Figure 8-30. Common types of dowel pins.

Regular dowel pins are 0.0002″ (0.005 mm) oversize. These have a plain steel finish. However, dowel pins are also available in 0.001″ (0.025 mm) oversize for repairs. These have a black finish to distinguish them from regular dowel pins. 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, **Figure 8-31**.

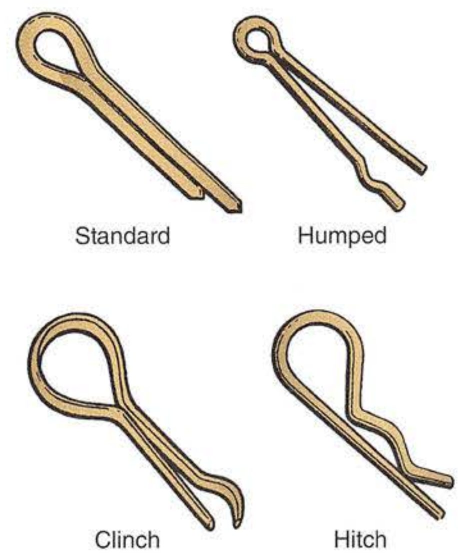
8.2.2 Cotter Pins

A cotter pin is fitted into a hole drilled crosswise through a shaft, **Figure 8-32**. The pin prevents parts from slipping or rotating off. Other types of retaining devices are replacing the cotter pin.



Goodheart-Willcox Publisher

Figure 8-31. A taper pin is often used to lock a handle to its shaft.



Goodheart-Willcox Publisher

Figure 8-32. Types of cotter pins.

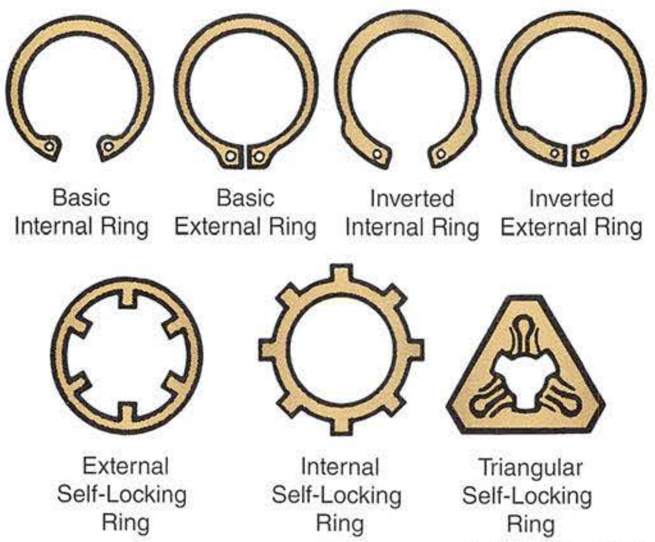
8.2.3 Retaining Rings

The retaining ring, **Figure 8-33**, has been developed for both internal and external applications. Retaining rings reduce both the cost and the weight of the product on which they are used. Most retaining rings must be seated in grooves, **Figure 8-34**, but the self-locking type does not require this special recess. Special pliers are needed for rapid installation and removal of some retaining rings, **Figure 8-35**.

8.2.4 Rivets

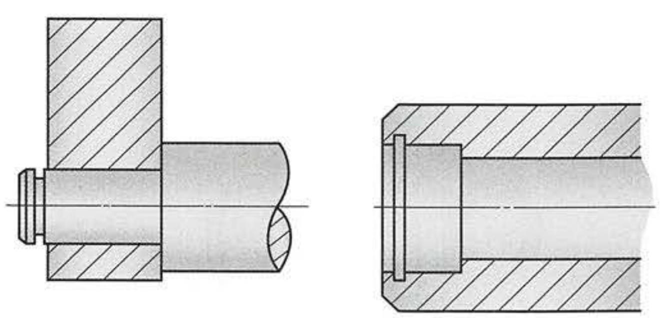
Permanent assemblies can be made with rivets, **Figure 8-36**. Solid rivets can be set, or deformed to become larger on one end, by hand or machine methods.

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 8-37**. Common types of blind rivets are shown in **Figure 8-38**.



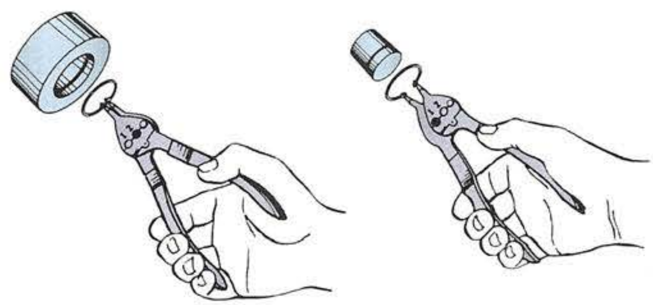
Goodheart-Willcox Publisher

Figure 8-33. Retaining rings.



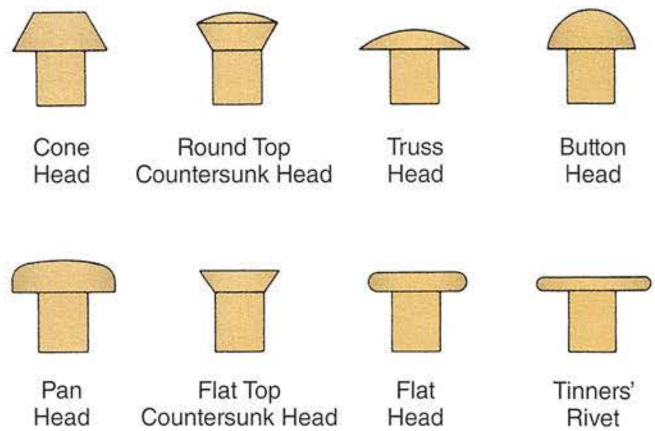
Goodheart-Willcox Publisher

Figure 8-34. Grooves are machined into parts to receive retaining rings. They eliminate many other expensive machining operations.



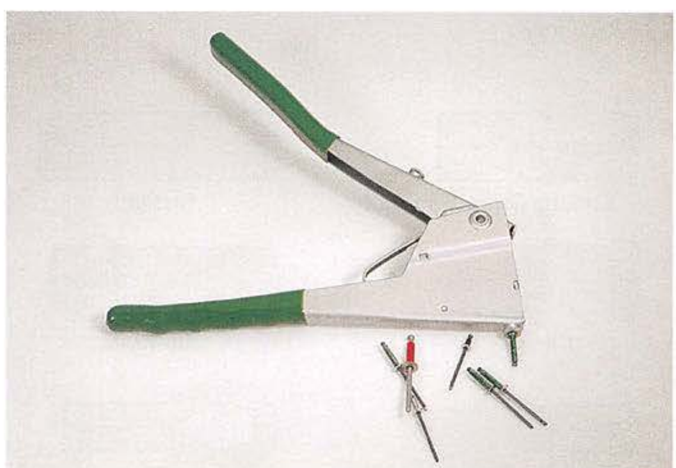
Waldes Kohinoor Inc.

Figure 8-35. Special pliers are used to install some types of retaining rings.



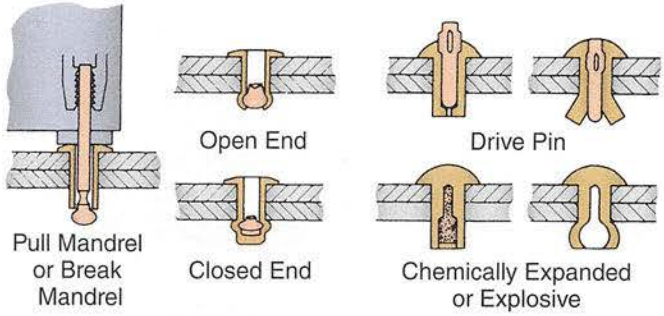
Goodheart-Willcox Publisher

Figure 8-36. Rivet head styles.



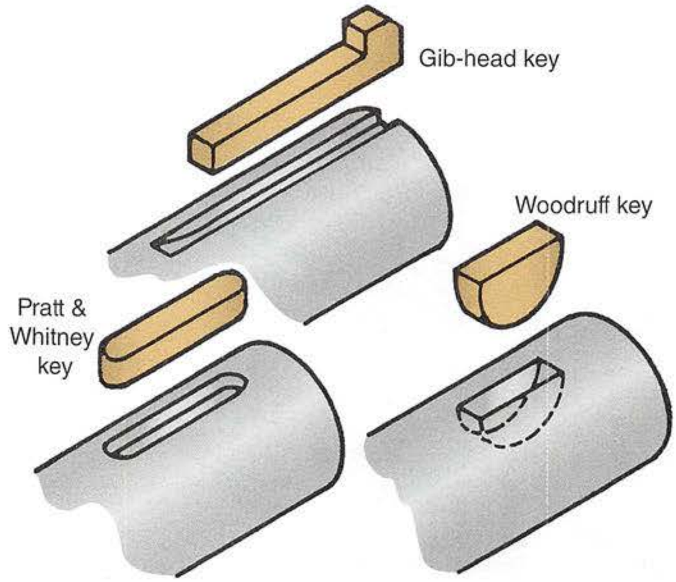
Goodheart-Willcox Publisher

Figure 8-37. A pliers-type rivet gun is used to insert one type of blind rivet.



Goodheart-Willcox Publisher

Figure 8-38. Types of blind rivets.



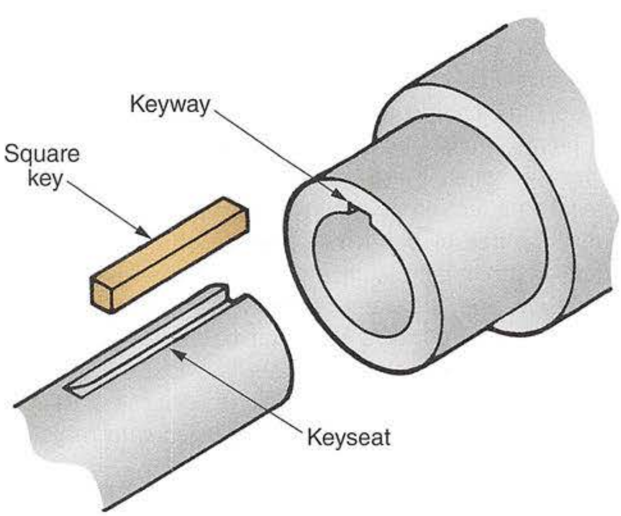
Goodheart-Willcox Publisher

Figure 8-40. Three types of keys.

8.2.5 Keys

A ***key*** is a small piece of metal that prevents a gear or pulley from rotating on its shaft. Half of the key fits into a keyseat on the shaft, and the other half fits into a keyway in the hub of the gear or pulley, **Figure 8-39**. Commonly used keys are shown in **Figure 8-40**.

- 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 it is rounded at both ends. It fits into a keyseat of the same shape.
- 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.



Goodheart-Willcox Publisher

Figure 8-39. A square key is used to prevent a pulley or gear from rotating on its shaft.

8.3 Adhesives

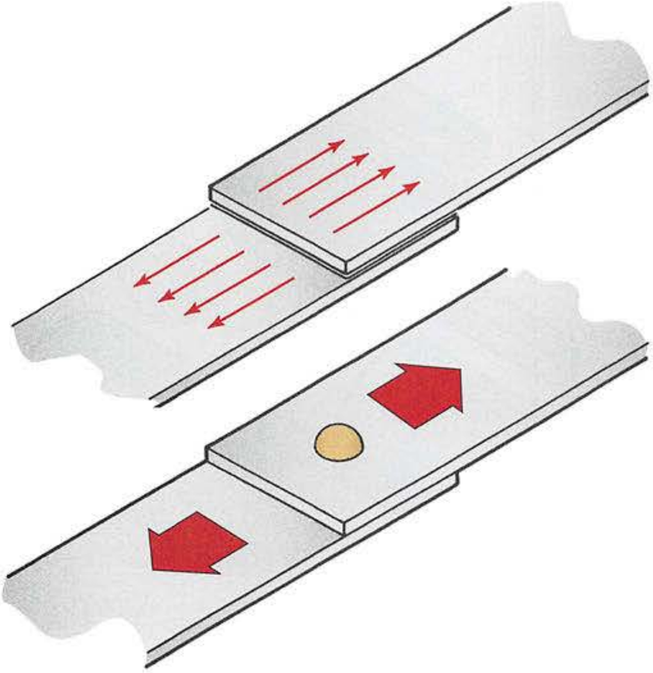
Adhesives provide another way to join metals and to keep threaded fasteners from vibrating loose. In some applications, the resulting joints are stronger than the metal itself. Adhesive-bonded joints do not require costly and time-consuming operations such as drilling, countersinking, riveting, or welding.

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 determines the type of adhesive used.

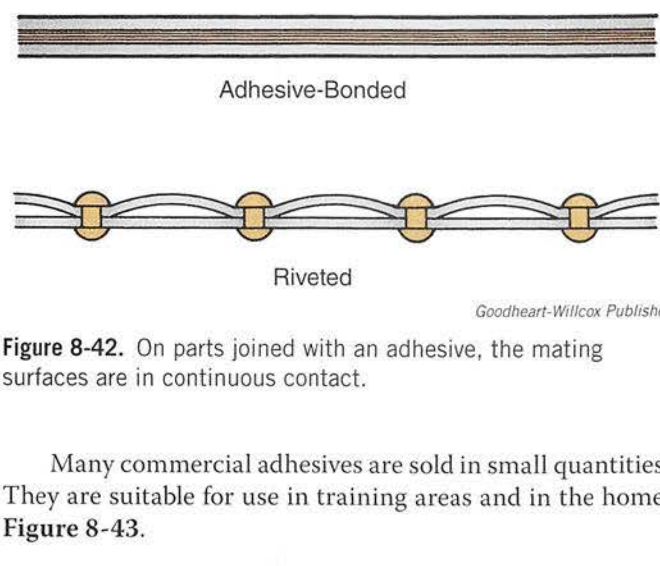
Adhesive-bonded assemblies offer many advantages over other fastening techniques:

- The load is distributed evenly over the entire joined area, **Figure 8-41**.
- There is continuous contact between the mating surfaces, **Figure 8-42**.
- Full strength of the mating parts is maintained, because holes do not have to be made to insert fasteners. The extreme heat required for joining methods, such as welding, is not necessary with adhesive bonding. This means there is no danger of heat distorting the work or affecting its heat treatment.
- 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.



Goodheart-Willcox Publisher

Figure 8-41. 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.



Goodheart-Willcox Publisher

Figure 8-42. On parts joined with an adhesive, the mating surfaces are in continuous contact.

Many commercial adhesives are sold in small quantities. They are suitable for use in training areas and in the home, **Figure 8-43**.

8.3.1 Types of Adhesives

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 that is being used increasingly in machining technology to make temporary bonds is

Goodheart-Willcox Publisher

Figure 8-43. 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.

cyanoacrylate quick-setting adhesive. Known by trade names such as Eastman 910®, Super Glue®, and Krazy Glue®, this type of adhesive is 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. Fragile parts can be glued to backup blocks for machining.

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.

For successful use of cyanoacrylate adhesives, the part and mounting surface must be prepared according to the adhesive manufacturer's directions.

GREEN MACHINING

Environmentally Friendly Adhesives

Manufacturing adhesives are considered eco-friendly, or green, when they meet several criteria. First, green adhesives have low or no VOC emissions. VOCs, or volatile organic compounds, are chemicals frequently found in paints, glues, and other coatings. VOCs are released as gases when these products are used and have a negative effect on air quality. Second, green adhesives must be free of petrochemicals, which are chemicals derived from petroleum or natural gas. Third, green adhesives may be water-based or use only a relatively small amount of solvent in their bases. Using little or no solvent makes the adhesives less toxic, less flammable, easier to store, and easier to dispose of safely. Last, green adhesives are also packaged with recycled or recyclable materials.

SAFETY NOTE

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. Unless a suitable solvent is available, surgery might be needed to separate the joined fingers. Should you get adhesive in your eyes, see a physician immediately.

8.3.2 Using Adhesives

Most adhesives require a five-step process to produce solidly bonded joints:

1. **Prepare the surface.** 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.

SAFETY NOTE

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.

2. **Prepare the adhesive.** The adhesive must be properly mixed and delivered to the work area, and equipment must be set up according to the manufacturer's directions.

SAFETY NOTE

Carefully follow all instructions on the adhesive container when mixing and using adhesives. Mix only the amount you will need. Promptly remove any adhesive from your skin by washing in water.

3. **Apply the adhesive.** Adhesive application may be done by brushing, rolling, spraying, dipping, or methods designed for a specific assembly.

SAFETY NOTE

Do not apply adhesives near an open flame. Solvents used in some adhesives are highly flammable or toxic. Apply them only in well-ventilated areas, and wear a suitable respirator.

4. **Assemble the materials.** Assembly involves positioning the materials to be joined. This often requires the use of jigs or fixtures for alignment.
5. **Allow the bond to develop.** Bond development is the process of evaporation of solvents and curing of the adhesive. It may involve application of pressure or heat.

Chapter Review

Summary

- Threaded fasteners include machine screws, machine bolts, cap screws, setscrews, stud bolts, eye bolts, thread-forming screws, thread-cutting screws, and drive screws.
- Nuts, inserts, and washers are commonly used with threaded fasteners.
- US Conventional and metric fasteners are *not* interchangeable.
- Nonthreaded fasteners include dowel pins, cotter pins, retaining rings, rivets, and keys.
- Adhesives for use on metals are available in liquid, paste, or solid form.

Review Questions

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

1. For maximum strength, a threaded fastener should screw into its mating part a distance equal to _____ times the diameter of the thread.
2. List four types of threaded fasteners. Briefly 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. To prevent a pulley from slipping on a shaft, a(n) _____ is often used.
6. The _____ bolt is threaded at both ends.
7. What can be done to make the removal of stubborn sheared bolts easier?
8. When is a jam nut used?
9. The shape of the _____ nut permits it to be loosened and tightened without a wrench.
10. Why are lock washers used?
11. Drive screws and rivets can be used to create a(n) _____ assembly.
12. While most _____ must be seated in grooves, a self-locking type does not require the special recess.
13. What advantages do adhesives offer over other fastening techniques?
    A. The load is distributed evenly over the entire area.
    B. There is continuous contact between the mating surfaces.
    C. The full strength of the mating parts is maintained.
    D. No external projections result in smooth surfaces.
    E. All of the above.
14. Briefly describe, in order, the steps that must be used to join metals with adhesives.
15. List at least three safety precautions that must be observed when using adhesives.

Match each brief description with the word it most accurately describes.

16. Developed for use in a confined area, where a joint is only accessible from one side.
17. Used where parts must be aligned accurately and held in absolute relation with one another.
18. Prevents a pulley or gear from slipping on a shaft.
19. Protects projecting threads.
20. Is hammered into a drilled or punched hole.
21. Used to make permanent assemblies.
22. Slot cut in a gear or pulley to receive a key.
23. Locks a regular nut in place.
24. Eliminates costly tapping operations.
25. Slot cut in a shaft to receive a key.

A. Rivet
B. Jam nut
C. Drive screw
D. Thread-cutting screw
E. Acorn nut
F. Dowel pin
G. Blind rivet
H. Keyway
I. Keyseat
J. Key

CHAPTER 9

Jigs and Fixtures

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Describe the two main types of jigs.
- Explain how fixtures are used.
- Explain how a tombstone is used with CNC machine tools.
- Identify suitable materials for making jigs and fixtures.

Technical Terms

bushing

fixture

jig

thermal expansion

tombstone

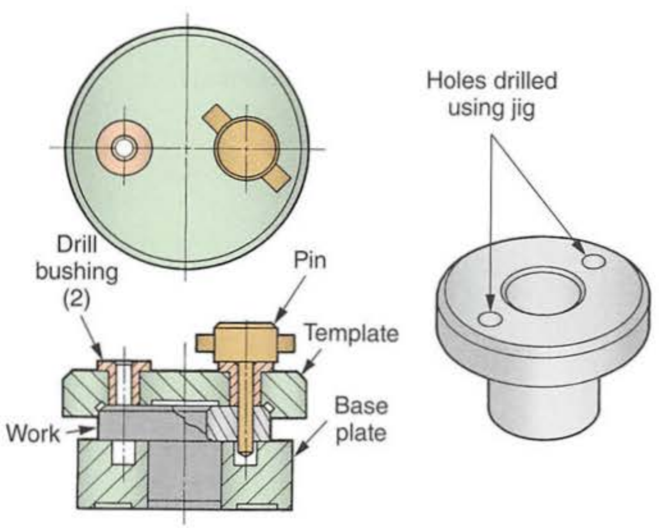
Jigs and fixtures are used extensively in production machine shops to hold work while machining or measuring operations are performed. When large numbers of identical and interchangeable parts must be produced and measured, they position the workpiece precisely so that all of the parts produced are within specifications and as uniform as possible. In general, the use of jigs and fixtures helps reduce errors which, in turn, reduces manufacturing costs. During measurement operations, they help reduce measurement error. Even when production quantities are small, the use of jigs and fixtures is often justified because they allow relatively unskilled workers to operate the machines.

Jigs and fixtures are also used in assembly work for operations such as welding or riveting. They position and hold the work to make the fabrication of standardized parts feasible.

9.1 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 9-1**.

The jig is seldom mounted solidly to the drill press table. For safety, however, it is usually placed between guide bars that are mounted solidly to the table, **Figure 9-2**.

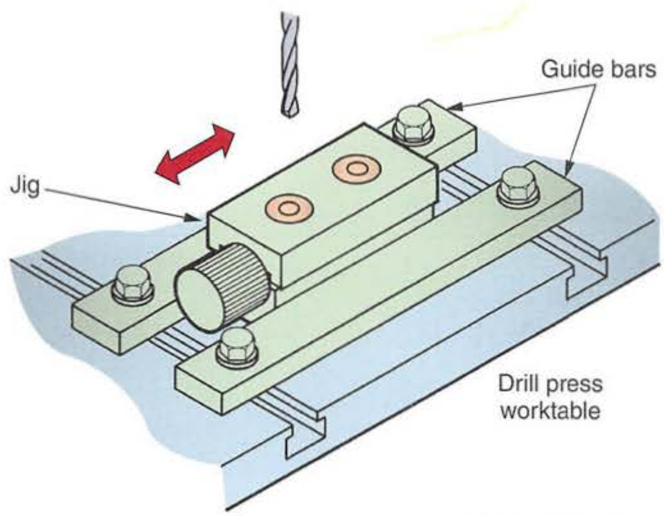


Goodheart-Willcox Publisher

Figure 9-1. With this circular drill jig, a pin is placed into the first hole after it is drilled. This holds the workpiece in position while the second hole is drilled. The base plate provides clearance for the drill as it breaks through the workpiece.

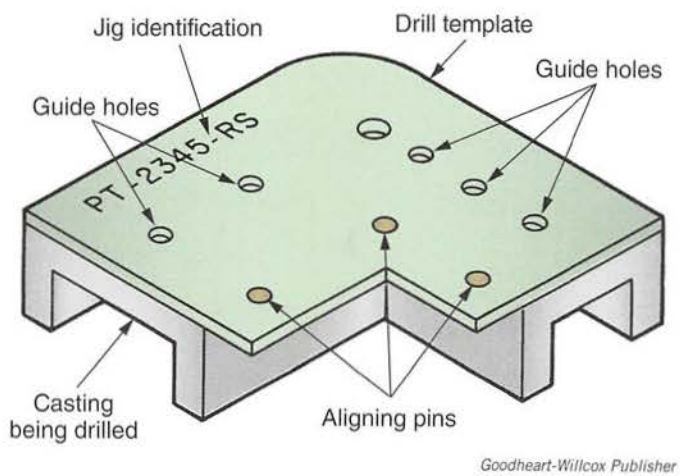
There are two general types of drill jigs: open jigs and box (or closed) jigs. The drill template or plate jig is the simplest form of the open jig. It consists of a plate with holes to guide the drill. The jig fits over the work, **Figure 9-3**.

In a more elaborate form of an open drill jig, **Figure 9-4**, 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.



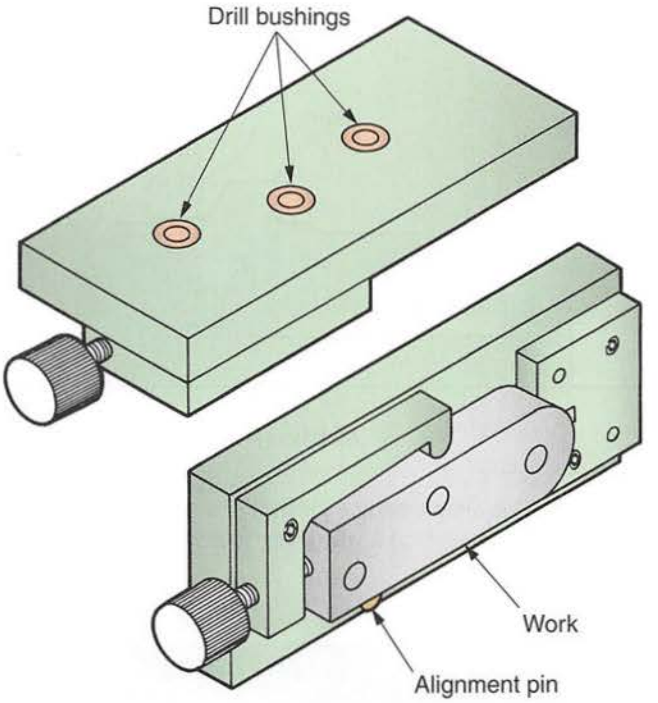
Goodheart-Willcox Publisher

Figure 9-2. A drill jig is placed between guide bars to prevent dangerous and undesirable "merry-go-round" rotation.



Goodheart-Willcox Publisher

Figure 9-3. A simple drill template. Identification numbers on jigs and fixtures allow these devices to be located easily when stored away between uses.



Goodheart-Willcox Publisher

Figure 9-4. This open 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.

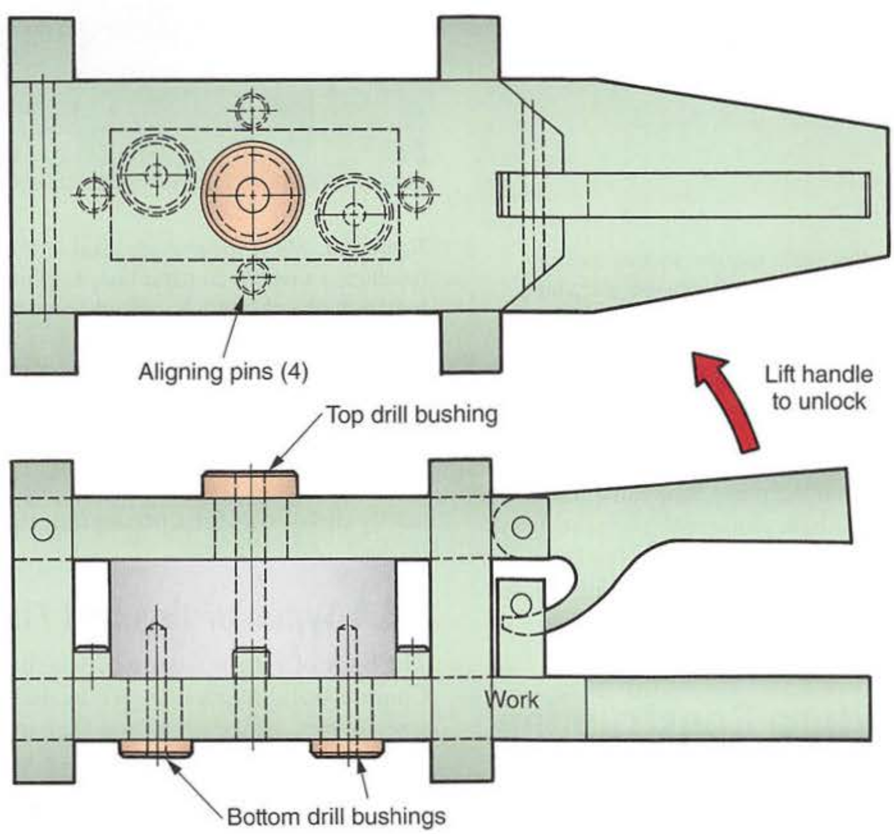
The box jig encloses the work, **Figure 9-5**. This type is more costly to make than an open jig, but it is often used when holes must be drilled in several directions. **Figure 9-6** illustrates a box jig in its simplest form. The work is fitted into the jig through a hinged or swinging cover. The clamps that hold the work in place are permanently mounted to the jig.

When several different operations must be performed on a job, a combination of open and box jigs is often used. Slip bushings are used to guide the drills. The slip bushings are then removed for subsequent operations, such as reaming, tapping, countersinking, counterboring, or spotfacing.

9.2 Fixtures

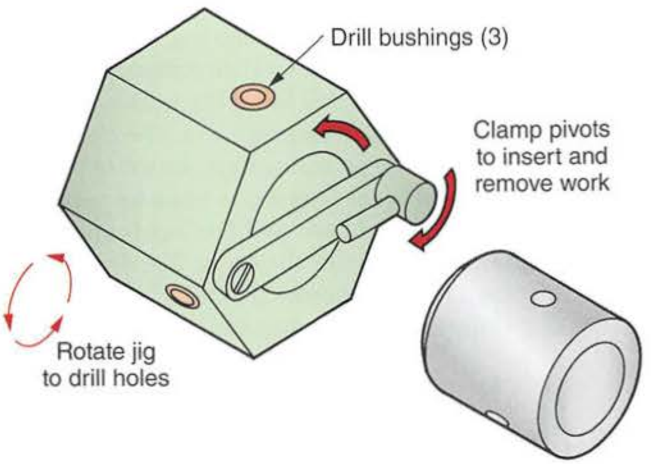
A ***fixture*** is used to position and hold a workpiece while machining or measurement operations are performed on it, **Figure 9-7**. Unlike a jig, a fixture does not guide the cutting tools.

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, milling machine (vertical or horizontal), lathe, band saw, or grinder. Fixture designs range from simple vise jaw modifications, **Figure 9-8**, to the very large, complex devices used by the aerospace industry, **Figure 9-9**.



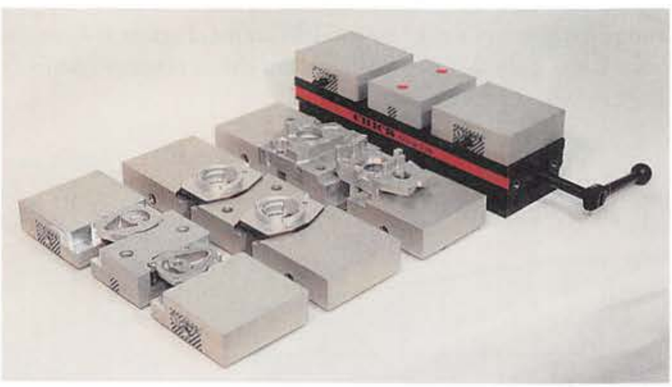
Goodheart-Willcox Publisher

Figure 9-5. A box drill jig. Lowering the handle locks the work in the jig.



Goodheart-Willcox Publisher

Figure 9-6. A light box jig used to drill three equally spaced holes in a base end cap. Since only a limited production run was required for this product, it was not necessary to construct a more elaborate jig.



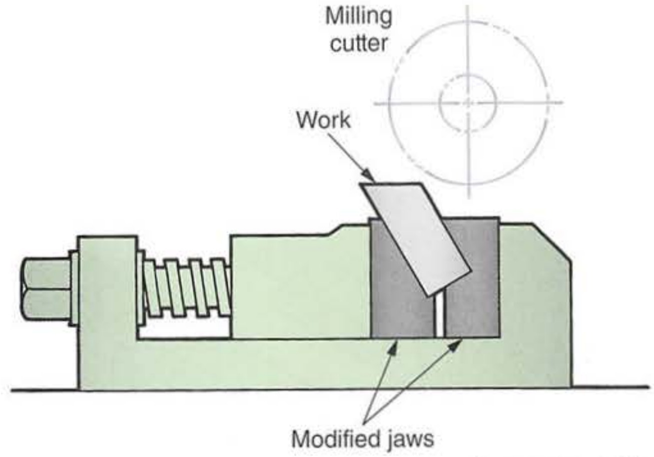
Chick Machine Tool, Inc.

Figure 9-7. 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 work-holding system, are shown in the background. New setups can be made in a very short period of time.

Fixtures used for measuring parts may have indicators or may be mounted onto the base along with the nesting system to measure parts. They can also be used to hold parts for automated measurement, such as with a coordinate measuring machine.

9.3 Jig and Fixture Construction

Jigs and fixtures 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.



Goodheart-Willcox Publisher

Figure 9-8. 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.

Peter R Foster IDMA/Shutterstock.com

Figure 9-9. Many fixtures are used in the assembly of an aircraft fuselage. In fixture construction, extreme accuracy is critical, requiring use of lasers to ensure precise alignment.

For short production runs, jig and fixture kits can be used to create custom nests to hold workpieces for machining and measurement. The costs associated with building custom jigs and fixtures can be high, and it is difficult to justify these costs for a production run of a small number of pieces.

9.3.1 Types of Jigs and Fixtures

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 9-10**. Fixture-holding devices have been developed for machining centers and other CNC machine tools that permit multiple setups, **Figure 9-11**. A ***tombstone*** is a fixture-holding device that is commonly used with CNC machine tools, **Figure 9-12**. Tombstones are made from heavy castings and are precisely

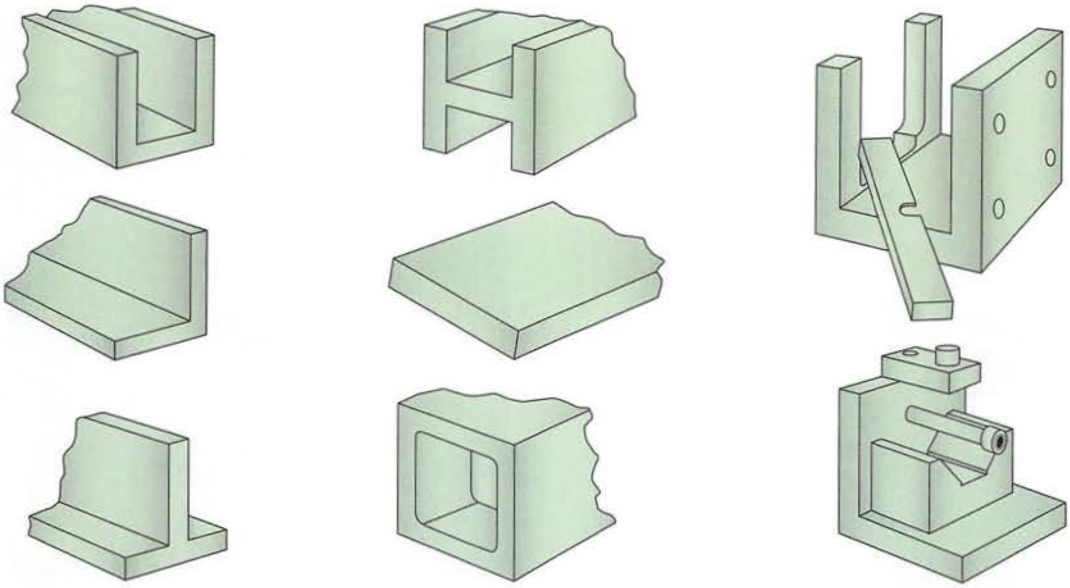


Figure 9-10. Standard cast-iron shapes are machined parallel and square to save time and money in designing and building jigs and fixtures. Sections of different shapes can be bolted together to form complex holding devices. Two completed units are shown at the right of the illustration.

WORKPLACE SKILLS

Preparing for a Job Interview

A job interview gives you the opportunity to learn more about a company and to convince the employer that you are the best person for the position. The employer wants to know if you have the skills needed for the job. Adequate preparation is essential for making a lasting, positive impression. The following are some ways to prepare for the interview:

- **Research the employer and the job.** Know the mission of the employer and specifics about the job. Also, try to learn what the company looks for when hiring new employees.
- **List the questions you want answered.** For example, do you want to know if there is on-the-job training? Have several questions prepared in advance. When the employer asks if you have questions, saying no can indicate a lack of interest or preparation.
- **Decide what to wear.** Dress appropriately, usually one step above what is worn by your future coworkers. For instance, casual clothing is acceptable for individuals who will do manual labor or wear a company uniform. If the job involves greeting the public or working in an office environment, a suit may be more appropriate. Always appear neat and clean.
- **Practice the interview.** Have a friend or family member interview you or practice in front of a mirror until you are happy with your responses.
- **Know where to go for the interview.** Verify the address of the interview location and plan to arrive at least 10 minutes early.

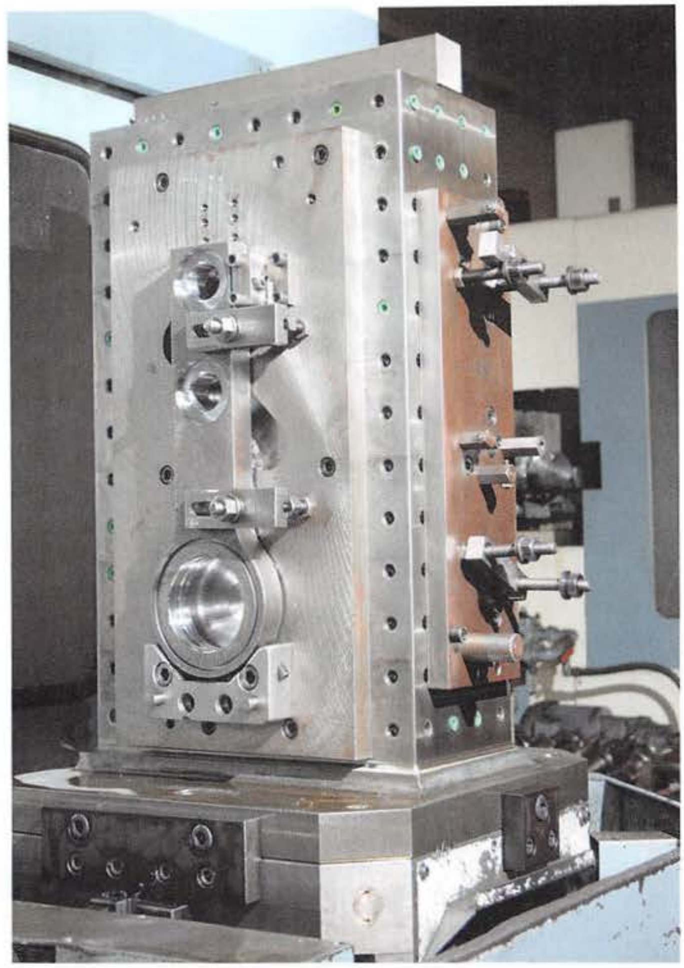
Good preparation will make you feel more confident and comfortable during the interview. Be polite, friendly, and cheerful during the process. Answer all questions carefully and as completely as you can. Be honest about your abilities.

A prospective employer may ask you to take employee tests. Some employers administer tests to job candidates to measure their knowledge or skill level under stress. Since all employers support a drug-free workplace, most will likely require you to take a drug test if hired. You can ask those who have completed similar tests what to expect.

After the interview, send a letter to the employer within 24 hours, thanking him or her for the interview. If you get a job offer, respond to it quickly. If you do not receive an offer after several interviews, evaluate your interview techniques and seek ways to improve them.

© Copyright Jergens, Inc. 2013

Figure 9-11. Fixture-holding devices for machining centers and other CNC machine tools. Work-holding pockets are cut directly into the jaw blocks. Pivoting the vertical setup brings the next set of workpieces into position.



Tibor Machine Products

Figure 9-12. A typical tombstone with drilled and tapped mounting holes. The work-holding fixtures have been mounted directly to the tombstone. A well-designed fixture holds the workpiece securely and accurately, while allowing quick part changes.

machined. They are mounted on a machine's worktable. Fixtures are then mounted directly onto the tombstones. The tombstone may be blank, have drilled and tapped mounting holes, or have T-slots for mounting fixtures.

9.3.2 Custom Jigs and Fixtures

Jigs and fixtures must be precisely designed and manufactured because they contribute to the accuracy of the final product. Whether a jig or fixture is used for machining or measurement, the more precisely it has been made, the less machining or measurement error will occur when it is used. The quality of the machining and measurement of the parts held in the jig or fixture reflect the quality of the nesting system.

Common materials for jigs and fixtures include aluminum, steel, and hardened steel. If the machining tolerances for the part are large, aluminum components may be used. However, if the tolerances are tight, aluminum is not a good choice because of the thermal expansion of aluminum. The ***thermal expansion*** of a material is the rate at which the material expands or contracts based on the surrounding temperature. The thermal expansion of a basic aluminum alloy is 0.000013″ (13 microinches) per inch of length per one degree of temperature change on the Fahrenheit scale. For example, consider a fixture base that is six inches long when the material is at 72° F. If the temperature increases to 82° F during the day, then the aluminum plate expands by:

$$0.000013'' \times 6'' \times 10^\circ = 0.00078''$$

This amount of change can reduce the accuracy of machining and measurement operations that use the jig or fixture. Therefore, hardened steels whose thermal expansions are less than half that of aluminum are used for jig and fixture components for precision tolerance machining and measurement. Also, carbide inserts are usually used for the surfaces that contact the part during machining. Carbide is harder and more resistant to wear, so the jig or fixture requires fewer inspections and repairs over long production runs.

Chapter Review

Summary

- Jigs and fixtures hold the workpiece during machining operations.
- Open and box jigs guide drills or cutting tools.
- Fixtures hold the work in place but do not guide the machine tools.
- Fixture-holding devices for CNC machine tools allow multiple setups.
- When parts are to be made to tight tolerances, jigs and fixtures must be made of a material, such as hardened steel, that has a small rate of thermal expansion.

Review Questions

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

1. Why are jigs and fixtures used in production machine shops when large numbers of identical parts must be produced?
2. *True or False?* Jigs and fixtures allow relatively unskilled workers to operate machines.
3. Jigs and fixtures _____.
   A. hold the work
   B. position the work
   C. are used in assembly work
   D. All of the above.
   E. None of the above.
4. What is a jig?
5. Hardened steel _____ in a jig guide the drill or cutting tool during machine operation.
6. Jigs fall into two general types. List and briefly describe each type.
7. When using a combination of open and box jigs, _____ bushings are used to guide the drills and then removed for subsequent operations.
8. What is a fixture?
9. *True or False?* Fixtures are used to guide cutting tools.
10. The common fixture-holding devices made of heavy, precisely machined castings are known as _____.
11. Why can jigs and fixtures be made of aluminum only if the required precision for the part to be manufactured is low?
12. What is the purpose of a carbide insert on a jig or fixture?

CHAPTER **10**

Cutting Fluids

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Understand why cutting fluids are necessary.
- List the four general types of cutting fluids.
- Describe each type of cutting fluid.
- Discuss how cutting fluids should be applied.
- Understand the process for selecting a cutting fluid.
- Explain where to find safety and hazard information about cutting fluids.

Technical Terms

bacteriostatic additive
chemical cutting fluid
cutting fluid
emulsifiable oil
gaseous fluid
mineral oil
semichemical cutting fluid

Cutting fluids are substances formulated to help cool and lubricate various metalworking tasks. They are designed to perform the following functions:

- Cool the work and cutting tool, **Figure 10-1**.
- Improve surface finish quality.
- Lubricate to reduce friction and cutting forces, thereby extending tool life.
- Minimize material buildup on tool cutting edges.
- Protect machined surface against corrosion.
- Flush away chips, **Figure 10-2**.

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.

GREEN MACHINING

Health and Environmental Impacts of Cutting Fluids

Traditional cutting fluids can produce hazards for workers and the environment. Fluids can become aerosols during machining and be breathed in by machinists, harming the lungs and skin. Cutting fluids also create environmental challenges after use when they are disposed of as waste. Waste products from cutting fluids can pollute water and other natural environments. These drawbacks can be reduced or eliminated by using "green" cutting fluids. New environmentally friendly, vegetable-based cutting fluids are safer for machinists, tend to last longer, and offer other advantages to machine processes over mineral oils or chemical fluids.

BIG Kaiser Precision Tooling Inc.

Figure 10-1. To provide the most efficient heat removal, coolant fluid should flood the cutting tool and the entire area being machined.

MAG IAS, LLC

Figure 10-2. An additional function of cutting fluids is to flush away chips from the area where cutting is taking place.

10.1 Types of Cutting Fluids

There are four basic types of cutting fluids. These include mineral oils, emulsifiable (water-based) oils, chemical and semichemical fluids, and gaseous fluids.

10.1.1 Mineral Oils

Cutting fluids made from mineral oil may be used straight or combined with additives. Straight ***mineral oils*** are excellent lubricants, but they do not dissipate heat well. They are a good choice for light-duty (low speed, light feed) operations.

Mineral oil cutting fluids offer the following advantages:

- They are noncorrosive.
- They have excellent lubricating properties.
- They are more environmentally friendly than most cutting fluids.

Their disadvantages include the following:

- They have the poorest cooling capabilities of any type of cutting fluid.
- They are more costly than other types of cutting fluids.
- They pose a greater danger from smoke and fire.
- They can cause health problems in operators if not managed properly.

Mineral oils have a tendency to become rancid, so the tank containing them must be cleaned periodically and

WORKPLACE SKILLS

Evaluating Job Offers

When considering a job offer or comparing two or more positions, you should explore the following work factors:

- **Physical surroundings.** Where is your workspace located? Is the atmosphere conducive to your style of working? Is parking provided? Is public transportation close by?
- **Work schedule.** Will the workdays and work hours mesh with your lifestyle? Is occasional overtime work required?
- **Income and benefits.** Is the proposed salary fair? Will you receive benefits that are just as valuable as extra income? How much sick leave is granted during the year? Is personal or emergency leave available? What is the vacation leave policy? Are there medical and life insurance benefits? Is there a credit union? Will the company pay tuition for college courses or special programs related to your job? Is a cafeteria on the premises? Does it offer food to employees at reduced cost?
- **Job obligations.** Will you be expected to join a union or other professional organization? If so, what are the costs? Will you be expected to attend meetings after work?
- **Advancement potential.** Is there opportunity for advancement? After demonstrating good performance, how soon can you seek a position with more responsibilities? Before you can advance, are there special expectations, such as a higher degree? Are training programs provided?

Talking about advancement requires considerable diplomacy. After all, you should not appear too eager to leave the job for which you are interviewing. Many employers expect a new employee to remain at least one year at that job. If you place undue emphasis on advancement, you will appear uninterested in the current opening.

the fluid replaced. ***Bacteriostatic additives*** may be used to inhibit the growth of bacteria.

SAFETY NOTE

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.

10.1.2 Emulsifiable Oils

Emulsifiable oils, also known as *soluble oils*, 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 can be used in most moderate- and light-duty machining operations. For economy and best machining results, these oils must be mixed according to the manufacturer's recommendations. The manufacturer's recommendations take into account the material being machined and the machining operation performed.

Emulsifiable cutting fluids offer the following advantages:

- They provide increased cooling capacity in some applications.
- They are cleaner to work with than other cutting fluids.
- They provide cooler and cleaner parts for the machinist to handle.
- They are less expensive than other types of cutting fluids.
- They present no fire hazard.

Emulsifiable oils have the following disadvantages:

- Fluid maintenance must be performed on a routine basis to control rancidity. (Bacteriostatic additives can be used to control bacterial growth.)
- Since water is mixed with the oils, the risk of corrosion is present.

In addition, the fluid must be monitored to ensure the ratio of oil and water is properly maintained. During use, the water tends to evaporate, so water needs to be added to keep the oil in solution.

SAFETY NOTE

Water-based cutting fluids must never be used when machining magnesium. Magnesium dust and chips are flammable, and burning magnesium reacts violently with water.

10.1.3 Chemical and Semichemical Cutting Fluids

Chemical cutting fluids contain no oil and are not actually liquids. Examples include graphite, mica, and white lead. 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 chemical and emulsifiable oil cutting fluids.

Chemical and semichemical cutting fluids offer the following advantages:

- They dissipate heat rapidly.
- They are clean to use.
- Residue is easy to remove after machining.
- They are easy to mix and do not become rancid.

Their disadvantages include the following:

- Some formulas have minimal lubricating qualities.
- Fluids may cause skin irritation in some workers. Material safety information should always be kept in the shop when using chemical or semichemical cutting fluids.
- When they become contaminated with other oils, disposal can be a problem.

10.1.4 Gaseous Fluids

Compressed air is the most commonly used ***gaseous fluid*** coolant. It cools by forced convection (the transfer of heat through the mechanical movement of a gas or liquid). Different types of gases—including compressed shop air, nitrogen, and carbon dioxide—can be used in compressed form.

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. Gaseous fluids work well for machining nonmetals with low melting points, such as plastics and wax bases.

Cooling with gaseous fluids offers the following advantages:

- They clear the cut area of removed material.
- Their use eliminates environmental concerns associated with other types of cutting fluids.

Their disadvantages include the following:

- Gaseous fluids have no lubricating qualities.
- They may not adequately cool the work area between the cutter and the workpiece.
- As compressed gases blow chips away, the chips are blown at high velocity, requiring workers in surrounding areas to be shielded from the flying chips.

SAFETY NOTE

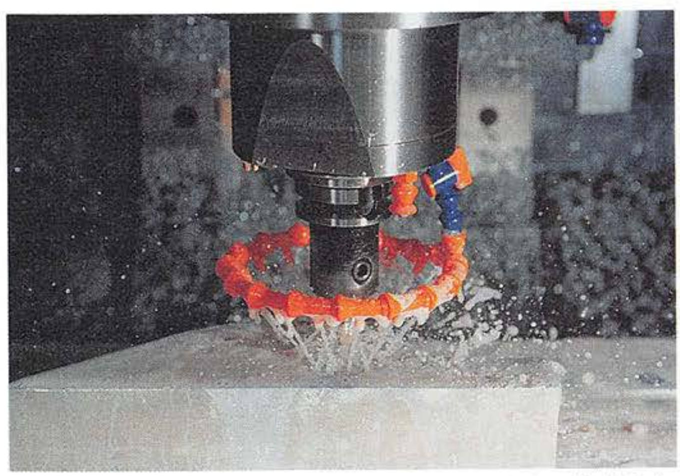
Never direct compressed air at yourself or another person. Compressed air can enter the body through a cut or opening and cause an air bubble to form in the bloodstream, leading to a potentially fatal blockage of a blood vessel.

10.2 Application of Cutting Fluids

Machining and grinding applications require a continuous flood of fluid around the cutting tool and work to provide efficient removal of the heat generated, **Figure 10-3**. Coolant nozzles must be positioned carefully so that, in addition to cooling the work area, the cutting fluid carries the chips away. In some machining operations, a conveyor system, **Figure 10-4**, is used to remove chips and cutting fluid from the cutting area. The cutting fluid is filtered to remove contaminants (unwanted debris) and returned to the machine's coolant tank for reuse.

10.3 Choosing a Cutting Fluid

The selection of a cutting fluid depends on the machining operation and the material being machined. Refer to the data published by cutting fluid manufacturers for recommended cutting fluids for a specific job. Recommendations



Lockwood Products, Inc.

Figure 10-3. 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.

Goodheart-Willcox Publisher

Figure 10-4. 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 from the cutting fluid. The filtered fluid is returned to the machine for reuse.

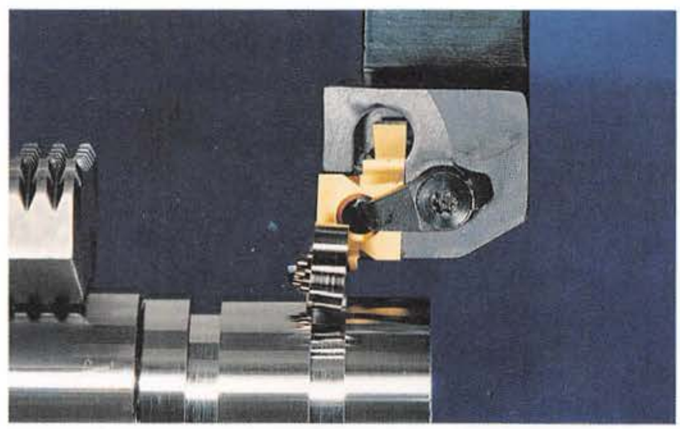
for cutting fluid use are also included in the chapters of this textbook dealing with each type of machine tool. In general, however, cutting fluids (except gaseous fluids) are compatible with high-speed steel (HSS) and carbide tooling. Since carbide tooling operates at higher cutting speeds and generates higher cutting temperatures, cutting fluids that have high cooling rates should be used in such applications. See **Figure 10-5**. Machining with ceramic tooling does not require the use of cutting fluids, **Figure 10-6**.

10.4 Cutting Fluid Safety

With the variety of cutting fluids on the market today, machinists need to be aware of the safety data sheets (SDSs) that each fluid manufacturer provides for each type of coolant they market. Each SDS documents the proper procedures for storing, handling, and eventually disposing of the fluid. When personal protective equipment (PPE) is required to handle the material safely, the SDS provides guidelines for selecting the proper equipment. SDSs for all fluids should be kept in the area where the materials are used, as well as in a central location so that they are readily available for first responders in the event of an accident.

CNC Software, Inc. © Copyright 1983-2013. All rights reserved.

Figure 10-5. Carbide cutting tools, like this end mill, are widely used, but they require coolants that can remove heat at higher rates.



Carboloy

Figure 10-6. Standard turning operation on a vertical turning center. Cutting fluids are not necessary with ceramic tooling.

Chapter Review

Summary

- Cutting fluids cool the work and the cutting tool, improve surface quality, lubricate, minimize material buildup, protect against corrosion, and flush away chips.
- The four types of cutting fluids are mineral oils, emulsifiable oils, chemical and semichemical cutting fluids, and gaseous fluids.
- The selection and application of cutting fluids depends upon the material being machined, specific machine operation, the type of cutting tool, and cutting speed and feed.
- Safety data sheets provide safety and hazard information for every cutting fluid.

Review Questions

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

1. List five functions of cutting fluids.
2. List the four basic types of cutting fluids.
3. What type of cutting fluid provides good lubrication but does not dissipate heat well?
4. Why do mineral oil cutting fluids have limited use?
5. A(n) _____ is a substance that limits the growth of bacteria.
6. What type of cutting fluids are also known as *soluble oils*?
7. What advantages do the emulsifiable oil cutting fluids have over mineral oil cutting fluids?
8. _____ cutting fluids contain no oils.
9. When small amounts of mineral oil are added to a chemical cutting fluid, the result is known as a(n) _____ cutting fluid.
10. What are the advantages of chemical and semichemical cutting fluids?
11. What is dangerous about using compressed air to cool the area being machined?
12. What does continuous flooding of the work area and cutting tool with cutting fluid accomplish?
13. Where can you find information about the recommended cutting fluid for a particular job?
14. How can you determine whether personal protective equipment is needed when working with a specific cutting fluid?

CHAPTER **11**

Sawing and Cutoff Machines

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Identify the various types of sawing and cutoff machines.
- Describe how to set up and use a power hacksaw.
- Explain how to select and install a band saw blade.
- Understand how to correct common problems with using power hacksaws and band saws.
- Identify the types of metal-cutting circular saws.
- Safely operate sawing and cutoff machines.

Technical Terms

abrasive cutoff saw
all-hard blade
cold circular saw
flexible back blade
friction saw
horizontal band saw
three-tooth rule

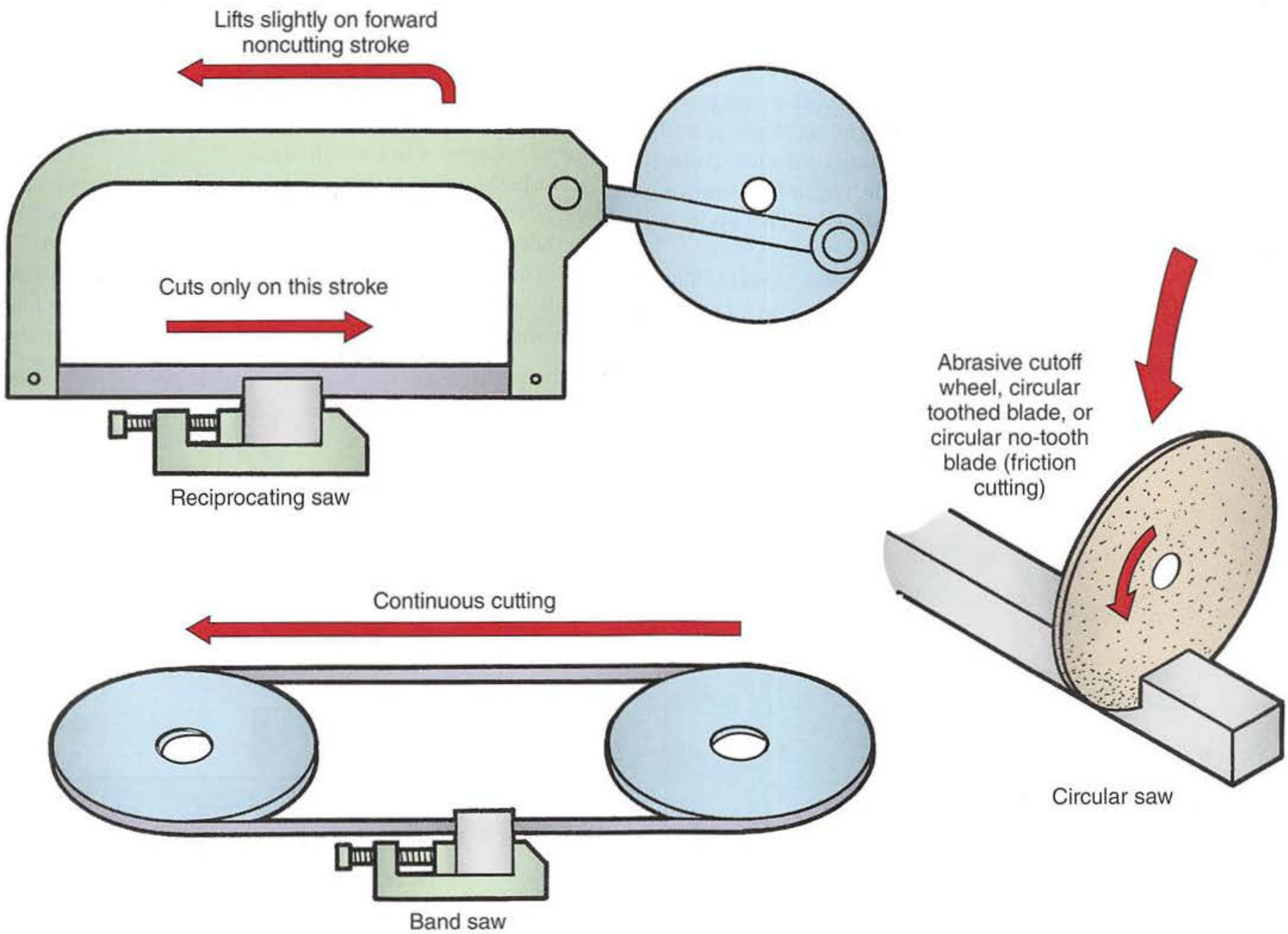
The first step in most machining jobs is to cut the stock to the required length. This can be done using power saws, **Figure 11-1**.

alterfalter/Shutterstock.com

Figure 11-1. 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.

11.1 Metal-Cutting Power Saws

There are three principal types of metal-cutting power saws, **Figure 11-2**. Reciprocating saws (power hacksaws) use a back-and-forth (reciprocating) cutting action. The cutting is done on the backstroke. The blade is similar to that of a handheld hacksaw, but it is larger and heavier. Band saws have a continuous blade that moves in one direction. Circular 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.



Goodheart-Willcox Publisher

Figure 11-2. The three principal types of metal-cutting power saws.

11.2 Power Hacksaw

A power hacksaw, **Figure 11-3**, uses a reciprocating motion to move the blade across the work. The blade cuts on the backstroke. Types of feeds available include:

- **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**—exerts a uniform pressure on the blade 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. Gravity feed is an example of definite pressure 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 power hacksaws 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. A swivel vise permits angular cuts to be made quickly, **Figure 11-4**. Quick-acting vises allow faster manual clamping of the workpiece. Power stock feed, power clamping of the workpiece, and automatic cycling can automate the cutting operation. Automatic cycling moves the work out the required distance, clamps it, and makes the cut automatically. The cycle is repeated on completion of the cut.

High-speed cutting requires the use of a coolant. The coolant reduces friction, increases blade life, and prevents chip-clogged teeth. Cast iron and some soft brass alloys, unlike most materials, do not require coolant.

Worakit Sirijinda/Shutterstock.com

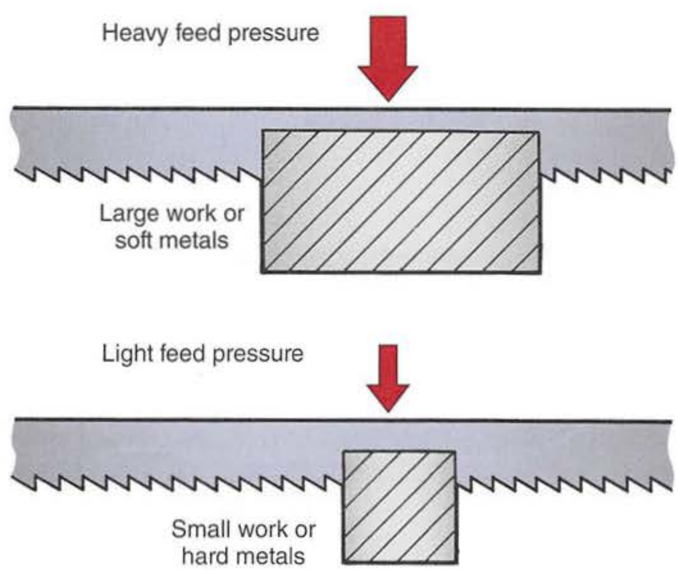
Figure 11-4. A swivel vise permits angular cuts.

11.2.1 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 at all times while cutting. Large sections and soft materials require a coarse-tooth blade. Small or thin work and hard materials require a fine-tooth blade.

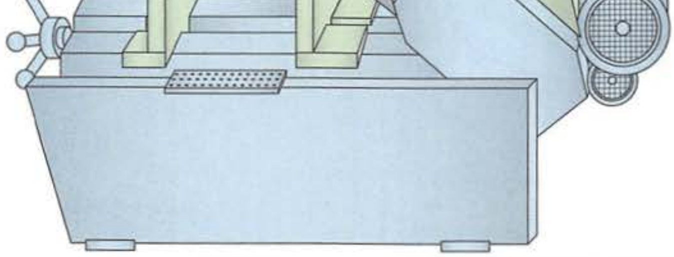
For best cutting action, apply heavy feed pressure on soft materials and large work. Use light feed pressure on hard materials and work with small cross sections, **Figure 11-5**.

Blades are made in two principal types: flexible back and all-hard. The choice depends on use. A ***flexible back blade*** should be used when safety requirements demand



Goodheart-Willcox Publisher

Figure 11-5. Apply heavy feed pressure on soft metals and large work. Use light pressure on hard metals and work with small cross sections.



Goodheart-Willcox Publisher

Figure 11-3. An industrial reciprocating power hacksaw.

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 the majority of cutting jobs, the ***all-hard blade*** is best for straight, accurate cutting under a variety of conditions.

SAFETY NOTE

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 fly apart, causing injuries.

Blades are also made from tungsten and molybdenum steels, and with tungsten carbide teeth on steel alloy backs. The following rules of thumb can be used to select the correct blade:

- Use a 4-tooth blade to cut large sections or readily machined metals.
- Use a 6-tooth blade to cut harder alloys and for 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.

11.2.2 Mounting a Power Hacksaw Blade

The blade of a hacksaw must be mounted to cut on the power (back) stroke. The blade must also lie perfectly flat against the mounting plates, **Figure 11-6**. A properly tensioned blade provides long blade life and accurate cuts.

Many techniques have been developed for properly mounting and tensioning blades. Use a torque wrench and consult the manufacturer's literature. If information about the 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 become slightly elongated, **Figure 11-7**.

The blade becomes more firmly seated after the first few cuts and stretches slightly. The blade requires retensioning (retightening) before further cutting can be done.

11.2.3 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 11-8**.

If several sections are to be cut, use a stop gage, **Figure 11-9**. Apply an ample supply of coolant if the machine has a built-in coolant system.

11.3 Power Band Saw

The ***horizontal band saw***, **Figure 11-10**, is frequently referred to as a *cutoff machine*. It offers the following advantages over the power hacksaw:

- **Greater precision.** The blade on a band saw can be guided more accurately than the blade on a power hacksaw. It is common practice to cut directly "on the line" when band sawing because finer blades can be used.

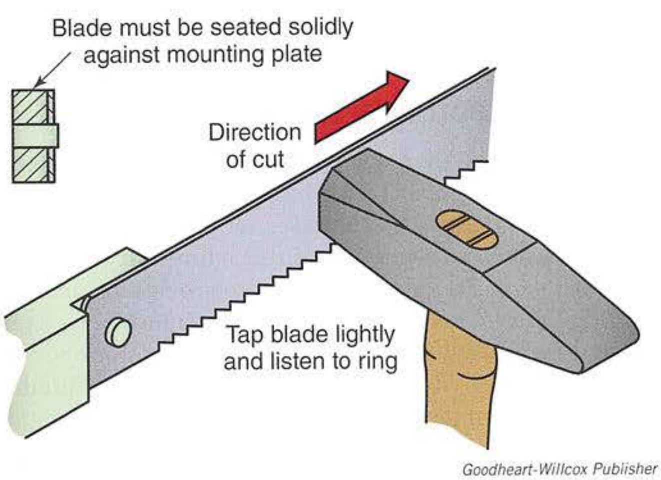


Figure 11-6. The blade must be adjusted to cut on the backstroke. Make sure it is perfectly flat against the mounting plates before tensioning. Tighten the blade until a low musical ring is heard when the blade is tapped with a small hammer. Since blades have a tendency to stretch slightly after making a few cuts, tension should be checked and, if necessary, adjusted.

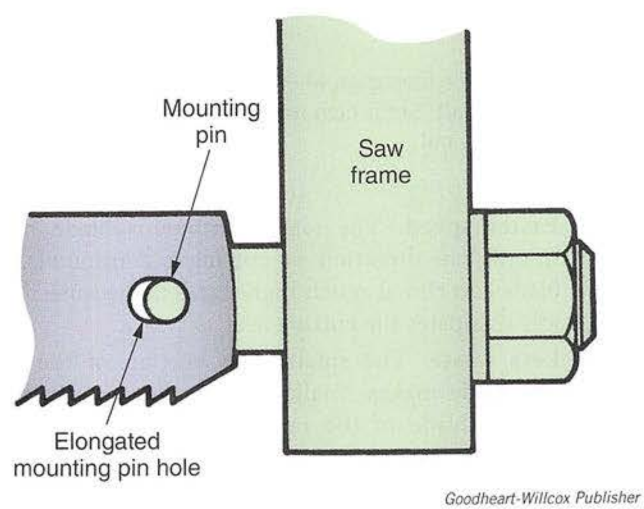
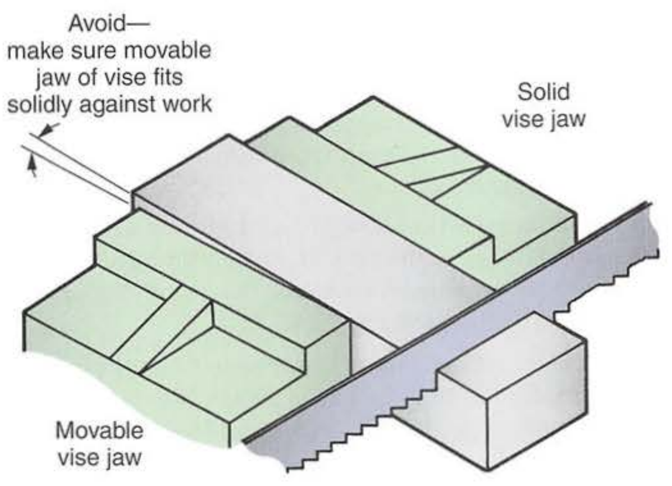


Figure 11-7. Pin holes on a properly tensioned blade are slightly elongated, rather than round.

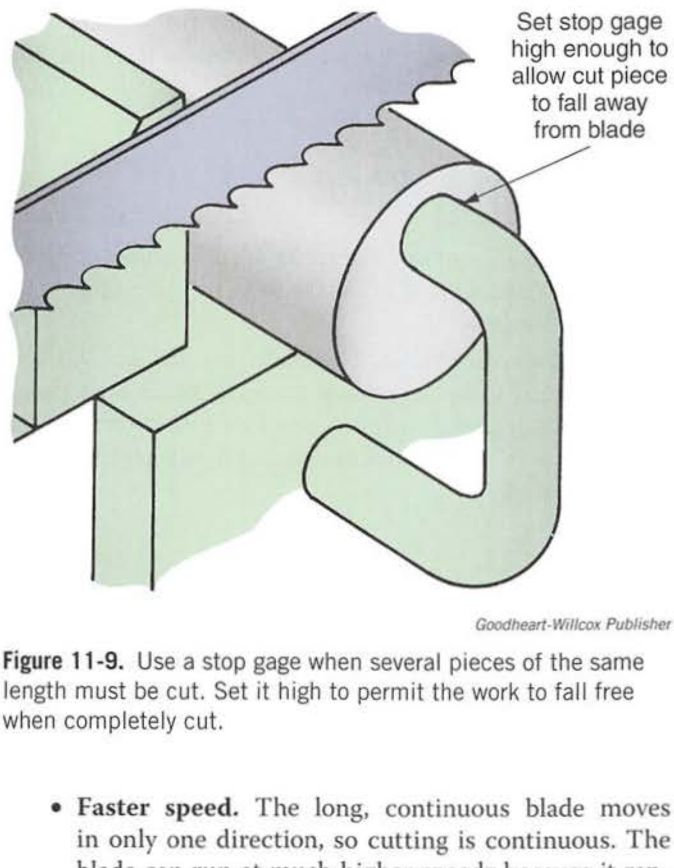


Goodheart-Willcox Publisher

Figure 11-8. 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.

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

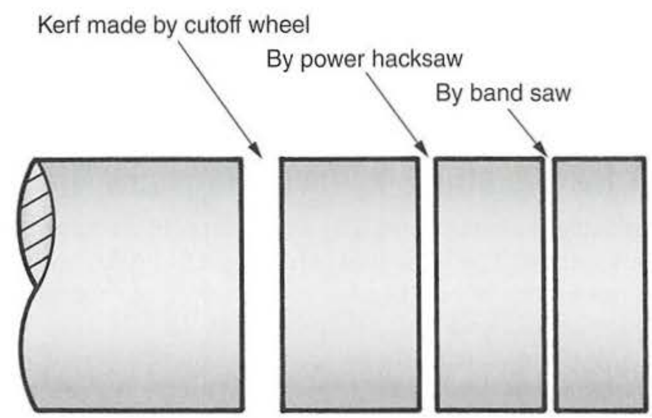
Figure 11-10. Horizontal band saw with built-in coolant system, adjustable hydraulic down feed, automatic shutoff, quick positioning vise, and –45° to +60° swivel mount.



Goodheart-Willcox Publisher

Figure 11-9. Use a stop gage when several pieces of the same length must be cut. Set it high to permit the work to fall free when completely cut.

- **Faster speed.** The long, continuous blade moves in only one direction, so cutting is 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 blade of the reciprocating power hacksaw, **Figure 11-11**.



Goodheart-Willcox Publisher

Figure 11-11. Differences in the amount of metal converted to chips (waste) by each cutoff machine.

11.3.1 Selecting a Band Saw Blade

Band saw blades are made with straight teeth, raker teeth, or wavy teeth, **Figure 11-12**. Most manufacturers also make variations of these sets. The raker set provides greater cutting efficiency and a smoother surface finish. It is preferred for general use. Straight tooth sets also provide for smoother cutting finishes, but they do not create as much noise as the raker design. Wavy sets work best for thinner stock because they create less vibration and noise and tend to produce fewer burrs than the other sets.

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,

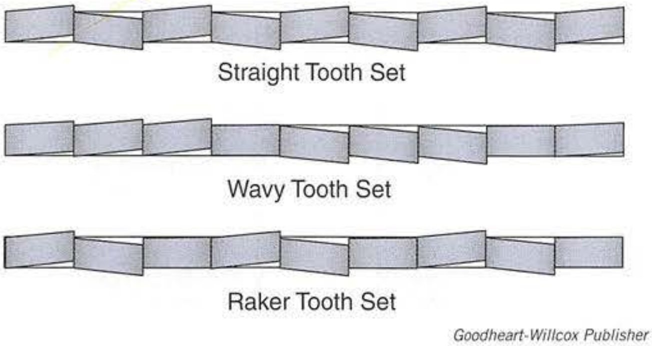


Figure 11-12. Saw blades commonly have straight, raker, or wavy teeth. Raker teeth are preferred for general use, cutting large solid sections, and cutting thick plate.

copper, and soft brasses. The hook tooth blade pattern is recommended for most nonferrous metallic materials. See **Figure 11-13**. 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.

A new development in saw blades is the stepped-back band saw blade. The stepped-back band saw blade has a patterned edge on the back of the blade opposite the cutting teeth. This design works with opposing guides that are positioned on either side of the workpiece. The guides provide downward pressure onto the back edge of the blade, causing the blade to rock in an up-and-down motion as the teeth are being fed through the workpiece. The rocking motion allows the cutting speed of the saw to be increased, **Figure 11-14**. The rocking motion also prevents the cutting teeth from constantly riding on surfaces that are parallel to the cut, resulting in longer blade life. This is particularly advantageous when cutting workpieces that have large cross-sectional areas.

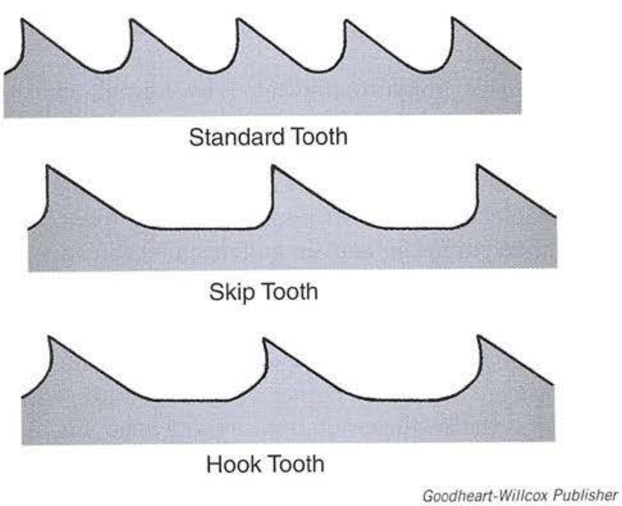


Figure 11-13. Standard tooth blades, with rounded gullets, are usually best for ferrous metals, hard bronzes, and hard brasses. Skip tooth blades provide for more chip clearance without weakening the blade body. They are recommended for cutting aluminum, magnesium, copper, and soft brasses. Hook tooth blades offer two advantages over skip tooth blades: easier feeding and less "gumming up."

11.3.2 Installing a Band Saw Blade

SAFETY NOTE

Wear heavy leather gloves to protect your hands when installing a band saw blade.

Careful blade installation allows the saw to work at top efficiency. Adjust the blade guides to provide adequate support, **Figure 11-15**. Proper blade support is required to cut true and square with the holding device.

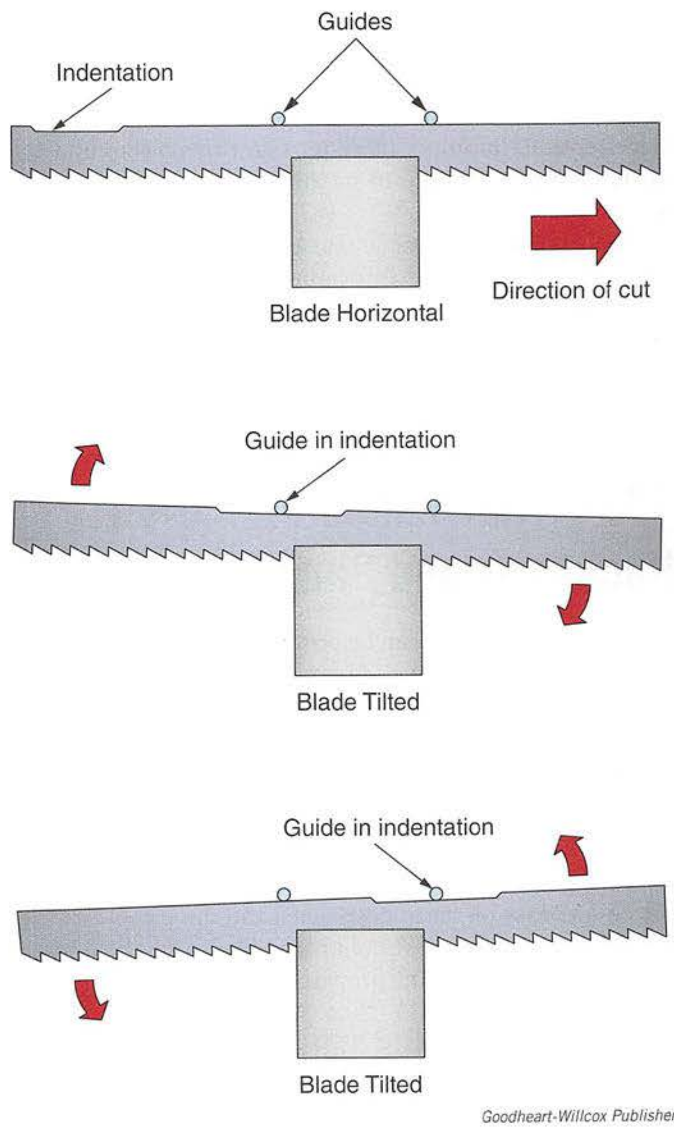


Figure 11-14. A stepped-back band saw blade fits with opposing guides to achieve an up-and-down motion that allows saw speed to increase.

AMT—The Association for Manufacturing Technology

Figure 11-15. Adjust blade guides to provide adequate blade support. Otherwise, the blade will not cut true.

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 hacksaw. Most problems are caused by poor machine condition. Problems can be kept to a minimum by following a maintenance program on a regular basis. This typically includes checking wheel alignment, guide alignment, feed pressure, and hydraulic systems.

11.4 Troubleshooting Power Hacksaws and Band Saws

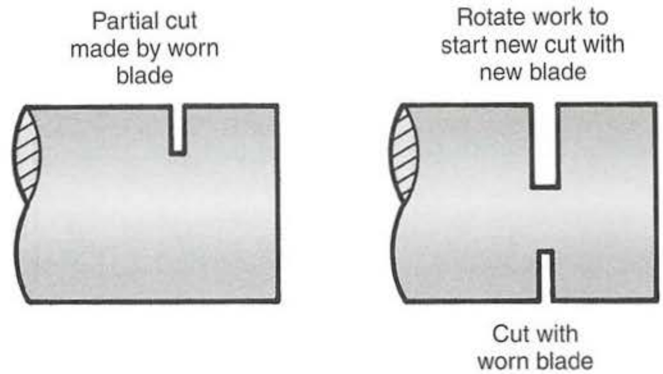
Most sawing problems can be prevented by careful planning and observing a few rules. These rules apply to both power hacksaws and band saws.

11.4.1 Blades Breaking

Blades usually break when they are dropped on the work. A loose blade or excessive feed can also cause the blade to fracture. Loose work can cause blade damage, as can making a cut on a corner or sharp edge when the three-tooth rule is not observed. Most causes of broken blades can be avoided by setting up the machine properly.

11.4.2 Crooked Cutting

Crooked cutting 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 11-16.** A loose blade or a blade rubbing on a clamping fixture can also cause crooked



Goodheart-Willcox Publisher

Figure 11-16. 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.

cutting. It can also be caused by excessive blade pressure on the work or by worn saw guides.

11.4.3 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 breaks out.

11.4.4 Premature Blade Tooth Wear

When blade teeth wear prematurely, the teeth become rounded and dull quickly. Insufficient feed pressure (indicated by light, powdery chips) is one of the major causes of premature blade tooth wear. Insufficient pressure can be corrected by increasing cutting pressure until a full curled chip is produced.

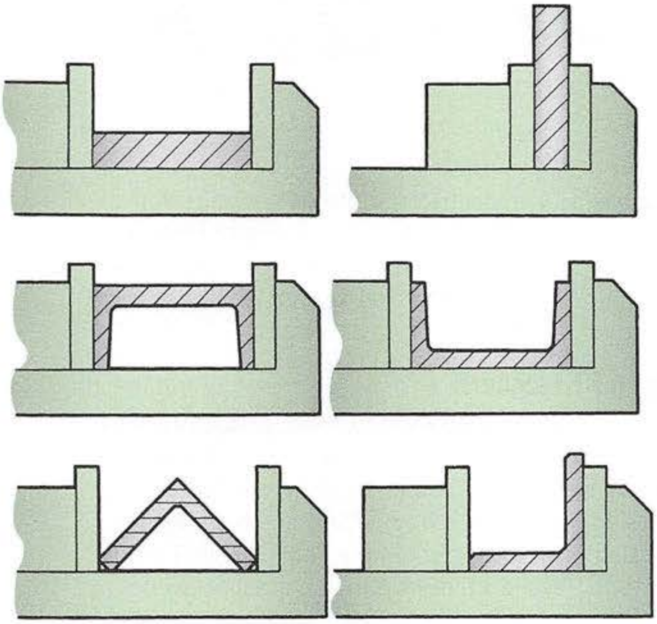
Excessive pressure (indicated by burned chips) also causes premature wear. If too much pressure is the culprit, reduce feed pressure until a full curled chip is formed.

Lack of coolant or a poorly adjusted machine can also cause rapid wear. Correct by following the manufacturer's recommendations for coolant and machine adjustments.

11.4.5 Teeth Strip Off

A common cause of teeth stripping or snapping off a blade is starting a cut on a sharp corner. A machine setup with a flat starting surface will greatly reduce tooth stripping. Also, be sure the work is clamped securely. Loose work can also cause the teeth to strip, **Figure 11-17.**

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 (fewer than three teeth cutting) also cause the teeth to snap off. Make sure the blade is properly mounted and is cutting on the power stroke.



Goodheart-Willcox Publisher

Figure 11-17. Recommended ways to hold sharp-cornered work for cutting. A carefully planned setup ensures that at least three teeth will be cutting, greatly extending blade life.

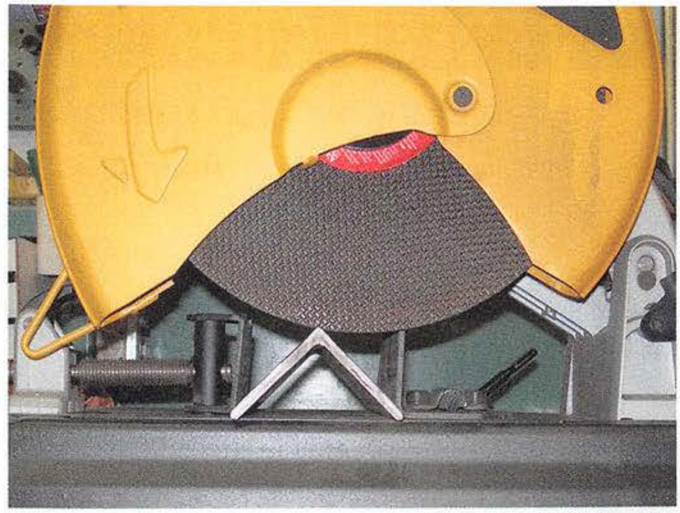


Photo courtesy of Weiler Corporation

Figure 11-18. An industrial abrasive cutoff saw.

11.5 Metal-Cutting Circular Saws

Metal-cutting circular saws are found in many areas of metalworking. Primarily production machines, these saws fall into the following three classifications:

- Abrasive cutoff saw.
- Cold circular saw.
- Friction saw.

An ***abrasive cutoff saw,*** **Figure 11-18,** 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. Heat-resistant abrasive wheels are available for high-speed cutoff of hot stock.

The two general types of abrasive cutting are dry and wet cutting. 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 11-19,** has 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.

W.J. Savage Co.

Figure 11-19. Cold circular saw. This machine can be fitted with carbide-tipped blades or abrasive discs.

CAREER CONNECTION

High School CTE Teacher

What does a high school CTE teacher do?

Career and technical education (CTE) teachers prepare high school students for future careers in technical fields. These teachers may specialize in particular fields or introduce students to the range of technical careers available. Some classrooms and labs prepare students with hands-on experiences that reflect working environments. There are many types of CTE teachers with many different specialties, including culinary instructors, technical and trades teachers, and engineering professors.

What education and skills are needed to be a high school CTE teacher?

CTE teachers almost always require field experience for their particular subject. Culinary professors have been chefs, and welding instructors have been welders. In addition to being outstanding professionals, future CTE instructors must also have teaching skills, such as patience, public speaking ability, and the ability to explain in concise terms what they did as a professional to students who will become professionals.

Before making a change to teaching, interested professionals typically work in the field for 3 to 5 years. They often have bachelor's degrees in their field or in education. Teachers that work in public schools require state-specific certification or licensure.

What is it like to be a high school CTE teacher?

High school CTE teachers teach during standard school hours and do much the same physical labor as other teachers. They may specialize in one subject or in several, and they may also run specialized programs to best teach career readiness. According to the *Occupational Outlook Handbook*, the vast majority of CTE teachers are employed by secondary schools, and the median annual wage for this profession is $52,800 per year.

A ***friction saw*** operates at very high speeds (20,000 surface feet per minute or 6000 meters per minute) and actually melts its way through the metal. The friction saw blade may or may not have teeth. The teeth are used primarily to carry oxygen to the cutting area. These machines have many applications in steel mills. For example, they can be used to cut red-hot billets (sections of semifinished steel).

11.6 Power Saw Safety

Keep the following guidelines in mind while working with any type of power saw:

- Never attempt to operate a sawing machine while your senses are impaired by medication or other substances.
- Get help when you are lifting and cutting heavy material.
- Clean oil, grease, and coolant from the floor around the work area.
- Use special care when handling pieces with burrs. Burrs on cut pieces are sharp.
- 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 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.
- 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.

Chapter Review

Summary

- Reciprocating power hacksaws, power band saws, and circular metal-cutting saws are the principal types of saws used in machining operations.
- Power hacksaws can have a positive feed or a definite pressure feed.
- Power band saws offer greater precision, faster speeds, and less waste than other metal-cutting saws.
- Proper blade tension is important for both power hacksaws and band saws.
- Follow the three-tooth rule for the best cutting action.
- Select the correct feed pressure, tooth set, and blade pattern to ensure efficient and safe cutting.
- Most sawing problems can be prevented by careful planning and observing a few rules.
- Abrasive cutoff saws, cold circular saws, and friction saws are the three classifications of metal-cutting circular saws.
- Always follow proper safety procedures when operating a power saw.

Review Questions

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

1. List the three basic types of metal-cutting saws.
2. The ______ saw has a back-and-forth cutting action.
3. The power hacksaw only cuts on the ______ stroke.
4. When using a power sawing machine, which materials do *not* require coolant?
5. What is the "three-tooth rule" for sawing?
6. List the two principal types of hacksaw blades.

For questions 7–9, give the "rule of thumb" for selecting the correct blade.

7. Select ______ teeth per inch for cutting large sections or readily machined materials.
8. Select ______ teeth per inch for cutting harder alloys and miscellaneous cutting.
9. Select ______ teeth per inch for cutting on the majority of light-duty machines, where work is limited to small sections requiring moderate to light feed pressure.
10. List three methods used to put proper tension on a power hacksaw blade.
11. When is a stop gage used?
12. What three advantages does the band saw offer over other types of power saws?
13. Band saw blades are made with three types of teeth. List them.

For questions 14–16, give the tooth pattern best suited for cutting the given materials.

14. The ______-tooth pattern is best suited for cutting most ferrous metals.
15. The ______-tooth pattern is preferred for cutting aluminum, magnesium, copper, and soft brass.
16. The ______-tooth pattern is recommended for most nonferrous metallic materials.
17. What is the most common cause of crooked cutting?
18. Name four possible causes of premature wear of saw blade teeth.
19. List the three types of circular metal-cutting saws.
20. List five safety precautions to be observed when operating a power saw.

CHAPTER 12
Drills and Drilling Machines

Chapter Outline

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.
- Explain the safety rules that pertain to drilling operations.
- Identify and describe common drills and drill-holding devices.
- Describe common work-holding devices.
- Use appropriate cutting speeds and feeds for drilling procedures.
- Identify common cutting fluids.
- Sharpen a twist drill.
- Describe basic drilling operations, including countersinking, counterboring, spotfacing, tapping, reaming, and microdrilling.

Technical Terms

blind hole
center finder
chuck
counterboring
countersinking
cutting speed
drift
drill gage
drilling machine
drill point gage
feed
flutes
pilot hole
reaming
spotfacing
tapping
twist drill

Drills and drilling machines are available in an array of types, sizes, and materials for various applications. These versatile cutting tools are used primarily for creating round holes, though they can also perform specialized machining operations.

12.1 Drilling Machines

A ***drilling machine*** is a power-driven machine that holds both the material and cutting tool, bringing them together to create a round hole in the material. Many different types of drilling machines are used in industry. The type of machine used depends on the operation being performed, the size of the workpiece, and the variety of operations required of the machine.

12.1.1 Types of Drilling Machines

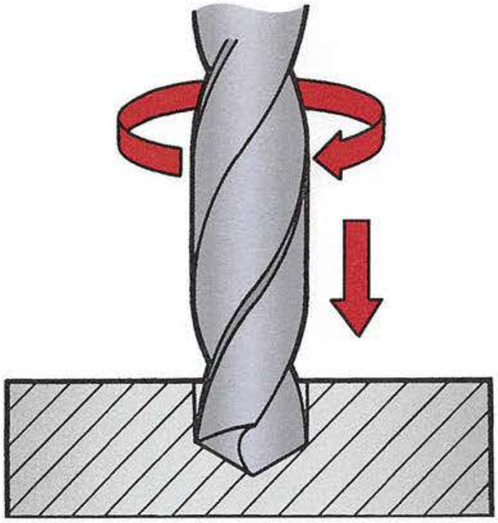
The most common drilling machine is the drill press, **Figure 12-1**. A drill press operates by rotating a cutting tool, or drill, against the material with sufficient pressure to cause the drill to penetrate the material. See **Figure 12-2**.

The size of a drill press is determined by the largest diameter circular piece that can be drilled on center, **Figure 12-3**. 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.

Bench drill presses can be used to drill holes in small workpieces, **Figure 12-4**. 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 12-5**.

A radial drill press is designed to handle large drilling work. The drill head moves back and forth on an arm that extends from the massive machine column. The arm can be



Goodheart-Willcox Publisher

Figure 12-2. Drilling is the operation most often performed on a drill press. Both rotating force and a downward pushing force are needed for drilling.

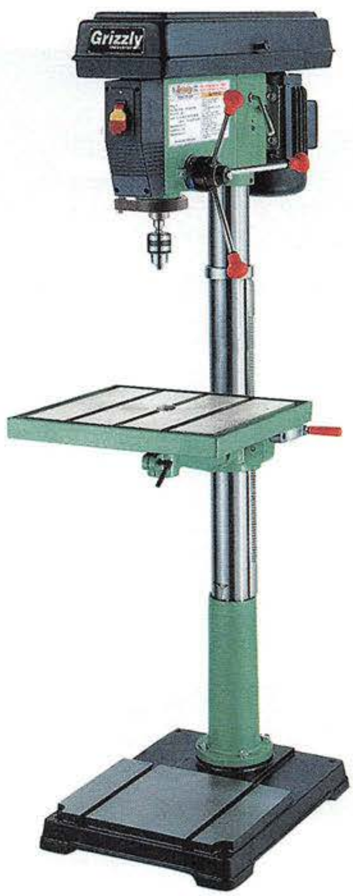
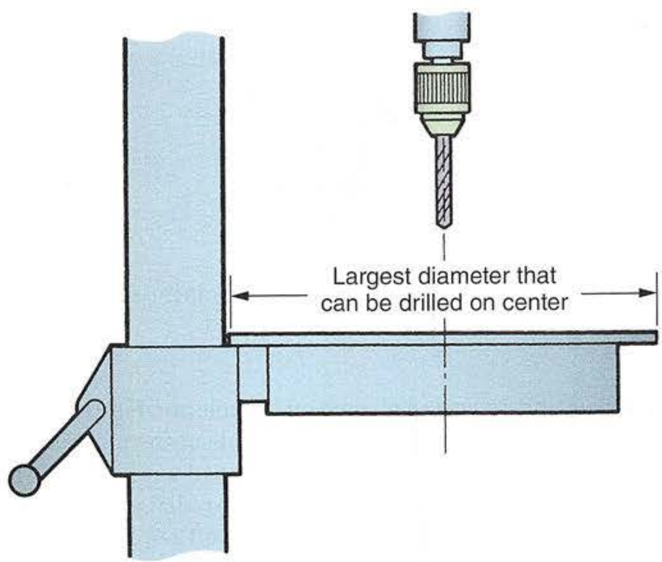


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 12-1. A 12-speed 20″ drill press.



Goodheart-Willcox Publisher

Figure 12-3. Drill press size.

Goodheart-Willcox Publisher

Figure 12-4. A small bench drill press can be very useful.

Makita USA, Inc.

Figure 12-5. Portable electric drills are manufactured in a wide range of sizes. This model is battery powered.

moved up and down and pivoted on the column, **Figure 12-6**. 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. Smaller bench radial drill presses are used to drill smaller holes. See **Figure 12-7**. These units are not as expensive as a full-size radial drill press.

Sharp Industries, Inc.

Figure 12-6. This radial drill press can drill large-diameter holes in large workpieces.

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

Figure 12-7. Bench radial drill press.

Portable magnetic drills can be used in the shop and on the jobsite. These machines can be positioned in an upright, horizontal, or vertical position when drilling. See **Figure 12-8**.

Gang drilling machines consist of several drill assemblies, **Figure 12-9**. 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. This allows several operations to be performed without changing drills.

Machining centers operate under computer numerical control (CNC). The center can be programmed to drill holes efficiently and accurately as part of a machining sequence. The drill position and speed are programmed into the center. No drill jigs are required. See **Figure 12-10**.

Robotic drilling machines are basically programmable, mobile drilling machines. The machine is programmed to move along one workpiece or among several workpieces, drilling at specified locations.

Clausing Industrial, Inc.

Figure 12-9. 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.

Hougen Manufacturing, Inc.

Figure 12-8. 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 used.

Haas Automation, Inc.

Figure 12-10. This CNC machine has a tool magazine capable of holding twenty tools.

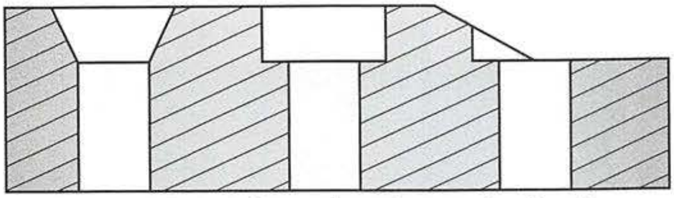


Figure 12-11. Sectional view of three drilled and machined holes. Countersinking cuts a chamfer in a hole. Counterboring prepares a hole to receive a fillister- or socket-head screw. Spotfacing machines a surface to permit a nut or bolt head to bear uniformly.

12.1.2 Uses of Drilling Machines

Drilling machines are primarily used to cut round holes. They can also be used for many different machining operations, **Figure 12-11**, including the following:

- ***Countersinking.*** Enlarging a hole at the workpiece surface along an angle to allow a screw head to be flush with the surface.
- ***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.
- ***Spotfacing.*** An operation performed to put a smooth, flat bearing surface surrounding a hole so that a nut or bolt head bears uniformly on the work.
- ***Tapping.*** This operation cuts screw threads into an existing hole.
- ***Reaming.*** An operation performed on an existing hole to enlarge and finish it to very accurate dimensions.

12.2 Drill Press Safety

Follow these guidelines to work safely with a drill press:

- Wear goggles when working on a drill press.
- Remove any jewelry and tuck in loose clothing so it does not become entangled in the rotating drill.
- 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.

GREEN MACHINING

Recycling Tungsten Carbide Drills

Machine shops collect and store worn-out tungsten carbide drills for resale. Industry recyclers pay high rates for scrap tungsten carbide and recycle it into raw material ready for new uses. Tungsten carbide is one of the most sought-after ceramic scraps because of its strength and suitability for reuse. Reselling these old tungsten carbide drills benefits shops economically while also saving resources.

12.3 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 12-12**. Then they are heat-treated and ground to exact size.

Most drills are made of high-speed steel (HSS) or carbon steel and are available with straight or taper shanks and with tungsten carbide tips. High-speed steel drills can be operated at much higher cutting speeds than carbon steel drills without danger of burning and drill damage. Coating drills with titanium nitride greatly increases tool life.

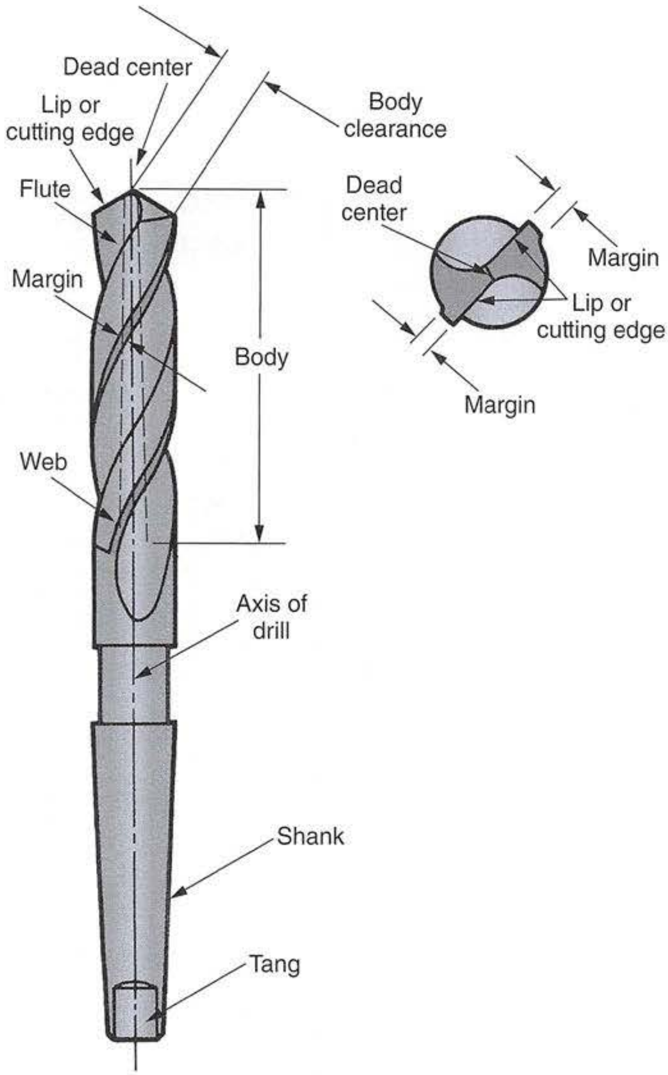
12.3.1 Parts of a Drill

The twist drill is an efficient cutting tool. It is composed of three principal parts: point, shank, and body, **Figure 12-13**.

Point

The point is the cone-shaped end that does the cutting. The point consists of the following components:

Figure 12-12. A common twist drill.



Goodheart-Willcox Publisher

Figure 12-13. Parts of a twist drill.

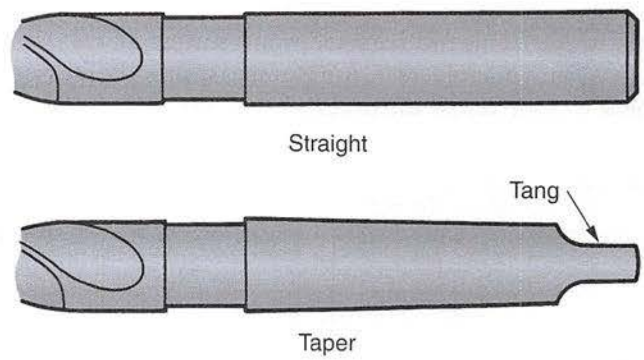
- The dead center is the sharp edge at the extreme tip of the drill. This should always be in the exact center of the drill axis.
- The lips are the cutting edges of the drill.
- The heel is the portion of the point back from the lips.

Lip clearance is the amount by which the surface of the point is relieved back from the lips.

Shank

The shank is the portion of the drill that mounts into the chuck or spindle. The shank can be either straight or tapered, **Figure 12-14**. 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 located on the end of 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.



Goodheart-Willcox Publisher

Figure 12-14. Types of twist drill shanks.

Body

The body is the portion of the drill between the point and the shank. It consists of the following components:

- 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.

12.3.2 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 size ranges listed include the most common drill sizes within each series. Larger and smaller size drills exist but are uncommon or specialized. **Figure 12-15** provides decimal equivalents of common drill sizes.

12.3.3 Drill Measurements

Always check the drill diameter before using it. Using the wrong size drill can be a very expensive and time-consuming mistake. Most drills, with the exception of small drills in the number series, have their 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 12-16** for both techniques.

Diameter can also be checked with a ***drill gage***, **Figure 12-17**. Drill gages are made for various drill series.

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

(Continued)

Goodheart-Willcox Publisher

Figure 12-15. Decimal equivalents of drill sizes.

Inch	mm	Letter Sizes	Decimals of an Inch
	5.9		.2323
		A	.2340
15/64			.2344
	6		.2362
		B	.2380
	6.1		.2402
		C	.2420
	6.2		.2441
		D	.2460
	6.25		.2461
	6.3		.2480
1/4		E	.2500
	6.4		.2520
	6.5		.2559
		F	.2570
	6.6		.2598
		G	.2610
	6.7		.2638
17/64			.2656
	6.75		.2657
		H	.2660
	6.8		.2677
	6.9		.2717
		I	.2720
	7		.2756
		J	.2770
	7.1		.2795
		K	.2810
9/32			.2812
	7.2		.2835
	7.25		.2854
	7.3		.2874
		L	.2900
	7.4		.2913
		M	.2950
	7.5		.2953
19/64			.2969
	7.6		.2992
		N	.3020
	7.7		.3031
	7.75		.3051
	7.8		.3071
	7.9		.3110
5/16			.3125
	8		.3150
		O	.3160
	8.1		.3189
	8.2		.3228
		P	.3230
	8.25		.3248
	8.3		.3268

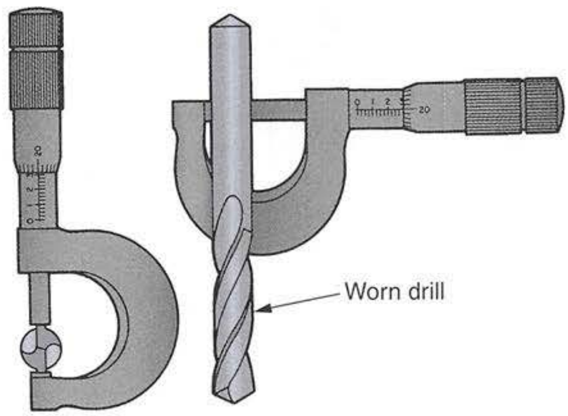
Inch	mm	Letter Sizes	Decimals of an Inch
21/64			.3281
	8.4		.3307
		Q	.3320
	8.5		.3346
	8.6		.3386
		R	.3390
	8.7		.3425
11/32			.3438
	8.75		.3345
	8.8		.3465
		S	.3480
	8.9		.3504
	9		.3543
		T	.3580
	9.1		.3583
23/64			.3594
	9.2		.3622
	9.25		.3642
	9.3		.3661
		U	.3680
	9.4		.3701
	9.5		.3740
3/8			.3750
		V	.3770
	9.6		.3780
	9.7		.3819
	9.75		.3839
	9.8		.3858
		W	.3860
	9.9		.3898
25/64			.3906
	10		.3937
		X	.3970
		Y	.4040
13/32			.4063
		Z	.4130
	10.5		.4134
27/64			.4219
	11		.4331
7/16			.4375
	11.5		.4528
29/64			.4531
15/32			.4688
	12		.4724
31/64			.4844
	12.5		.4921
1/2			.5000
	13		.5118
33/64			.5156
17/32			.5313
	13.5		.5315

Inch	mm	Decimals of an Inch
35/64		.5469
	14	.5512
9/16		.5625
	14.5	.5709
37/64		.5781
	15	.5906
19/32		.5938
39/64		.6094
	15.5	.6102
5/8		.6250
	16	.6299
41/64		.6406
	16.5	.6496
21/32		.6563
	17	.6693
43/64		.6719
11/16		.6875
	17.5	.6890
45/64		.7031
	18	.7087
23/32		.7188
	18.5	.7283
47/64		.7344
	19	.7480
3/4		.7500
49/64		.7656
	19.5	.7677
35/32		.7812
	20	.7874
51/64		.7969
	20.5	.8071
13/16		.8125
	21	.8268
53/64		.8281
27/32		.8438
	21.5	.8465
55/64		.8594
	22	.8661
7/8		.8750
	22.5	.8858
57/64		.8906
	23	.9055
29/32		.9063
59/64		.9219
	23.5	.9252
15/16		.9375
	24	.9449
61/64		.9531
	24.5	.9646
31/32		.9688
	25	.9843
63/64		.9844

Inch	mm	Decimals of an Inch
1		1.0000
	25.5	1.0039
1 1/64		1.0156
	26	1.0236
1 1/32		1.0313
	26.5	1.0433
1 3/64		1.0469
1 1/16		1.0625
	27	1.0630
1 5/64		1.0781
	27.5	1.0827
1 3/32		1.0938
	28	1.1024
1 7/64		1.1094
	28.5	1.1220
1 1/8		1.1250
1 9/64		1.1406
	29	1.1417
1 5/32		1.1562
	29.5	1.1614
1 11/64		1.1719
	30	1.1811
1 3/16		1.1875
	30.5	1.2008
1 13/64		1.2031
1 7/32		1.2188
	31	1.2205
1 15/64		1.2344
	31.5	1.2402
1 1/4		1.2500
	32	1.2598
1 17/64		1.2656
	32.5	1.2795
1 9/32		1.2813
1 19/64		1.2969
	33	1.2992
1 5/16		1.3125
	33.5	1.3189
1 21/64		1.3281
	34	1.3386
1 11/32		1.3438
	34.5	1.3583
1 23/64		1.3594
1 3/8		1.3750
	35	1.3780
1 25/64		1.3906
	35.5	1.3976
1 13/32		1.4063
	36	1.4173
1 27/64		1.4219
	36.5	1.4370

Goodheart-Willcox Publisher

Figure 12-15. Decimal equivalents of drill sizes (continued).



Goodheart-Willcox Publisher

Figure 12-16. Measuring drill size with a micrometer.

However, 1/2″ drills are the largest that can be checked. New drills are checked at the points. Worn drills are checked at the end of the flutes.

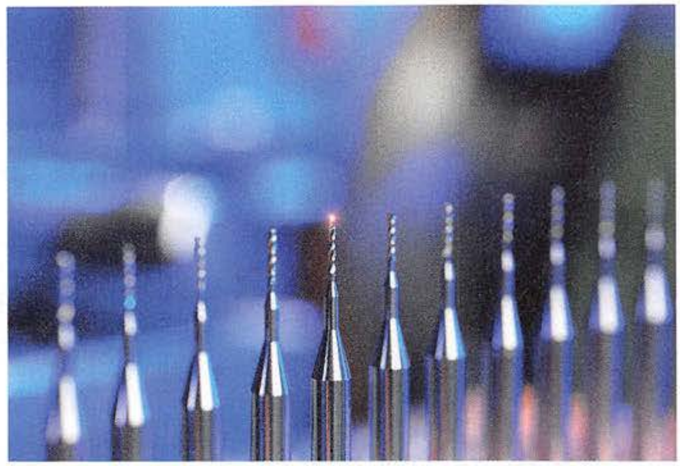
12.3.4 Types of Drills

Industry uses various types of drills to improve the accuracy of the drilled hole, to speed production, and to improve drilling efficiency. Some drills are designed for very specific operations or for drilling in specific materials.

A reduced-shank drill has a shank with a diameter that is smaller than the body. Reduced-shank drills allow larger holes to be drilled using machine tools with smaller chuck or spindle sizes.

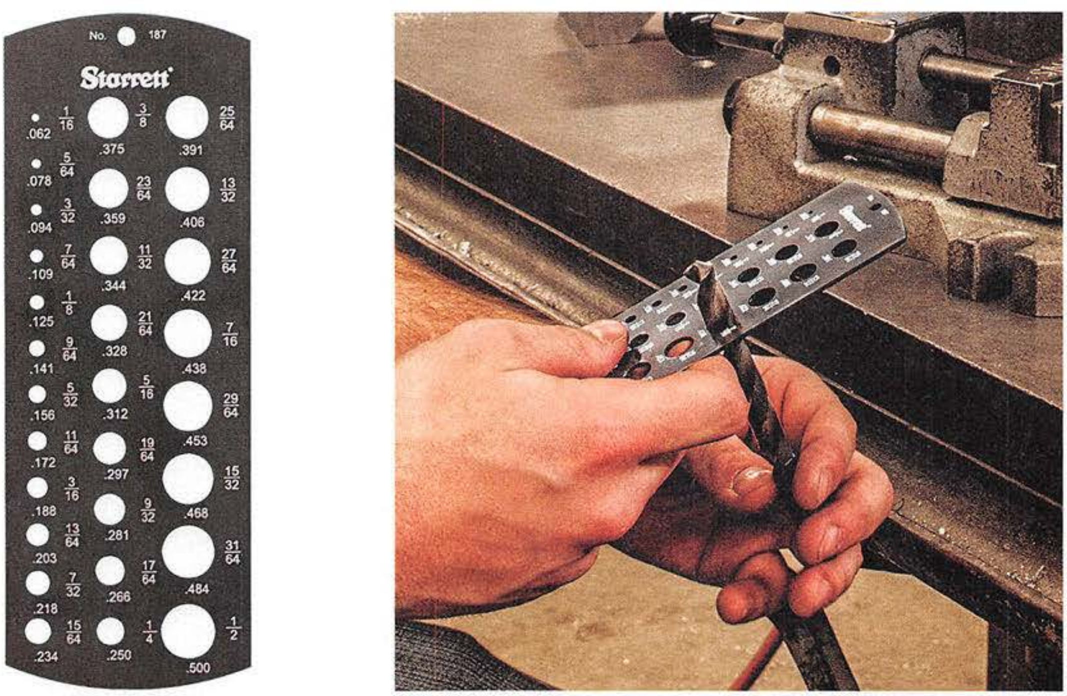
Microdrills, **Figure 12-18**, have diameters smaller than 0.020″ (0.508 mm). They require special drilling equipment and can drill holes as small as 0.0012″ (0.03 mm).

A coolant-hole drill has holes through the body, which permit cutting fluid or air to remove heat from the point,



BIG Kaiser Precision Tooling Inc.

Figure 12-18. Microdrills can be made of either HSS or tungsten carbide. Microdrills can drill holes as small as 0.0012″ (0.03 mm) through materials ranging from steel to cast iron to nonferrous alloys.



L. S. Starrett Co.

Figure 12-17. This drill gage is used to measure fractional size drills from 1/16″ to 1/2″. Similar gages are available for measuring letter, number, and metric size drills.

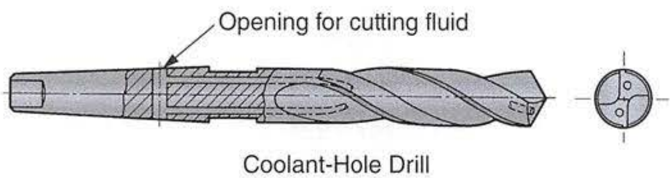
Figure 12-19. The pressure of the fluid or air also ejects the chips from the hole while drilling.

Step drills eliminate drilling operations in production work by cutting two different diameters in the same hole. They are used to create holes with counterbores in a single operation. **Figure 12-20** illustrates this type of drill. A combination drill and reamer also speeds up production by performing two operations at once, **Figure 12-21**.

Core drills are three- or four-flute drills used to enlarge core holes in a casting. See **Figure 12-22**. Drills with three or four flutes provide a better finish than drills with only two flutes.

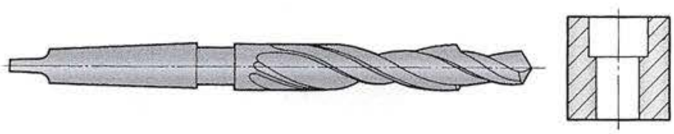
Half-round straight-flute drills, **Figure 12-23**, are designed to produce holes in brass, copper alloys, and other soft, nonferrous materials. Heavy-duty carbide-tipped versions are available for drilling hardened steels. These drills are manufactured in fractional and number sizes.

The straight-flute gun drill is designed for ferrous and nonferrous metals, **Figure 12-24**. It is usually fitted with a carbide cutting tip.



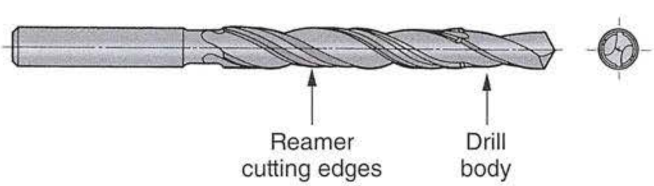
Goodheart-Willcox Publisher

Figure 12-19. A taper shank twist drill with holes to direct coolant to the cutting edges.



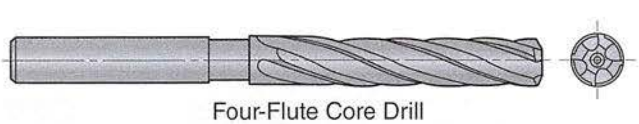
Goodheart-Willcox Publisher

Figure 12-20. A step drill designed to drill a hole and counterbore the hole all in the same operation.



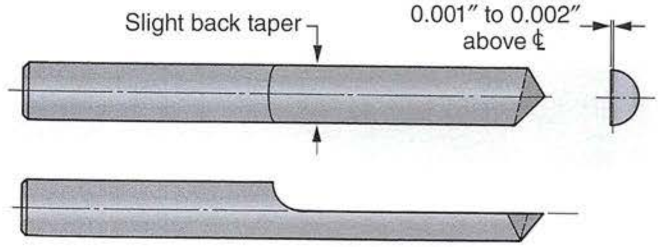
Goodheart-Willcox Publisher

Figure 12-21. Combination drill and reamer.



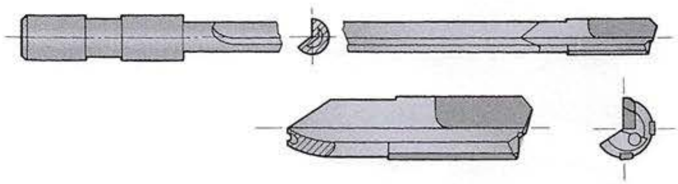
Goodheart-Willcox Publisher

Figure 12-22. This four-flute core drill is used to enlarge cored and drilled holes.



Goodheart-Willcox Publisher

Figure 12-23. A half-round drill.



Goodheart-Willcox Publisher

Figure 12-24. A straight-flute gun drill. The tip is shown in larger scale. The light-colored portions are tungsten carbide. The larger section does the cutting, and the smaller sections act as wear surfaces.

Indexable-insert drills, **Figure 12-25**, are capable of drilling at much higher speeds than HSS drills. The low-cost carbide inserts with multiple cutting edges eliminate costly sharpening. When the cutting edges are worn, only the insert is changed. There is no need to replace the entire drill.

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 usually made of tungsten carbide, **Figure 12-26**. 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.

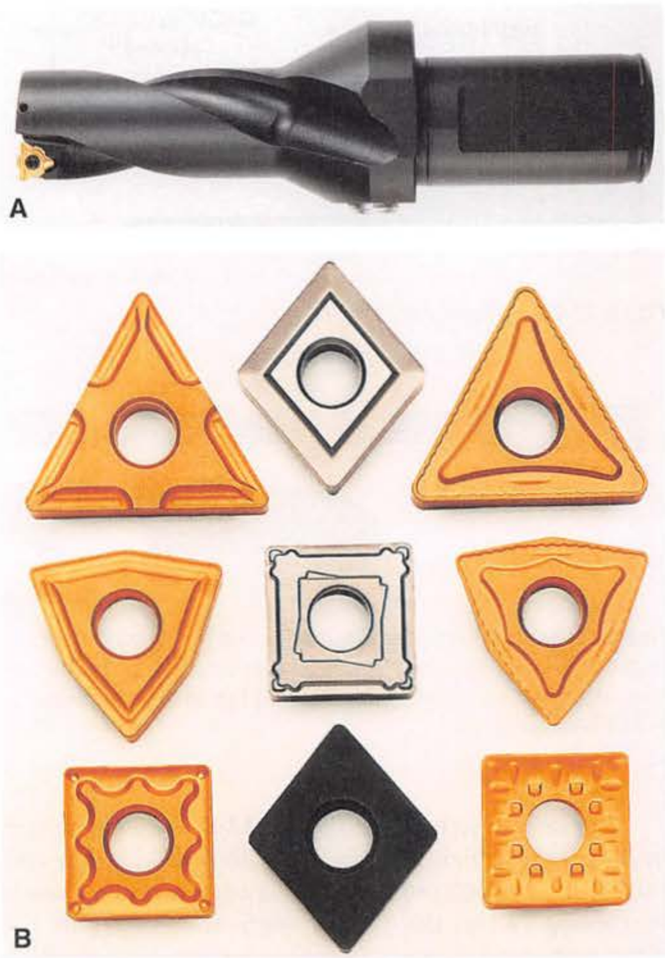
12.4 Drill-Holding Devices

The mechanism for holding a drill in the drill press depends on its shank. Drills with straight shanks are held by a movable jaw mechanism called a ***chuck***. See **Figure 12-27**. Drills with taper shanks are held in place by a tapered spindle, **Figure 12-28**.

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.

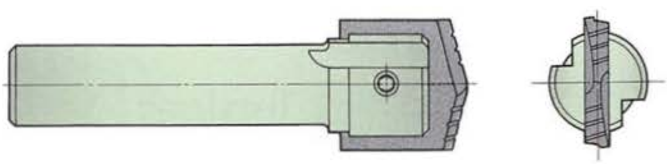
SAFETY NOTE

Always remove the key from the chuck when finished tightening and before turning on the drill press.



Iscar Metals, Inc.; Hertel Cutting Technologies, Inc.

Figure 12-25. 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.



Goodheart-Willcox Publisher

Figure 12-26. This spade drill has a replaceable carbide insert cutting tip. Replaceable inserts provide an economical way to drill large holes.

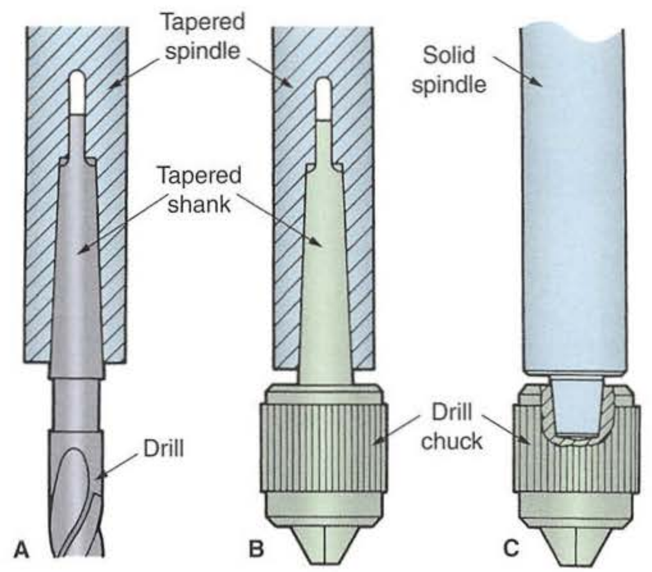
Taper shank drills must be wiped clean before the shank is inserted into the spindle. Nicks in the shank must be removed with an oilstone. Otherwise the shank will not seat properly.

SAFETY NOTE

Never attempt to use a taper shank drill mounted in a drill chuck.

Yukiwa Seiko USA, Inc.

Figure 12-27. A drill chuck can be tightened with a key to hold the drill. Keyless chucks are also available.



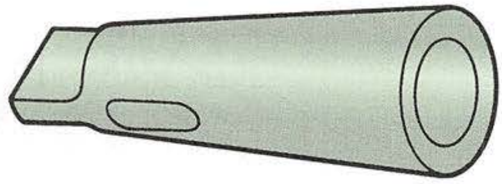
Goodheart-Willcox Publisher

Figure 12-28. Tapered spindles. A—Taper shank drill. B—Drill chuck with a taper shank. C—Solid spindle with a short external taper that fits into a drill chuck. The chuck is permanently attached to the spindle.

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 inserted in a sleeve to fit the spindle taper, **Figure 12-29**. Drills with shanks larger than the spindle opening can be fit by using a socket, **Figure 12-30**. The taper opening in the socket is larger than the taper on its shank. A socket should only be used when a larger drill press is not available.

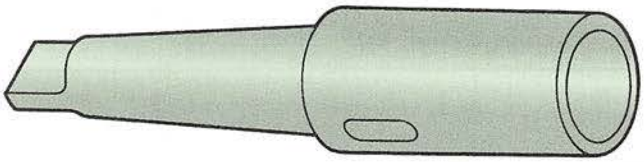
SAFETY NOTE

It is dangerous to overstress a drill press by using a drill larger than the machine's rated capacity.



Goodheart-Willcox Publisher

Figure 12-29. A drill sleeve is needed when the drill is too small for the spindle.



Goodheart-Willcox Publisher

Figure 12-30. A drill socket is used when the spindle is too small for the drill.

Sleeves, sockets, and taper shank drills are separated with a ***drift***, **Figure 12-31**. To use a drift, insert it in the slot with the round edge up, **Figure 12-32**. A sharp rap with a lead hammer will cause the parts to separate.

SAFETY NOTE

Never use a file tang instead of a drift. It will damage both the drill shank and the machine spindle so that other drills will not fit properly. The file could also shatter.

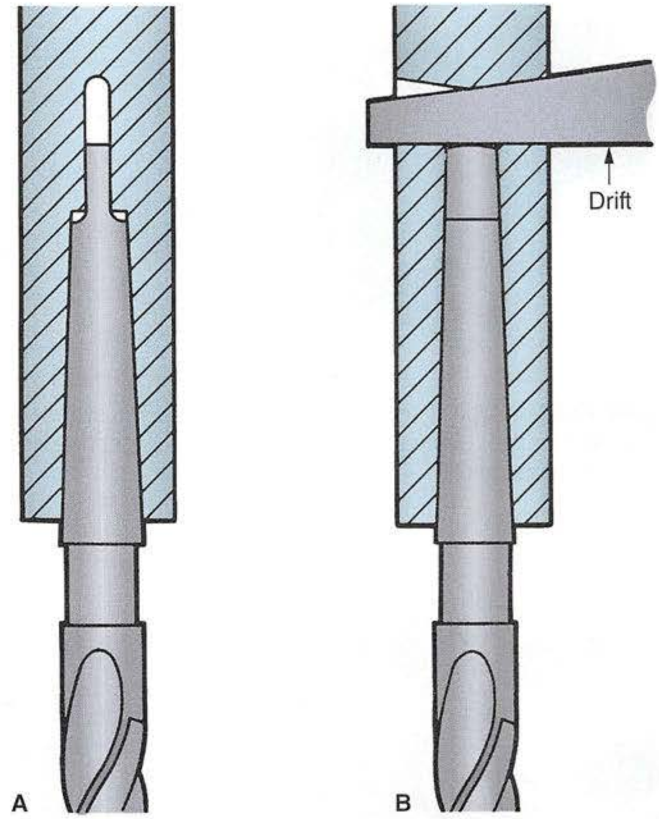
When removing a drill from the spindle, hold the drill to prevent it from falling to the floor. Dropping the drill may damage the drill point. Wrap a piece of clean cloth around the drill to protect your hand from metal chips.

12.5 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.

Goodheart-Willcox Publisher

Figure 12-31. A drift is used to remove taper shank tools from the drill spindle. Never use a file tang.



Goodheart-Willcox Publisher

Figure 12-32. Using a drift. A—The drill is locked in the spindle. B—Using a drift to remove the drill from the spindle.

SAFETY NOTE

Serious injury can result from work that becomes loose and spins on a drill press. This dangerous situation is nicknamed a "merry-go-round."

12.5.1 Vises

Vises are widely used to hold work, **Figure 12-33**. For best results, the vise must be bolted to the drill table.

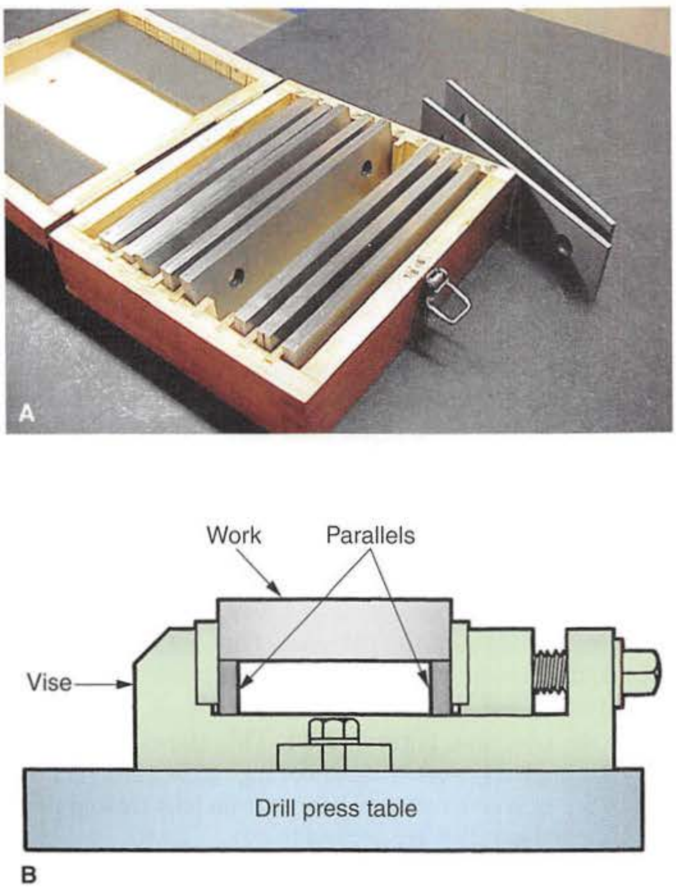
Parallels are often used to level the work and raise it above the vise base, **Figure 12-34**. This allows the drill to come through the work without damaging the vise. Parallels can be made from stock steel bars or from heat-treated steel. Heat-treated parallels are ground to size.

Seat 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 work is not seated properly.

An angular or cross-slide vise permits angular drilling without tilting the drill press table. See **Figure 12-35**. The cross-slide permits rapid alignment of the work. Some cross-slides are fitted with a vise, **Figure 12-36**. Others have a series of tapped holes for mounting a vise or another work-holding device.

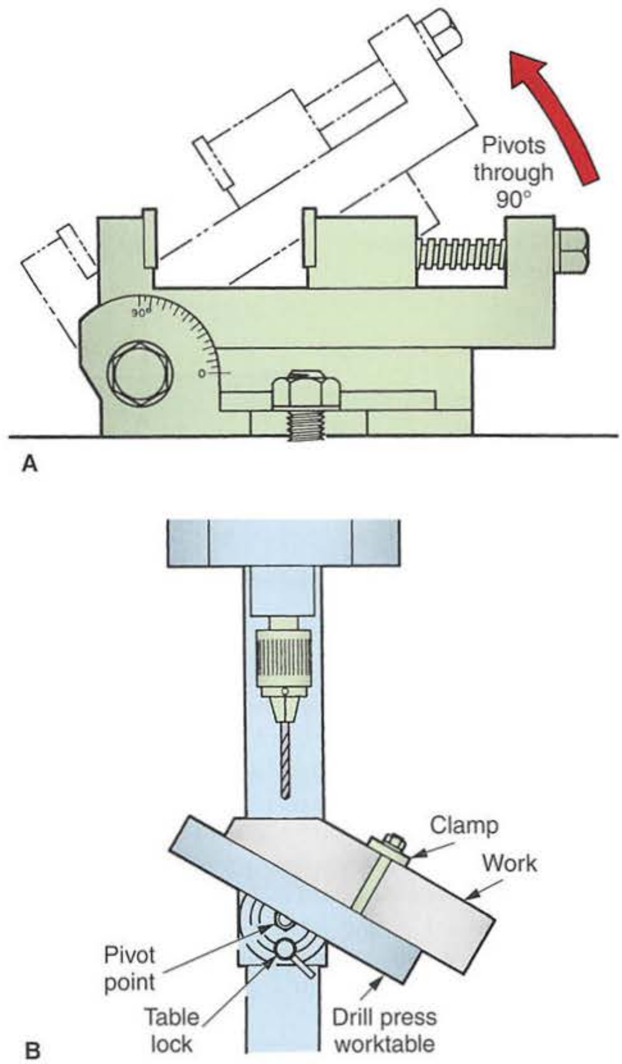
Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 12-33. A typical vise used on a drill press. The swivel base permits the vise body to pivot 180°. The quick-acting vise jaw locks parts quickly.



Goodheart-Willcox Publisher

Figure 12-34. Parallels. A—Steel parallels are available in a variety of sizes. B—Parallels are often used to raise work above the vise base. This prevents the drill from cutting into the vise as it goes through the work.



Goodheart-Willcox Publisher

Figure 12-35. 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.

12.5.2 V-Blocks

V-blocks support round work for drilling, **Figure 12-37.** 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 12-38.**

12.5.3 T-Bolts

T-bolts fit into the drill press table slots and fasten the work or clamping devices to the machine, **Figure 12-39.** For convenience, it is desirable to have an assortment of different

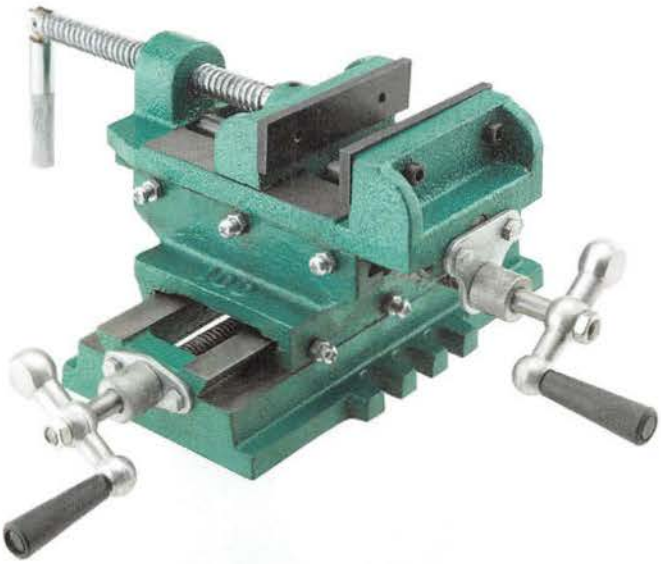
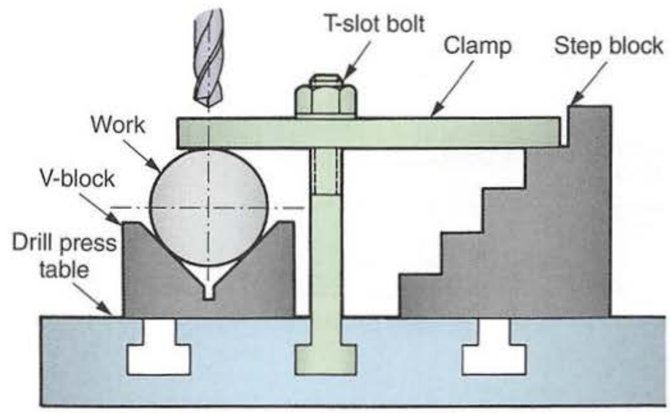


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 12-36. A cross-slide vise permits rapid alignment of work for drilling.

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

Figure 12-37. V-blocks fitted with clamps to hold round workpieces.



Goodheart-Willcox Publisher

Figure 12-38. One method of clamping large-diameter stock for drilling. Always check to be sure that the drill will clear the V-block when it comes through the work.

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

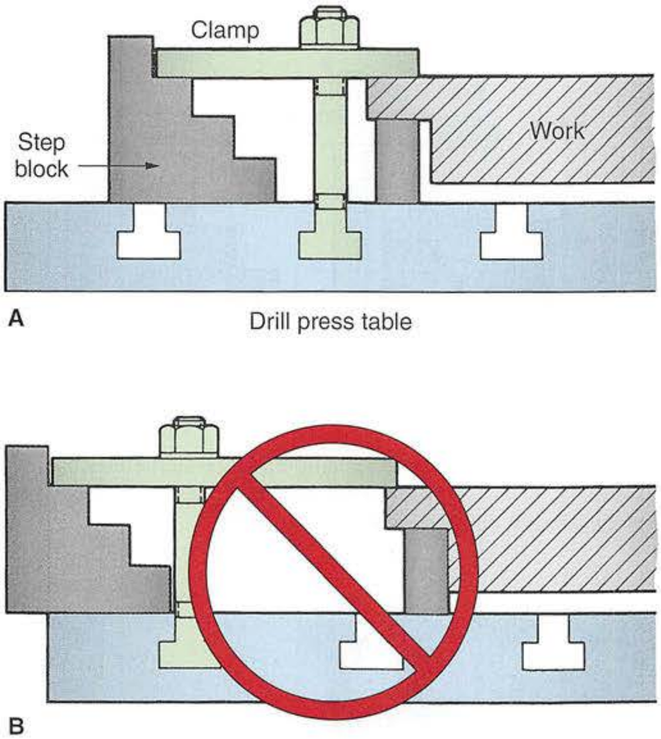
Figure 12-39. A few of the many types and sizes of T-bolts.

length T-bolts. A washer should always be used between the nut and the holding device. To reduce the chance of a setup working loose, place the bolts as close to the work as possible. See **Figure 12-40**.

12.5.4 Strap Clamps, Step Blocks, and Angle Plates

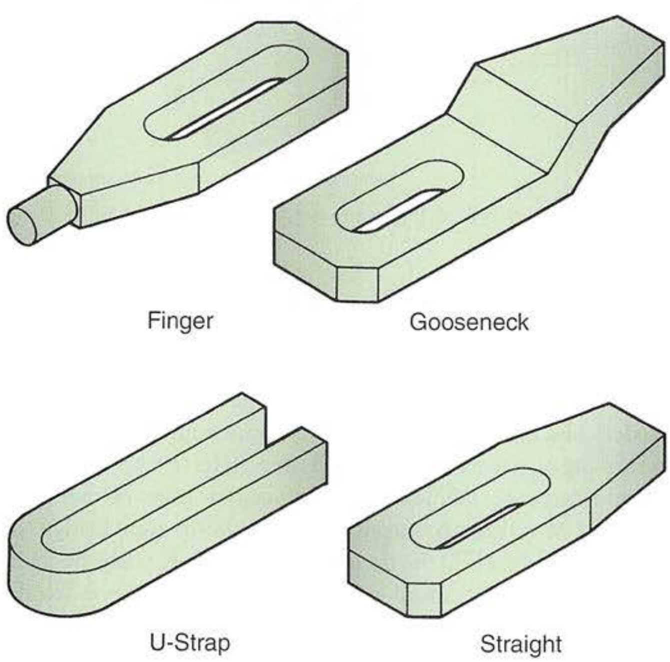
Strap clamps, **Figure 12-41**, make the clamping operation easier. The elongated slot permits some adjustment without removing the washer and nut. Use a strip of copper or aluminum to protect a machined surface that must be clamped.

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



Goodheart-Willcox Publisher

Figure 12-40. Examples of clamping techniques. A—Correct clamping technique. The clamp is parallel to the work. Clamp slippage can be reduced by placing a piece of paper between the work and the clamp. B—Incorrect clamping technique. The T-bolt is too far from the work. This allows the clamp to spring under pressure.



Goodheart-Willcox Publisher

Figure 12-41. Types of strap clamps.

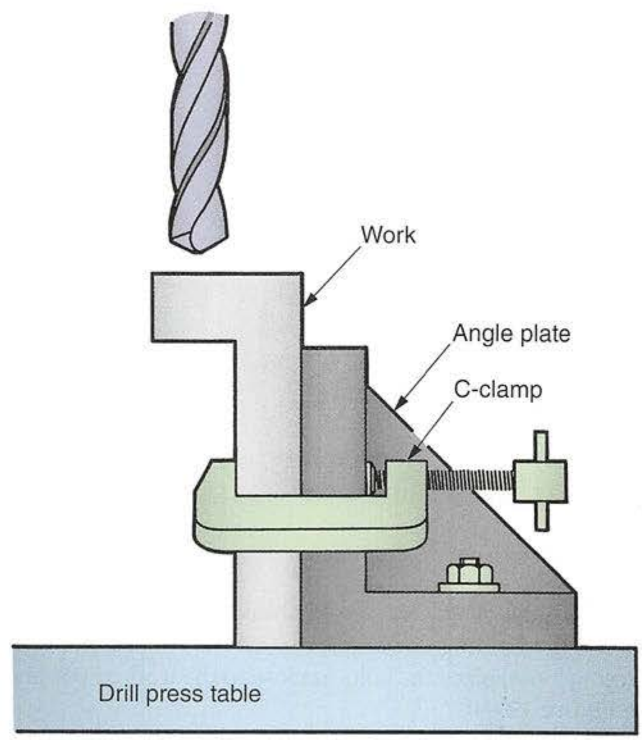
finger clamp permits the use of small holes or openings in the work for clamping.

A step block supports the strap clamp opposite the work, **Figure 12-42**. The steps allow the adjustments necessary to keep the strap parallel with the work.

An angle plate is often used when work must be clamped to a support. The angle plate is then bolted to the machine table, **Figure 12-43**.

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

Figure 12-42. Step blocks are used to support strap clamps.



Goodheart-Willcox Publisher

Figure 12-43. Work must sometimes be mounted against an angle plate to provide adequate support for drilling.

12.5.5 Drill Jig

A drill jig permits holes to be drilled in a number of identical pieces, **Figure 12-44**. This clamping device supports and locks the work in the proper position. The jig uses bushings to guide the drill to the correct location. This makes it unnecessary to lay out each individual piece for drilling.

12.6 Cutting Speeds and Feeds

The ***cutting speed*** is the speed at which the drill rotates. The ***feed*** is the distance the drill moves into the work with each revolution. Both are important considerations because they determine the time required to produce the hole. 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's cutting edge circumference travels per minute.

12.6.1 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.

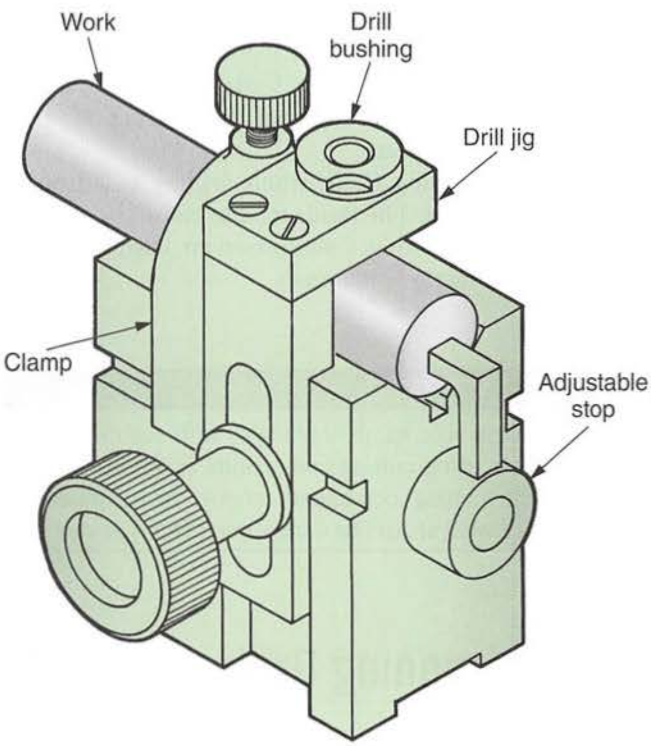
Because so many variables affect results, there can be no hard-and-fast rule to determine the exact cutting speed and feed for a given material. For this reason, the drill speed and feed tables in **Figure 12-45** indicate only recommended speeds and feeds. These are a starting point and can be increased 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 rate that is too low causes 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 feed is too high.

12.6.2 Speed Conversion

A problem arises in setting a drill press to the correct speed because its speed is given in revolutions per minute (rpm),



Goodheart-Willcox Publisher

Figure 12-44. A typical drill jig for holding round stock for drilling through center.

Recommended Speeds for Various Materials

Material	Speed (feet per minute)
Aluminum	200–300
Brass	150–350
Bronze	65–250
Cast iron	50–150
Stainless steel	30–100
Low-carbon steel	60–100
Medium-carbon steel	50–100
High-carbon steel	40–90

Recommended Feeds for Various Drill Sizes

Drill Diameter (inches)	Feed Rate (inches per revolution)
Less than 1/8	.001–.003
1/8–1/4	.002–.005
1/4–1/2	.004–.01
1/2–1	.008–.02
Greater than 1	.015–.025

Goodheart-Willcox Publisher

Figure 12-45. Drill speed and feed tables. These tables offer starting points for drilling different materials using drills of varying sizes. Feeds and speeds should be increased or decreased depending on the specific metal being drilled and the condition of the drill press.

but recommended drill cutting speed (CS) is given in feet or meters per minute (fpm or mpm). The following formula determines the rpm to operate any diameter drill (D) at any specified speed:

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

Drill speed problem. At what speed (rpm) must a 1/2″ diameter HSS drill rotate when drilling aluminum?

To solve this problem, follow the steps below:

1. Refer to the speed table, **Figure 12-45.** It gives the recommended cutting speed for aluminum (250 fpm).
2. Convert the drill diameter (1/2″) to a decimal fraction (0.5).
3. Substitute the values into the formula:

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

$$= \frac{4 \times 250}{05}$$

$$= 2000 \text{ rpm}$$

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)

12.6.3 Drill Press Speed Control Mechanisms

With some drill presses, it is possible to set a dial to the desired rpm. However, most conventional drilling machines cannot be set 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 12-46.** 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 the motor speed and pulley diameters are known.

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

Figure 12-46. With step-pulley speed control, the belt is transferred to different pulley ratios to change drill speed.

12.7 Cutting Fluids

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 fluids 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 of the chips to weld to the lips. Cutting fluids also improve hole finish and aid in the rapid removal of chips from the hole, **Figure 12-47.**

There are several kinds of cutting fluids. Many of them must be applied liberally. However, some newer fluids should be applied sparingly. Carefully read the instructions provided by the manufacturer.

Avoid using cutting fluids 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, works when drilling these materials.

SAFETY NOTE

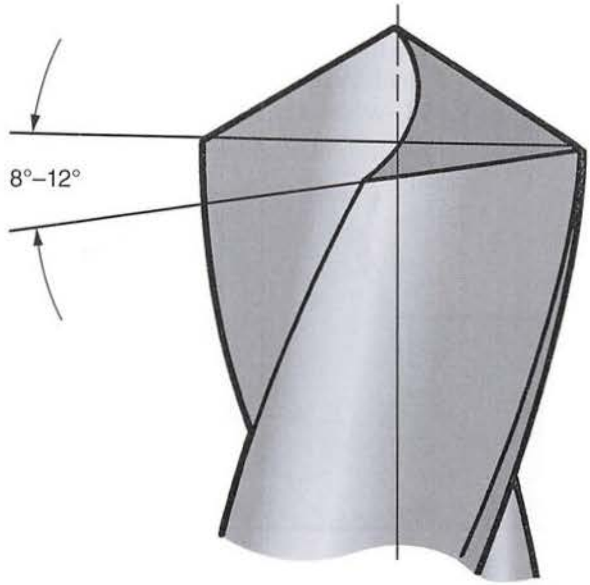
Always wash your hands thoroughly with soap and warm water after using cutting compounds and fluids. Before using the cutting compound, check the container to determine what should be done if you get any in your eye.

12.8 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.

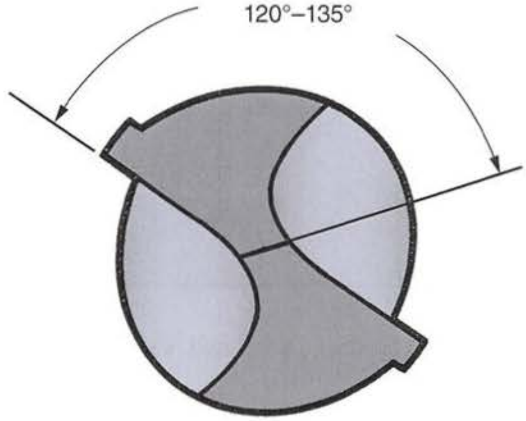
BIG Kaiser Precision Tooling Inc.

Figure 12-47. Cutting fluids minimize overheating and distortion of workpieces. Reduced heat also greatly extends the life of the drill. A coolant-hole drill allows the cutting fluid to flow through the drill to provide faster metal removal by flushing chips out of the hole.



Goodheart-Willcox Publisher

Figure 12-48. Lip clearance of 8" to 12" is satisfactory for most drilling.



Goodheart-Willcox Publisher

Figure 12-49. Proper angle of drill dead center.

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 naturally in still air. The shock of sudden cooling may cause it to crack.

12.8.1 Factors to Consider When Sharpening Drills

Three factors must be considered when repointing a drill. These factors are 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. To cut effectively, the heel (part of the point back of the cutting edge) must be relieved, **Figure 12-48**. Without this lip clearance, it is impossible for the lips to cut. If there is too much clearance, the cutting edges are weakened. Too little clearance results in the drill point merely rubbing without penetrating 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 12-49**.

Length and Angle of Lips

The material being drilled determines the proper point angle. The angles must be the same in relation to the axis. 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 12-50**.

Proper Location of Dead Center

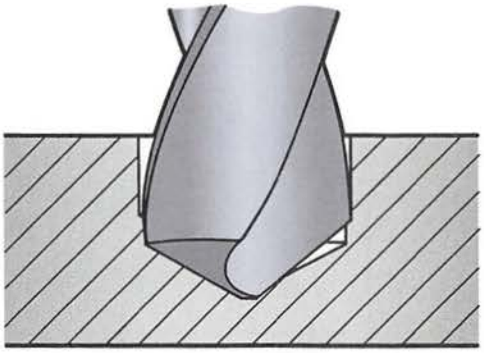
The drill dead center must be accurate. Lips of different lengths result in oversized holes, causing "wobble." This places tremendous pressure on the drill press spindle and bearings. See **Figure 12-51**.

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 12-52.**

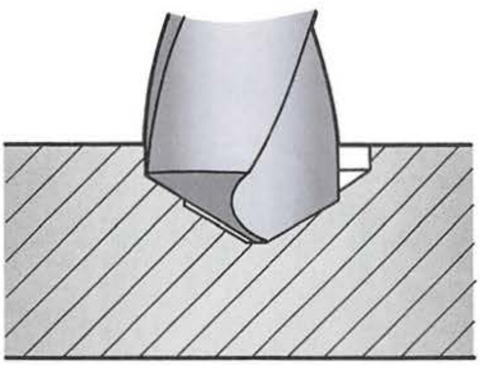
The web of a drill increases in thickness toward the shank, **Figure 12-53.** When a drill has been shortened by repeated grindings, the web must be thinned to minimize the pressure required to make the drill penetrate the material. The thinning must be done equally on both sides of the web and care 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 12-54.**



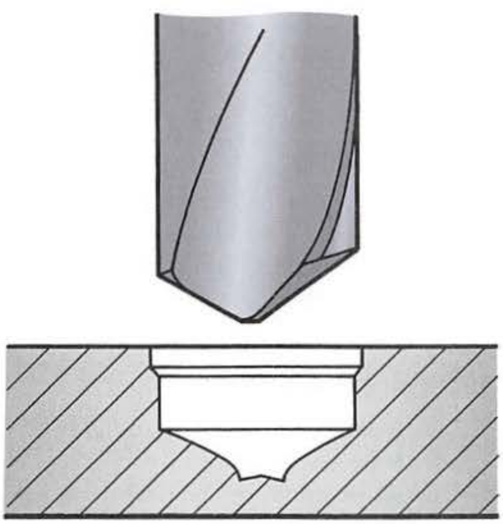
Goodheart-Willcox Publisher

Figure 12-50. Unequal drill point angles will produce an oversize hole.



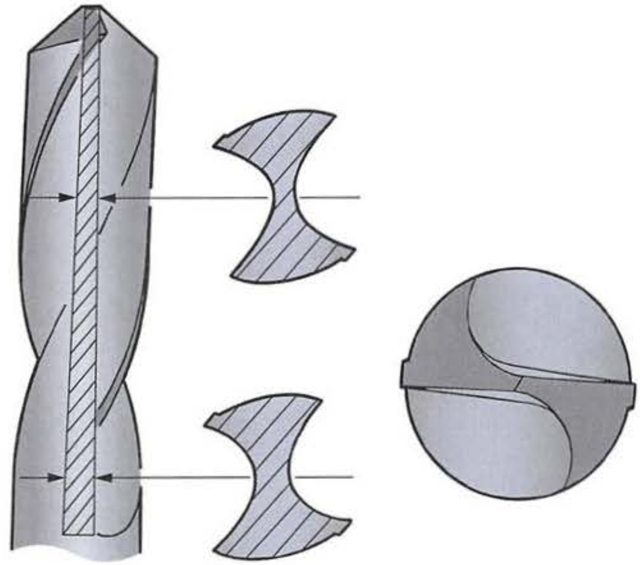
Goodheart-Willcox Publisher

Figure 12-51. Hole produced when drill is sharpened off center.



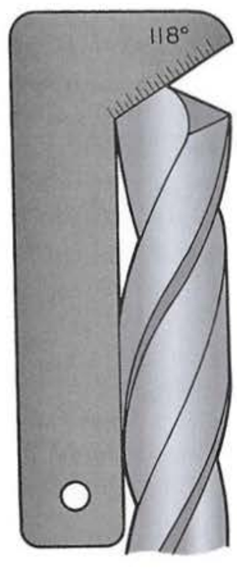
Goodheart-Willcox Publisher

Figure 12-52. This is the type of hole produced when the drill point has unequal point angles and is sharpened off center.



Goodheart-Willcox Publisher

Figure 12-53. The drill web tapers down toward the tip. The point is sometimes relieved to improve cutting action.



Goodheart-Willcox Publisher

Figure 12-54. Using a drill point gage helps ensure proper drill sharpening. For general drilling, an included angle of 118° is used.

12.8.2 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.

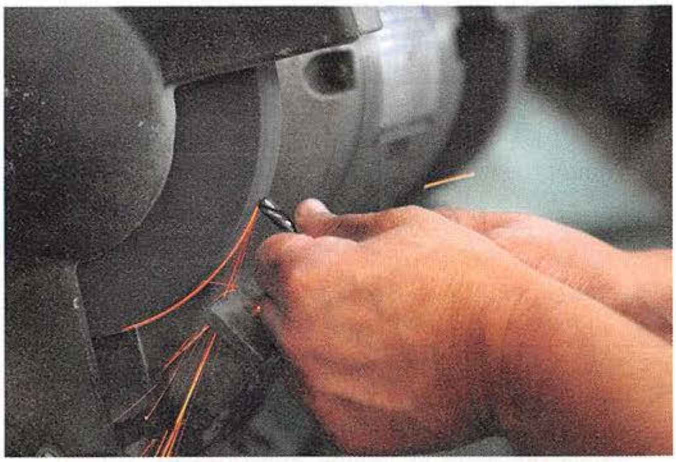
Many hand sharpening techniques have been developed. The following technique is suggested:

1. Grasp the drill shank with your right hand and the rest of the drill with your left hand. See **Figure 12-55.**
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 from center on a 7″ diameter wheel, for example).
3. Stand so the centerline of the drill is at a 59° angle to the centerline of the wheel, **Figure 12-56**. 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.
5. Repeat the operation on each lip until the drill is sharpened. Do not quench HSS 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.

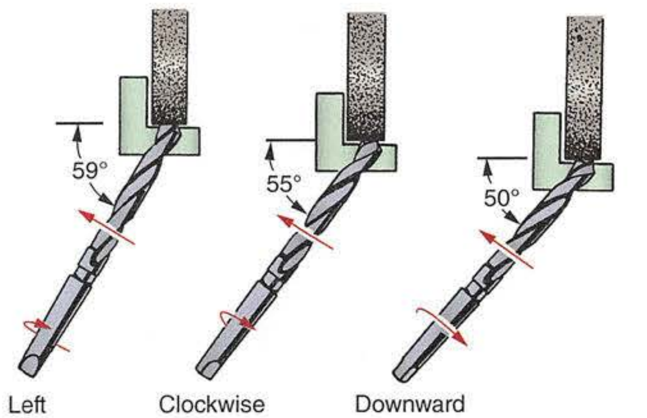
To test a sharpened drill, drill a hole in soft metal and observe the chip formation. When the drill is properly sharpened, chips will come out of the flutes in curled spirals of equal size and length. Tightness of the chip spiral is governed by the rake angle, **Figure 12-57**.

A standard drill point has a tendency to stick when used to drill brass. For drilling brass, sharpen the drill using the modified rake angle shown in **Figure 12-58**.



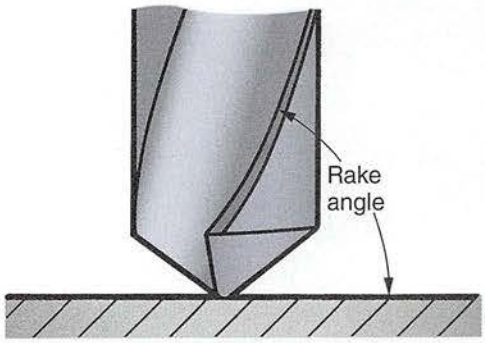
OSG Tap & Die, Inc.

Figure 12-55. One recommended way to hold a drill while sharpening it.



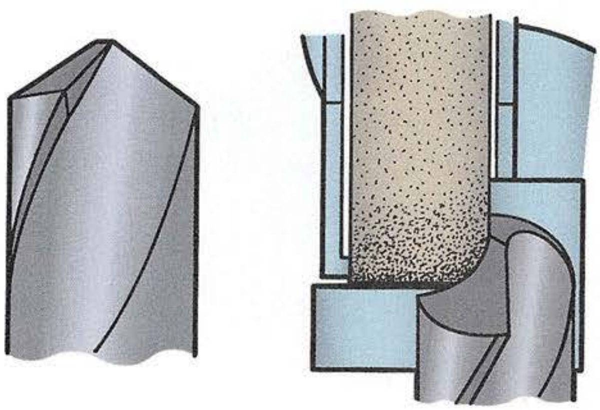
Goodheart-Willcox Publisher

Figure 12-56. 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.



Goodheart-Willcox Publisher

Figure 12-57. Rake angle of the drill.



Goodheart-Willcox Publisher

Figure 12-58. Modified rake angle for drilling brass.

12.8.3 Drill Grinding Attachments

A drill sharpening device ensures that cutting edges of the drill will be uniform. An attachment for conventional grinders is shown in **Figure 12-59**. 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.

12.8.4 Changing Drill Point Angles

In a manufacturing setting, where reducing the time it takes to drill holes can add up to a significant cost savings, the drilling operation can be made more efficient by changing the point angle on the tip of the drill. The typical included angle on most drills is 118°. However, drills with an included angle of 135° are also readily available. The flatter angle improves cutting efficiency on high-strength materials such as stainless steel. For softer materials, such as aluminum and magnesium, a sharper drill point of 90° to 118° can improve cutting speeds. See **Figure 12-60**.

Rush Machinery, Inc.

Figure 12-59. A drill and tool grinder with built-in point splitting and web thinning capabilities. The user can choose from a variety of drill points. Machines of this type are available in semiautomatic and automatic versions.

12.9 Drilling

Obeying a few simple rules will help you drill accurately. Use the following procedure:

1. Carefully study the drawing to determine hole locations before you begin. Lay out the positions and mark the intersecting lines with a prick punch.

SAFETY NOTE

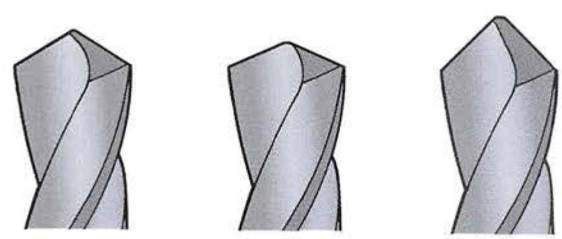
Observe extreme care in positioning the piece for drilling. A poorly planned setup may permit the drill to cut into the vise or drill table when it breaks through the work.

2. Secure an appropriate drill and check its size.
3. Mount the work solidly on the drilling machine.

SAFETY NOTE

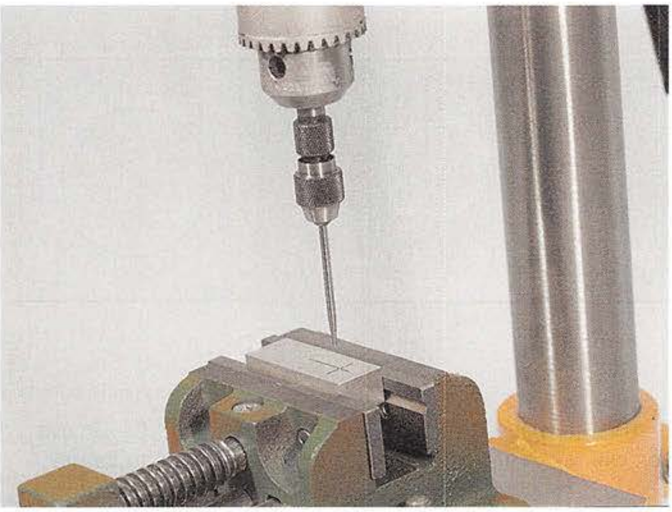
Never hold the work by hand. The workpiece could whip out of your hand and cause serious injuries.

4. Insert a ***center finder***, also known as a *wiggler*, into the drill chuck and position the point to be drilled directly under the chuck or spindle, **Figure 12-61**. Turn on the power and center the center finder point with your fingers. Position the work until the revolving center finder 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.
5. Remove the center finder and insert a center drill. Hand-tighten the chuck. Check to be sure the drill runs true. If it does, tighten the chuck with a chuck



Goodheart-Willcox Publisher

Figure 12-60. Drill point angles for common drilling operations.



Goodheart-Willcox Publisher

Figure 12-61. It is difficult to align a workpiece with centerlines by eye. To assist in this job, use a center finder (wiggler).

key. Remember to remove the key before starting the machine, **Figure 12-62.**

6. After center drilling, replace the center drill 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.

Goodheart-Willcox Publisher

Figure 12-62. After alignment with the center finder, center drilling ensures that the drill will make the hole in the proper location.

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 will 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. *Exception:* Do not use cutting fluid when drilling cast iron.
10. Clear chips and apply cutting fluid as needed.
11. The most critical part 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.

SAFETY NOTE

Never try to stop the chuck with your hand.

13. Clear the chips with a brush. Unclamp the work and use a file to remove all burrs.
14. Clean chips and cutting fluid from the machine. Wipe it down with a soft cloth. Return equipment to storage after cleaning.

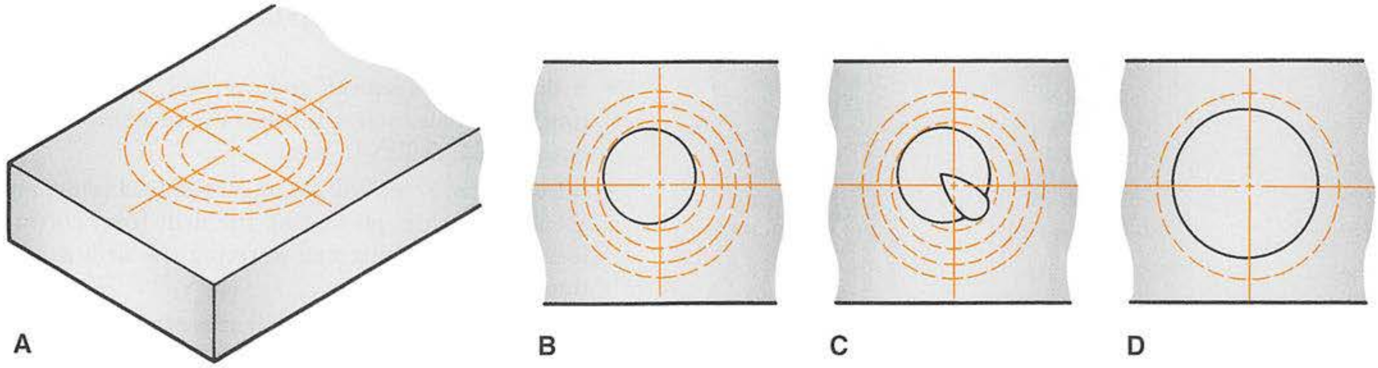
If a hole must be located precisely, additional precautions can be taken to ensure that the hole will be drilled where it is supposed to be drilled. After the center point has been determined, a series of proof circles are scribed, **Figure 12-63A.** They will serve as reference points to help determine whether the drill remains on center as it starts to penetrate the material.

Even when work is properly centered, the drill may "drift" when the hole is started. 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 must be corrected before the full diameter of the drill is reached, **Figure 12-63B.**

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 12-63C.** This groove will "pull" the drill point to the desired center. Repeat the operation until the hole is centered in the proof circles, **Figure 12-63D.**

12.9.1 Drilling Larger Holes

On drills larger than 1/2″ (12.5 mm), a conventional dead center requires more than 50% of the force required for drill penetration. Even then, the drills have a tendency to run



Goodheart-Willcox Publisher

Figure 12-63. 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 work only if the drill has not begun to cut to its full diameter.

off-center. Feed pressure can be greatly reduced and accuracy improved by first drilling a ***pilot hole*** (lead hole). See **Figure 12-64**.

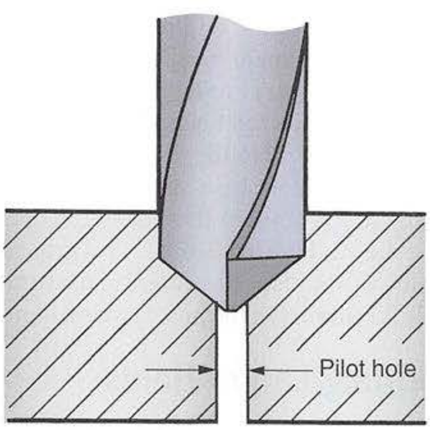
The small pilot hole permits pressure to be exerted directly on the cutting edges of the large drill, causing it to drill faster and more accurately. The pilot hole should have a diameter as large as, or slightly larger than, the width of the dead center.

A pilot hole can often be eliminated by changing the drill point geometry as shown in **Figure 12-65**. However, they require specialized grinding skills or dedicated drill sharpening equipment.

The Winslow-helical drill point is another design that eliminates the pilot hole and improves drilling efficiency. See **Figure 12-66**. It has a raised, S-shaped dead center that allows it to immediately cut metal into chips instead of pushing or extruding metal as a conventional straight dead center does.

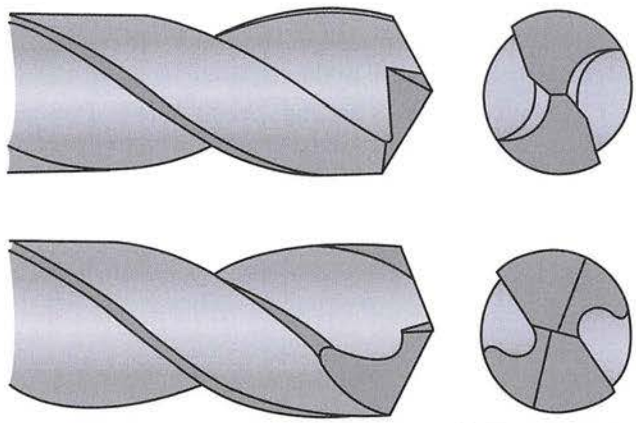
Because of the limited room for chips in drill flutes, it is always desirable to have them broken into small pieces. Coiling chips must be avoided, especially in deep hole drilling. Coiling tends to clog the flutes and prevent cutting fluid from reaching the cutting edges.

Ductile materials, including many aluminum alloys, require chipbreakers to produce proper chip breakup. **Figure 12-67** illustrates several effective chipbreaker designs.



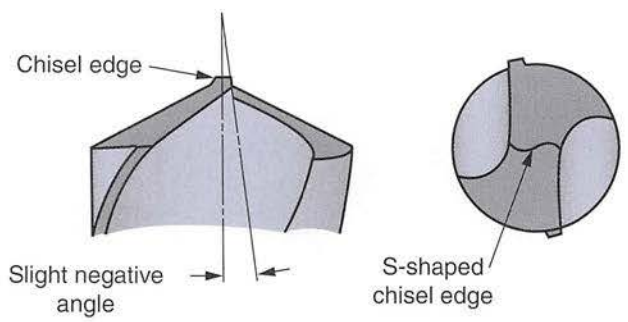
Goodheart-Willcox Publisher

Figure 12-64. A pilot hole makes drilling a large hole much easier.



Goodheart-Willcox Publisher

Figure 12-65. Pilot holes can often be eliminated by changing drill point geometry. Two designs are shown.



Goodheart-Willcox Publisher

Figure 12-66. The Winslow-helical drill point eliminates the need for a pilot hole, but it requires a specially designed drill point grinder.

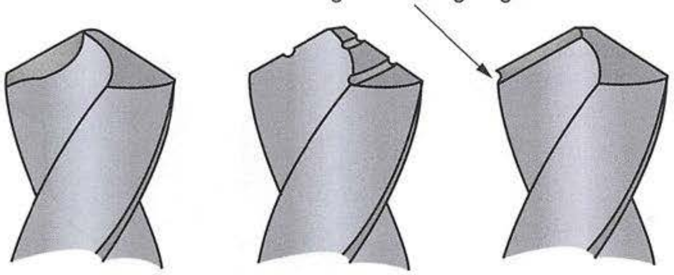


Figure 12-67. Chipbreaker designs that have proven successful in preventing metal coiling.

12.9.2 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 12-68**. 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:

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 the 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 12-69**. 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.

If a large number of identical parts must be drilled, a drill jig may be useful, **Figure 12-70**. The drill jig automatically positions and centers each piece for drilling.

12.9.3 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 12-71**. Using a drill press fitted with a depth stop or depth gage is the quickest way to achieve proper depth when drilling blind holes, **Figure 12-72**.

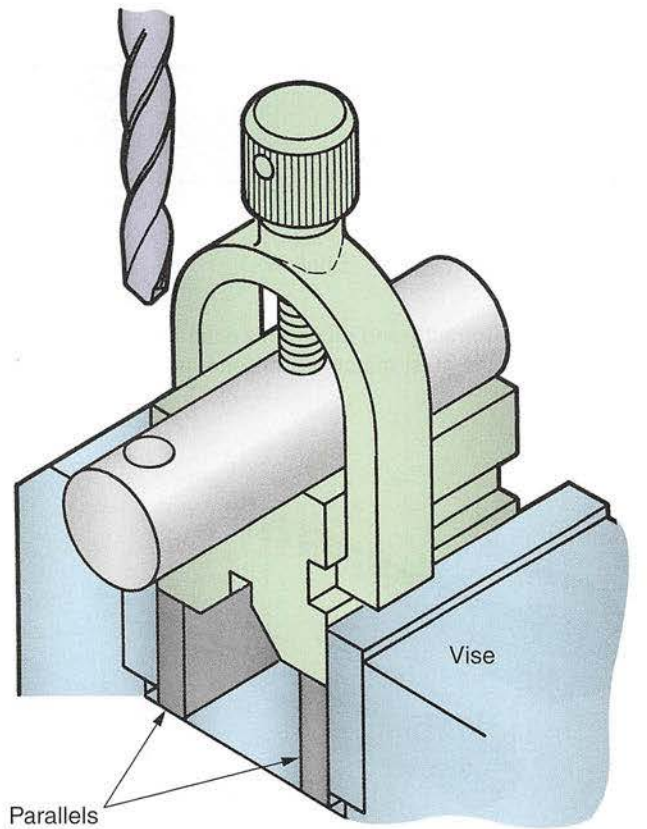


Figure 12-68. The V-block eliminates many difficulties when drilling round stock. Be sure the drill will clear the V-block when it comes through the material.

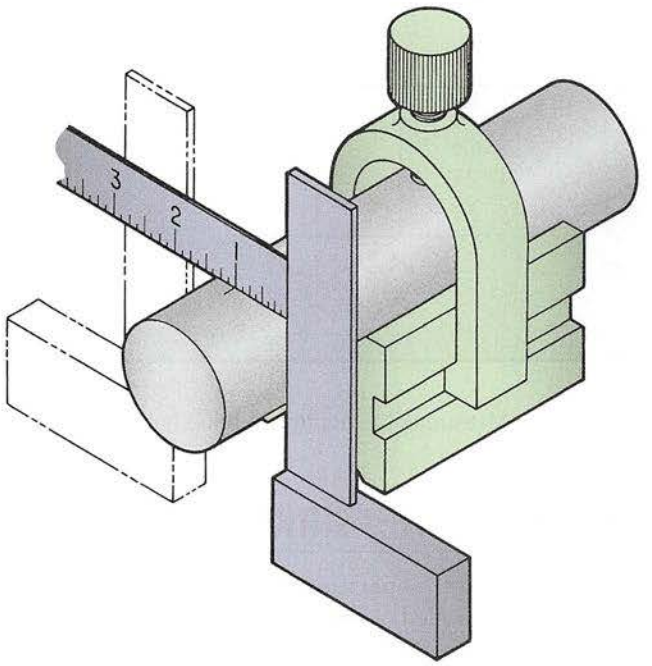
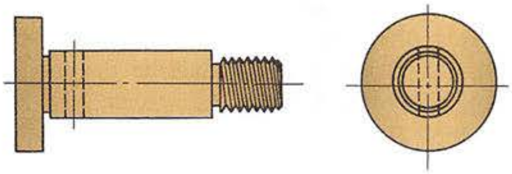
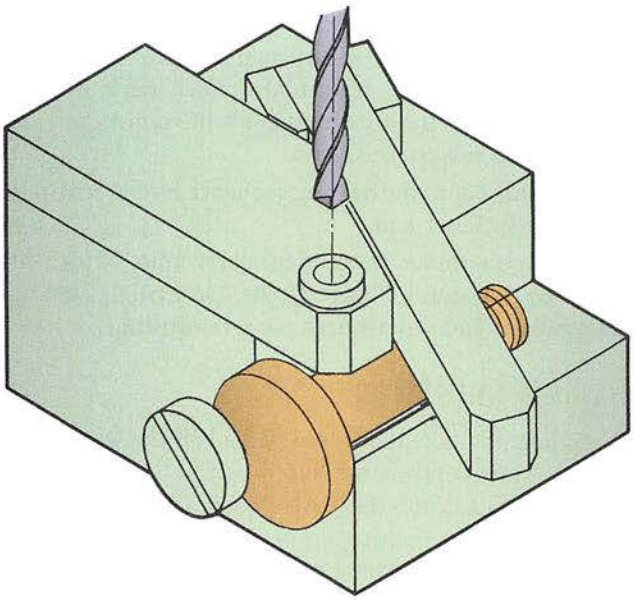


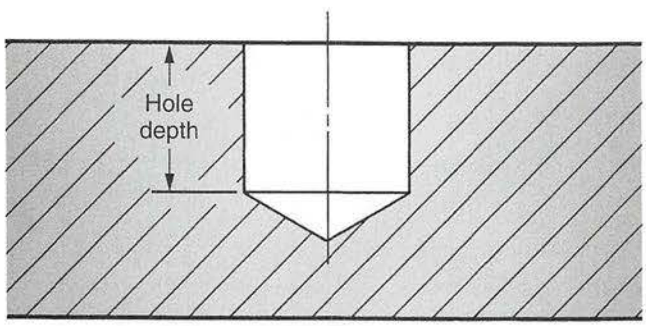
Figure 12-69. Using a square to center round stock in a V-block.





Goodheart-Willcox Publisher

Figure 12-70. This typical drill jig has an arm that lifts to allow easy insertion and removal of the part being drilled.

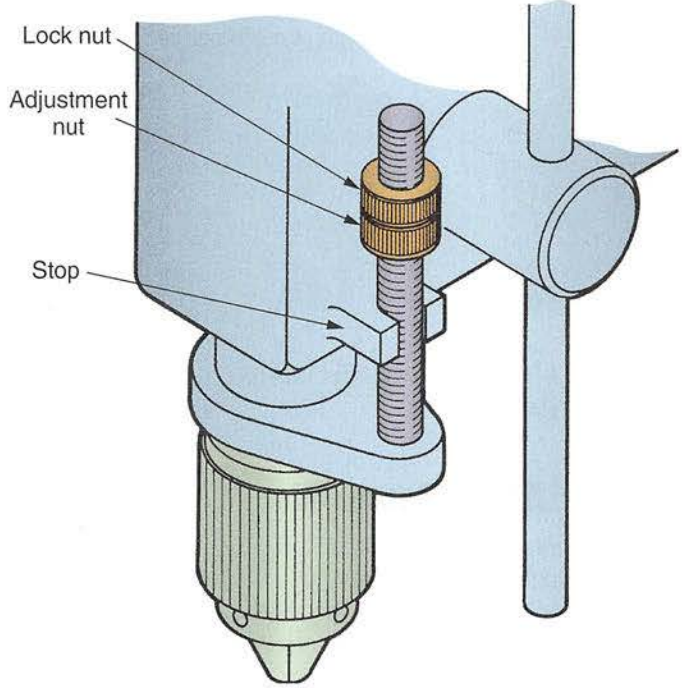


Goodheart-Willcox Publisher

Figure 12-71. Measuring the depth of a blind hole.

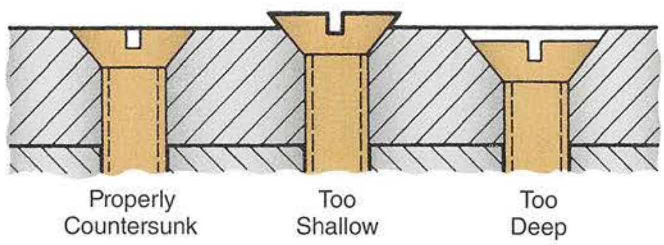
12.10 Countersinking

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, **Figure 12-73**. The tool used to machine countersinks is called a *countersink*, **Figure 12-74**. Countersinks are available with cutting-edge included angles of 60°, 82°, 90°, 100°, 110°, and 120°. Countersinks are also used for deburring holes.



Goodheart-Willcox Publisher

Figure 12-72. A depth gage attachment provides easy adjustment of the distance the drill moves into the work.



Goodheart-Willcox Publisher

Figure 12-73. Correctly and incorrectly countersunk holes. The countersink angle must match the fastener head angle.

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

Figure 12-74. These six-fluted countersinks come in various sizes.

CAREER CONNECTION

Industrial Production Manager

What does an industrial production manager do?

Industrial production managers are skilled leaders who manage the daily activities of manufacturing plants and other facilities. These professionals decide how best to use the time, resources, and people available to make sure production stays on schedule and within budget. They work with all members of their team and with managers from other departments to ensure the production process runs smoothly from start to finish.

What education and skills are needed to be an industrial production manager?

Managers typically have a bachelor's degree in industrial engineering or business administration. Certification in operations management, while not required, demonstrates higher levels of competency. Management positions also require expert use of soft skills, including interpersonal skills, problem-solving ability, leadership skills, and time management. In addition, managers may have extensive work experience.

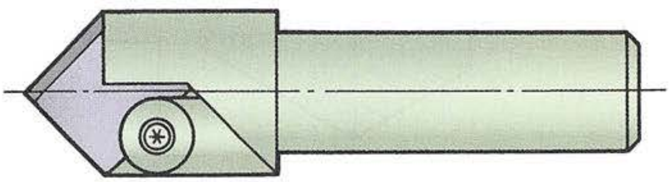
What is it like to be an industrial production manager?

Industrial production managers often work with manufacturing companies that produce fabricated metal products, transportation equipment, chemicals, and machinery. While at work, they often move between the production area and the office. Managers in the production area are exposed to the same hazards as workers and should follow safety procedures and wear protective equipment.

Recently, the *Occupational Outlook Handbook* reported median annual wages for industrial production managers of $93,900, with highest wages paid in chemical manufacturing.

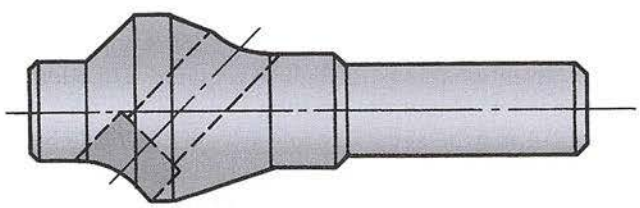
Countersinks with indexable carbide inserts, are available in a number of sizes and point angles, **Figure 12-75.** They have two cutting edges per insert and do not require resharpening. Cutting speeds are five to ten times higher than with HSS countersinks.

A countersink with a single cutting edge, **Figure 12-76,** is free-cutting and produces minimum chatter. Chips produced by the cutting edge pass through the hole and are ejected.



Goodheart-Willcox Publisher

Figure 12-75. Countersinks with indexing carbide inserts have a life five to ten times longer than similar HSS countersinks.



Goodheart-Willcox Publisher

Figure 12-76. Countersink with a single cutting edge and pilot.

To use a countersink, follow these general guidelines:

1. Use a cutting speed of about one-half that recommended for a similar size drill. This will minimize 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.

12.11 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 12-77.**

The counterbore tool has a guide, called a *pilot,* that keeps it positioned correctly in the hole. Solid counterbores are available. However, counterbores with interchangeable pilots and cutters are commonly used, **Figure 12-78.** They can be changed easily from one size cutter or pilot to another size. A drop of oil on the pilot prevents it from binding in the drilled hole.

Counterbores with indexable carbide inserts are also available, **Figure 12-79.** When the cutting edges become dull, new edges can be indexed into place without affecting the opening diameter. Costly sharpening is eliminated.

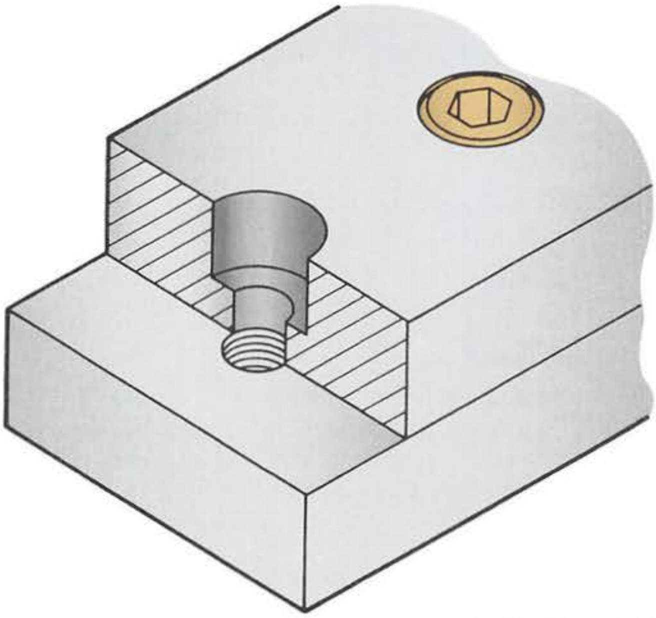


Figure 12-77. A sectional view of a hole that has been drilled and counterbored to receive a socket-head screw.

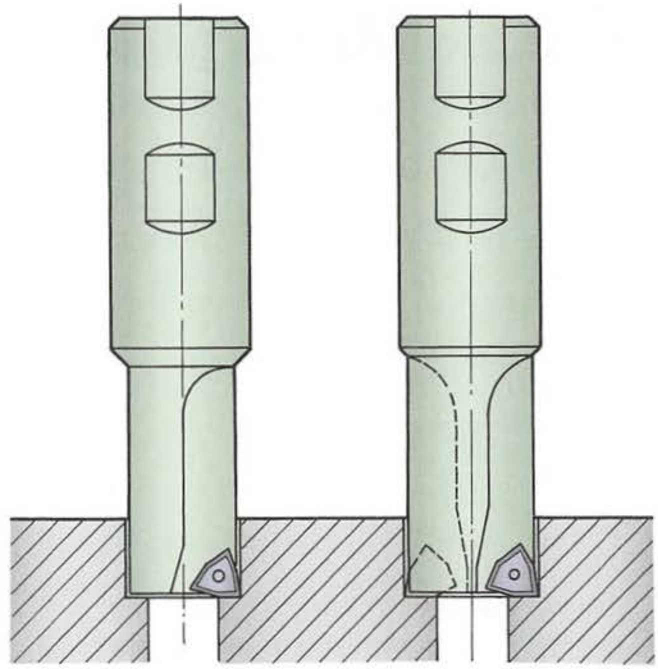


Figure 12-79. Counterbores with carbide indexable inserts. The inserts are rotated when a cutting edge becomes dull.

Figure 12-78. Different sizes of straight-shank counterbores.

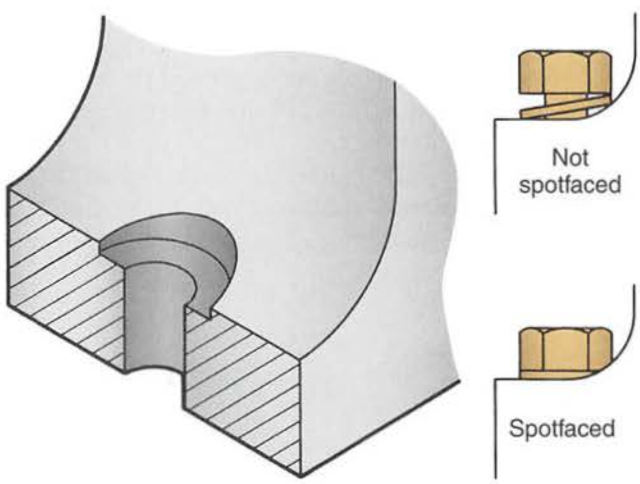


Figure 12-80. Sectional view of a casting with a mounting hole that has been spotfaced. Profile drawings show the casting before and after spotfacing. The bolt head cannot be drawn down tightly until the mounting hole is spotfaced.

12.12 Spotfacing

Spotfacing is the operation in which a circular spot is machined on a rough surface (such as a casting or forging) to provide a bearing surface for a bolt, washer, or nut. A counterbore may be used for spotfacing, although a special tool manufactured for inverted spotfacing is available, **Figure 12-80**.

Backspotface and backcounterbore tools are required to perform operations in areas where conventional tools cannot be used. See **Figure 12-81**. The cutting point is lifted up into the workpiece, rather than being pushed down into it, as shown in **Figure 12-82**.

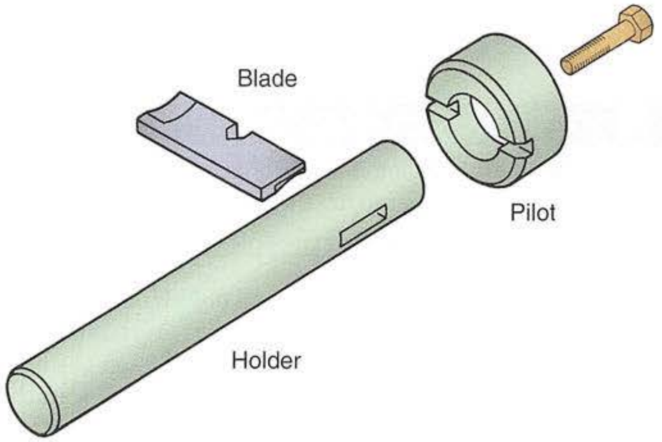
Large-diameter openings can be counterbored, spotfaced, or drilled with interchangeable drilling, spotfacing, and counterboring tools, **Figure 12-83**. A pilot hole is drilled with a conventional twist drill. The required size pilot and blade are inserted and the opening is made. For very large openings, it may be necessary to use a smaller blade before machining the specified size. Multitool blades are available in sizes from 1 1/8″ diameter to 4″ diameter.

Parlec, Inc.

Figure 12-81. 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.

Heule Precision Tools

Figure 12-82. An example of how backcounterboring and backspotfacing tools work.



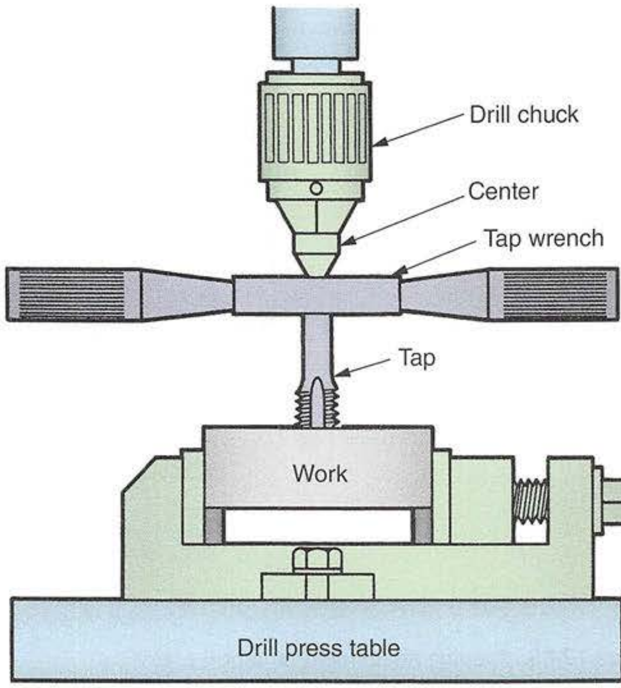
Goodheart-Willcox Publisher

Figure 12-83. An interchangeable drilling, spotfacing, and counterboring tool, also known as a *multitool*. Blades for producing holes up to 4″ in diameter are available.

12.13 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 12-84.**
2. With the work clamped in the machine, insert a small 60° center in the chuck. The center holds the tap vertically.
3. Place the center point in the tap's center hole.



Goodheart-Willcox Publisher

Figure 12-84. Setup for hand tapping on a drill press. Work must be mounted solidly.

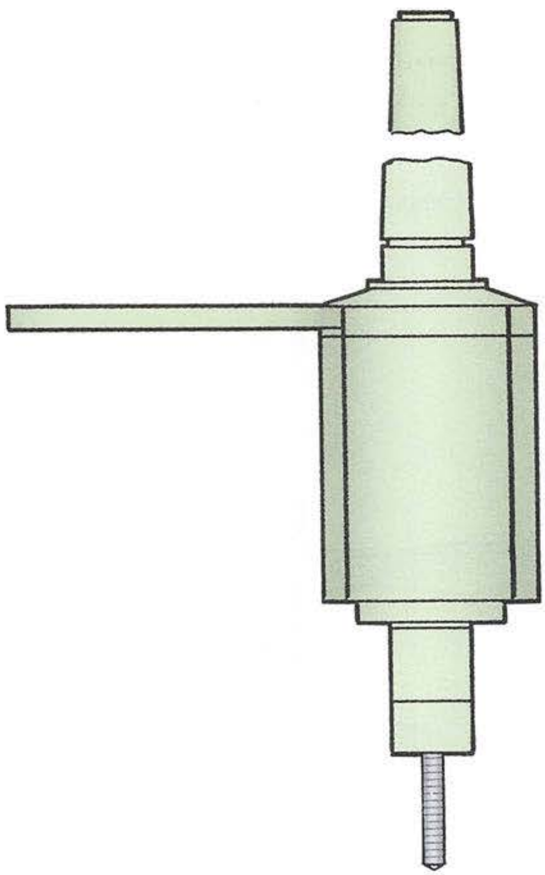
4. Feed the tap into the work by pressing down on the feed handle and turning the tap with a tap wrench.

SAFETY NOTE

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 be done with power only by using a tapping attachment, **Figure 12-85**. This device fits the standard drill press. It has reducing gears that slow 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.



Goodheart-Willcox Publisher

Figure 12-85. Tapping attachment on a drill press.

12.14 Reaming

Reaming produces holes that are extremely accurate in diameter and have an exceptionally fine surface finish. Machine reamers are usually manufactured from high-speed steel. Some are fitted with carbide cutting edges.

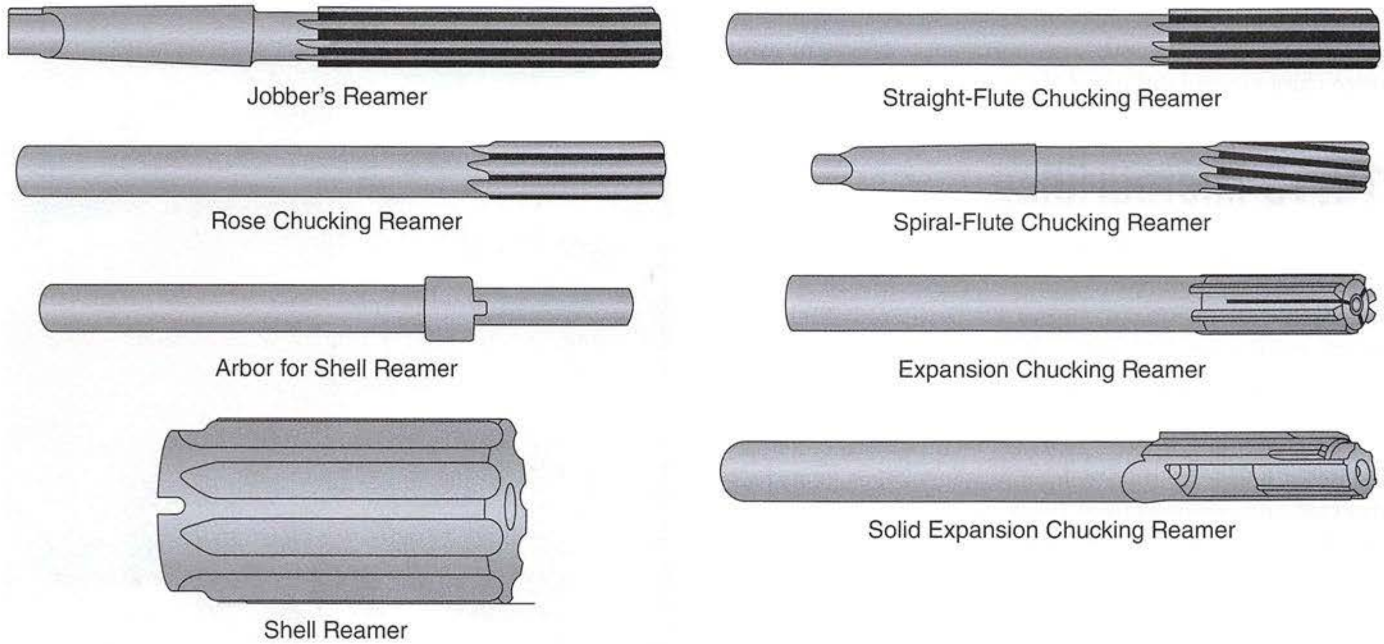
12.14.1 Types of Machine Reamers

Machine reamers are made in a variety of sizes and styles. Descriptions of a few common types follow. Refer to **Figure 12-86**.

- 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.
- Chucking reamers are manufactured with both straight and taper shanks. They are similar to a jobber's reamer but their flutes are shorter and deeper. They are 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.
- Expansion chucking reamers are 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 to expand, reduce tool rigidity. This diminishes accuracy and surface finish. Also, cutting-edge clearance is reduced as the reamer expands, creating friction. 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. 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.



Goodheart-Willcox Publisher

Figure 12-86. Each type of reamer is suited for a different application. The solid expansion reamer has tungsten carbide cutting edges for extended cutting life.

12.14.2 Using Machine Reamers

Reamers are expensive precision tools. The quality of the finish and accuracy of the reamed hole 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.
- Set the cutting speed for a high-speed steel reamer at about two-thirds that of a similar size drill.
- Set the feed 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.

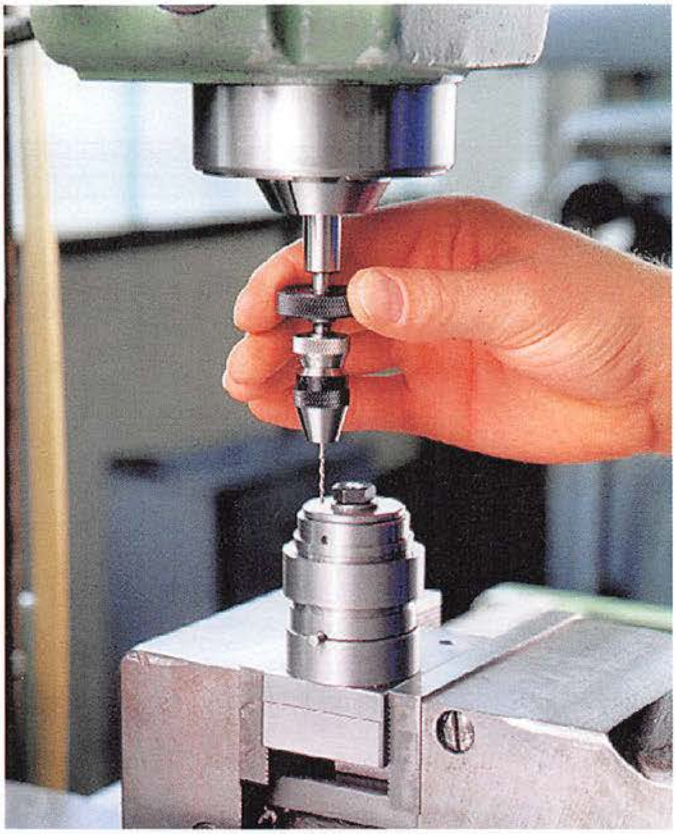
When not being used, reamers should be stored in separate containers or storage compartments. This will minimize chipping and dulling of the cutting edges.

12.15 Microdrilling

Specially designed machines, toolholding devices, and drills are required for drilling holes smaller than 0.020″ (0.508 mm) to within close tolerances, **Figure 12-87**. Microdrilling operations require very accurate spindles and collets to reduce drill flexing and breakage. Many microdrilling machines are controlled by computer numerical control (CNC) systems.

Microdrilling uses a "pecking" technique to cut 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 holes with other geometric shapes (such as square, rectangular, or hexagonal holes) are made by electrical discharge machining (EDM). EDM operations are discussed in more detail in a later chapter.



Royal Products, Division of Curran Manufacturing Corporation

Figure 12-87. This sensitive drill feed eliminates the need for a specialized microdrilling machine. It makes drilling small holes on a normal drill press easier, reduces tool breakage, and decreases the tendency for microdrills to "walk" off center.

Chapter Review

Summary

- Selection of the proper drilling machine, drill, and drill-holding device for specific drilling operations is important.
- Twist drills are the most common form of drill, but a large variety of special purpose drills are also available.
- Drills are held in the drill press by a chuck or tapered spindle.
- Proper work-holding devices, cutting speeds, feed rates, and cutting fluids are essential for safe, efficient, and accurate drilling operations.
- A properly sharpened drill is critical to efficient and safe drilling operations.
- A center finder can be used to help position the chuck or spindle directly below the point to be drilled.
- Drilling machines are used for a variety of operations other than drilling, including countersinking, counterboring, spotfacing, tapping, reaming, and microdrilling.
- Microdrills permit the drilling of very small holes, but require special tooling and the "pecking" technique.

Review Questions

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

1. A drill press 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 (HSS)
   B. carbon steel
   C. titanium
   D. Both A and B.
   E. None of the above.
4. List the two types of drill shanks.
5. _____ shank drills are used with a chuck.
6. _____ shank drills fit directly into the drill press spindle.
7. The spiral grooves that run the length of the drill body are called _____.
8. 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.
9. Drill sizes are expressed by what four series?
10. Name two techniques that can be used to determine a drill's size.
11. What device is used to enlarge a taper shank drill so it will fit a 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 a drill-holding device?
14. Name four types of work-holding devices used for drilling operations.
15. What happens to the drill if the feed rate is set too low?
16. Cutting 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.
17. What coolant should be used when drilling cast iron?
18. List the three factors that must be considered when repointing a drill.
19. What occurs when the cutting lips of a drill are not sharpened to the same lengths?
20. The _____ should be used frequently when sharpening to ensure a correctly sharpened drill.
21. The included angle of a drill point sharpened for general drilling is _____ degrees.
22. 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(n) _____ hole.
23. When drilling a pilot hole, the hole should be as large as, or slightly larger than, the width of the _____ of the drill point.
24. What is a blind hole?
25. How is the depth of a drilled hole measured?
26. What is the name of the operation used to cut a chamfer in a hole to receive a flat-head screw?
27. The operation used to prepare a hole for a fillister- or socket-head screw is called _____.

28. _____ is the operation that machines a circular spot on a rough surface to provide a bearing surface for the head of a bolt or nut.
29. What would happen if you tried to use drill press power to run a tap into the workpiece?
30. The _____ is almost identical to the hand reamer except that the shank has been designed for machine use.
31. A(n) _____ expansion reamer provides rigidity and accuracy not possible with conventional expansion reamers.
32. The cutting speed for a high-speed reamer is approximately _____ that for a similar-sized drill.
33. How should a reamer be removed from a finished hole?
34. Describe the "pecking" technique used in microdrilling.

CHAPTER 13
Offhand Grinding

Chapter Outline

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.
- Describe the use of wet and dry grinders.

Technical Terms

glazing
grinding
loading
offhand grinding
ring test
temper
tool rest
wheel dresser

G*rinding* is an operation that removes material by rotating an abrasive wheel or belt against the work, **Figure 13-1**. Grinding 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.

13.1 Bench and Pedestal Grinders

The bench grinder and pedestal grinder are the simplest and most widely used grinding machines. 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.

A bench grinder is one that has been fitted to a bench or table, **Figure 13-2**. The grinding wheels mount directly onto the motor shaft. A pedestal grinder is usually larger than a bench grinder and is equipped with a pedestal (base) that is fastened to the floor. See **Figure 13-3**. On most grinders, one wheel is coarse, for roughing, and the other is fine, for finish grinding.

Bench and pedestal grinders are classified as wet or dry. A dry grinder has no provision for cooling the work during grinding other than a water container. The part is dipped into the water. A wet grinder, **Figure 13-4**, has a coolant system built into the grinder. This system keeps the wheels constantly flooded with coolant. 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.

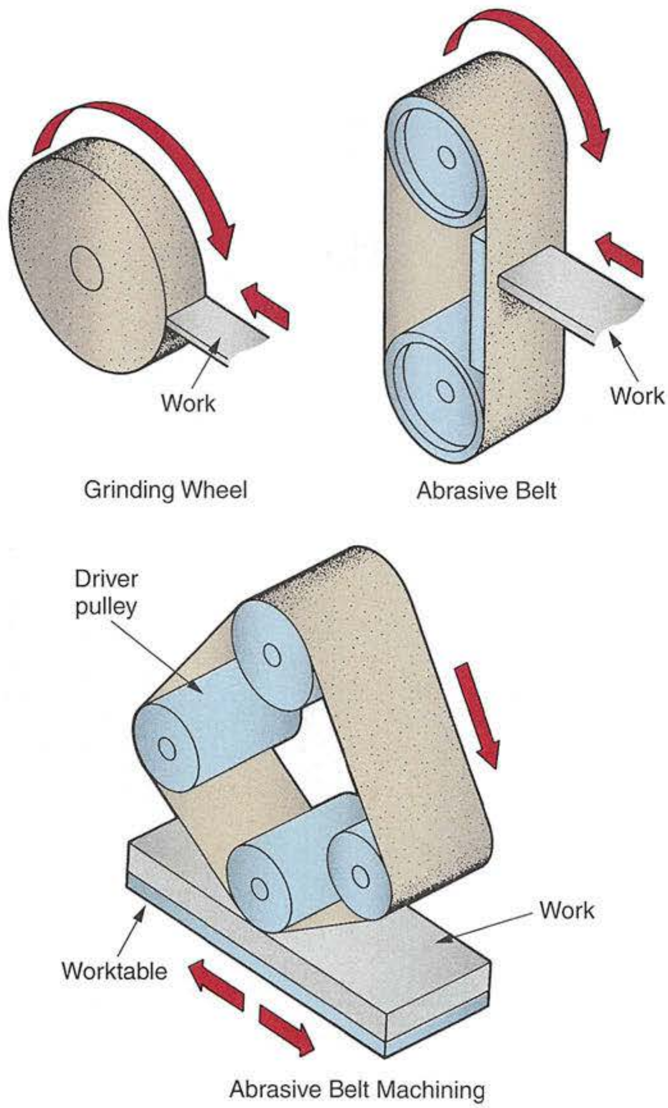
SAFETY NOTE

Wear safety glasses and be sure the grinder's eye shield is in place before doing any grinding.

The ***tool rest*** is provided to support the work being ground. It should be adjusted to within 1/16″ (1.5 mm) of the wheel, **Figure 13-5**. 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.

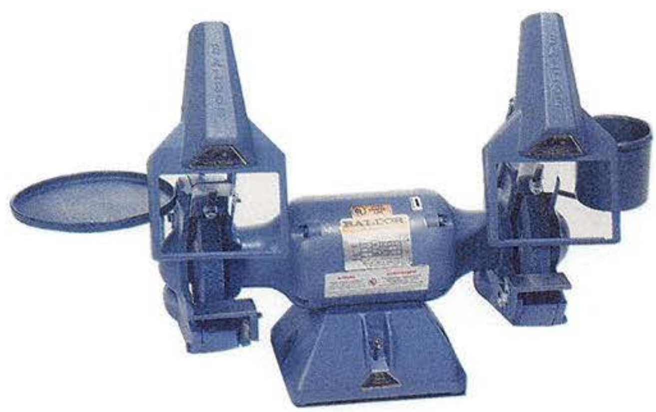
SAFETY NOTE

Do not make tool rest adjustments while the grinding wheels are revolving.



Goodheart-Willcox Publisher

Figure 13-1. Principles of operation for typical grinding machines.



Baldor

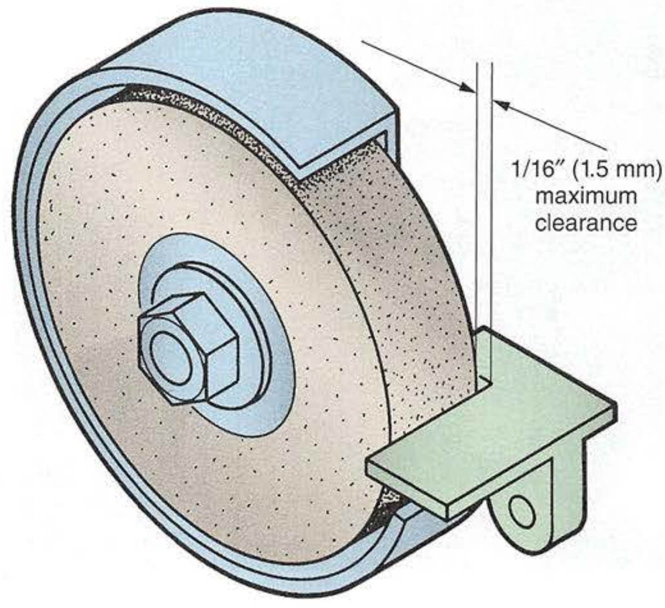
Figure 13-2. A bench grinder can be used for many tasks. Never operate a bench or pedestal grinder unless all safety devices are in place and in sound condition. The tool rest must be properly spaced and eye protection must be worn, even though the grinder has eye shields.

Baldor

Figure 13-3. This pedestal grinder has a roughing wheel on the left and a finishing wheel on the right.

Baldor

Figure 13-4. This grinder has a coolant attachment that keeps coolant dripping on the tool being sharpened.



Goodheart-Willcox Publisher

Figure 13-5. The tool rest must be spaced properly for safety. Maximum safe clearance is 1/16″ (1.5 mm).

13.2 Other Types of Grinders

Although bench and pedestal grinders are the most commonly used in the machine shop, other types of grinders are used for specific purposes. These include abrasive belt grinders and portable hand grinders.

13.2.1 Abrasive Belt Grinders

Abrasive belt grinders are heavy-duty versions of the belt and disc sanders found in woodworking, **Figure 13-6**. A 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 13-7**.

13.2.2 Portable Hand Grinders

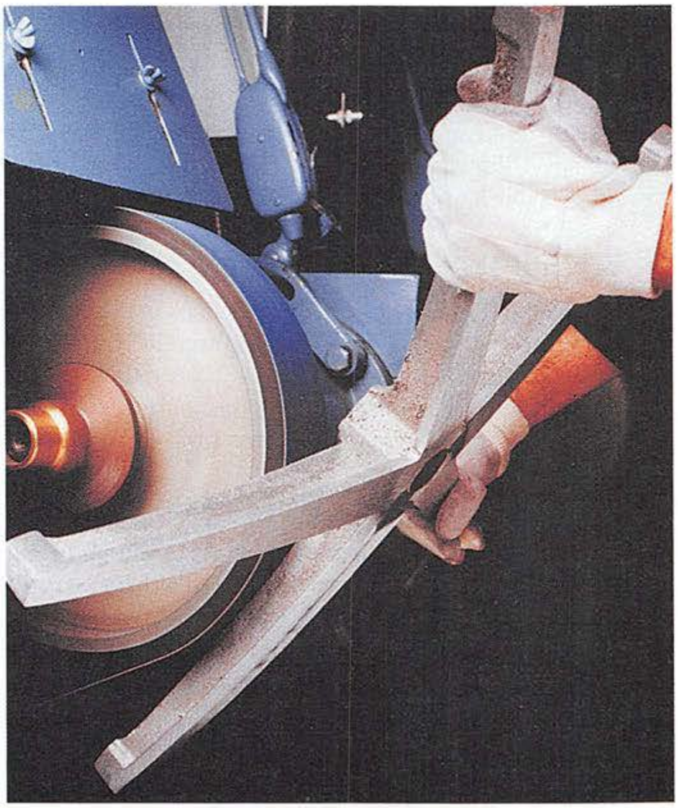
Many grinding jobs, from light deburring to die-polishing operations, are done with small, portable hand grinders. Flexible shaft grinders and precision microgrinders, **Figure 13-8**, are used to perform a variety of toolroom and production jobs. They can be powered by electricity or air.

13.3 Grinding Wheels

Grinding wheels are made of coarse abrasive grains bonded together into circular shapes, **Figure 13-9**. As the wheel is used, the dulled abrasive particles wear away from the wheel to reveal new, sharp particles underneath.

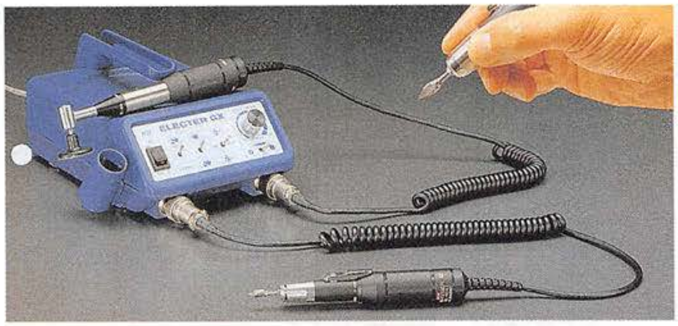
FEIN Power Tools, Inc.

Figure 13-6. An abrasive belt grinder. These machines can be used for offhand grinding or combined with other machine tools to perform more precise machining operations.



American Foundrymen's Society

Figure 13-7. An abrasive belt being used to clean up an aluminum casting of a base for an office chair.



NSK America

Figure 13-8. A precision electric microgrinder is helpful for working in recessed areas on parts.

Praethip Docekalova/Shutterstock.com

Figure 13-9. Grinding wheels are available in a variety of sizes and grains.

Grinding wheels can be a source of danger and should be examined frequently for concentricity (running true), roundness, and cracks. Wheels that have cracks or other defects can fly apart as the wheel speed increases after the grinder has been turned on. The pieces of the wheel can be thrown at high speed, causing damage to equipment or injury to workers in their path.

13.3.1 Ring Test

When changing wheels, use a ***ring test*** to check the new wheel before mounting it. First make sure the wheel is clean and free of dirt, dust, or any material that can deaden the sound of the ring. Next, suspend the ring through its center hole using a shaft or pin. The wheel can be lightly suspended on the operator's finger, as long as it can be turned easily. Using a piece of hard plastic, such as the handle of a screwdriver, tap the wheel on a line roughly 45° from vertical on the upper portion of the wheel. See **Figure 13-10.** Tap the wheel toward its outside edge, but stay at least one inch away from the edge. A wheel that is undamaged produces a clear, metallic ringing sound. A damaged wheel produces a dull thud.

If the wheel produces a ring, rotate it 180° so that the bottom is on the top, and test the other half of the wheel. The wheel passes the test if all four areas inspected yield a clear ringing sound. A wheel that passes the ring test is safe to place on the grinder.

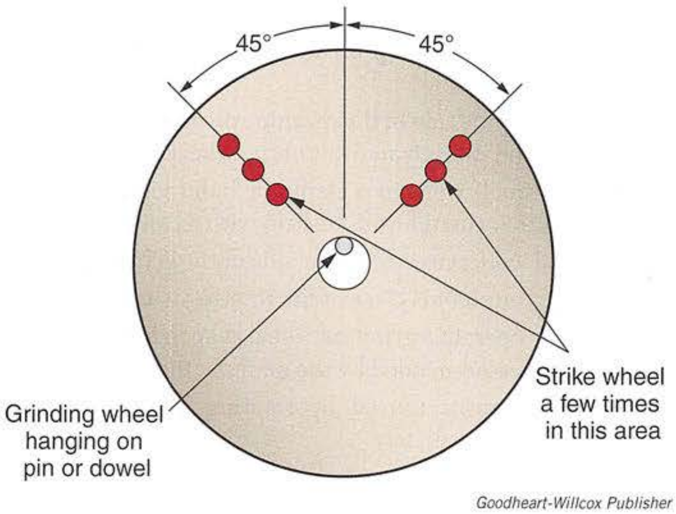


Figure 13-10. How to perform the ring test.

13.3.2 Wheel Dresser

The grinding wheels must run true and be balanced on the shaft. A ***wheel dresser*** is used to true the wheel and provide a better grinding surface, **Figure 13-11**. Support the wheel dresser on the tool rest and use both hands to hold it firmly against the wheel. Move it back and forth across the wheel face to remove a thin layer of stone. See **Figure 13-12**.

During normal grinding operations, the chips and dulled grains of the grinding wheel wear away to reveal new, sharp grains. ***Loading*** of the wheel can occur when grinding soft metals. Small pieces of the material clog the wheel instead of falling away. ***Glazing*** can occur when grinding very hard metals. The grains of the wheel tend to dull before they can fall away to reveal the sharp grains underneath. The shiny appearance of the grinding wheel surface indicates the amount of glaze.

Both loading and glazing cause the wheel to cut inefficiently. A wheel dresser should be used to remove any loading or glazing that may have formed during grinding operations.

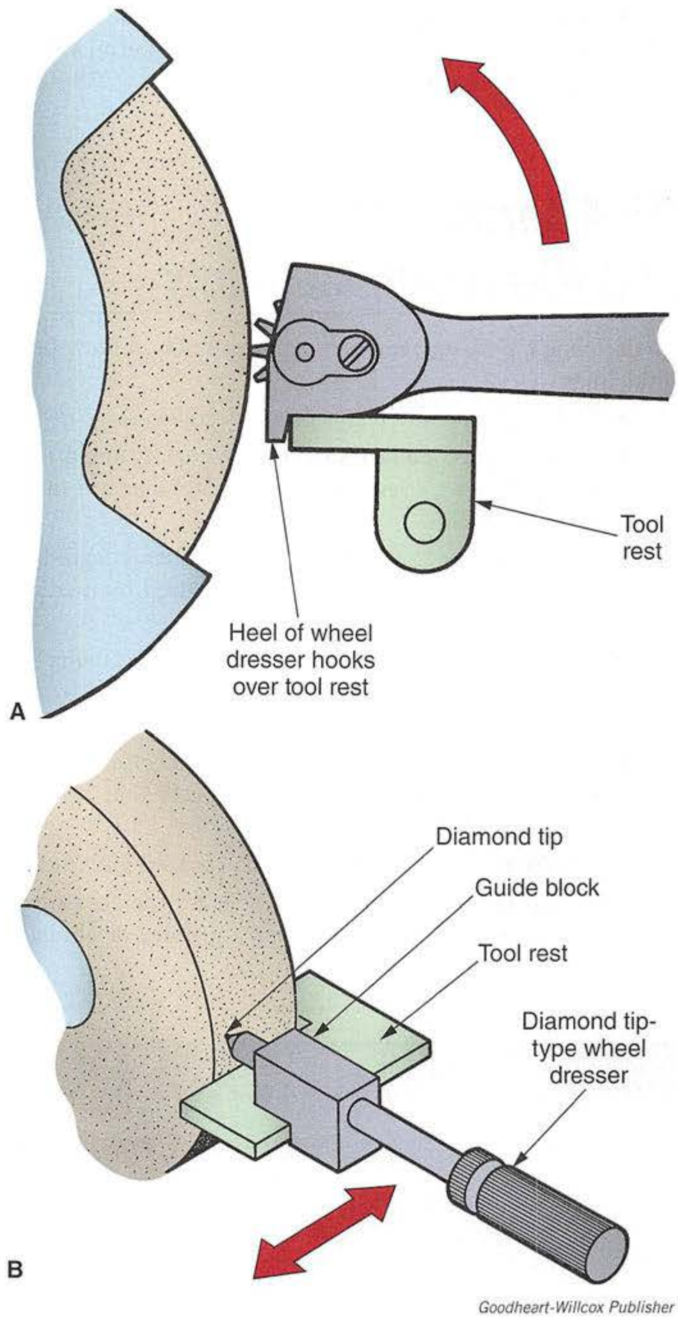


Figure 13-12. The proper way to use a mechanical wheel dresser. A—Move the tool back and forth over the face of the stone. Wear a dust mask, eye protection, and an apron when dressing wheels on grinders. B—Industrial diamonds are also used to dress and true grinding wheels. A guide block is used for grinders with slotted tool rests.

13.3.3 Grinding Rules

To obtain maximum efficiency from a grinder, observe the following recommendations:

Figure 13-11. A mechanical wheel dresser.

- Grind using the face of the wheel, not the sides.
- Move the work back and forth across the wheel face. This wears the wheel evenly and prevents grooves from forming.
- Keep the wheel dressed and the tool rests properly adjusted.

- When possible, grind soft metals (aluminum, brass, and copper) on an abrasive belt grinder because they tend to load grinding wheels.

13.4 Abrasive Belt and Wheel Grinder Safety

To use bench, pedestal, and belt grinders safely, follow these guidelines:

- 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.
- Check a grinding wheel for soundness before mounting 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.
- Be sure that 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.

13.5 Using a Dry 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 you apply too much pressure, the work will begin to "burn" or discolor. Minimize overheating by dipping the work into the water container from time to time. Take care to avoid excessive heat because the heat will "draw" (remove) the ***temper*** (hardness) and ruin the tool.

WORKPLACE SKILLS

Being an Efficient Employee

Employers want employees who are punctual, dependable, and responsible. They want their employees to be capable of taking initiative and working independently. Other desirable employee qualities include organization, accuracy, and efficiency.

No matter what your job is or how long you have had it, you can always improve your efficiency. Efficiency is a measure of how much you can do—and how well you can do it—with the resources you have. Like the world in general, the workplace is constantly changing. Look critically at the work you do, whether it is schoolwork or work at a part-time or full-time job. Examine the methods you use to get your work done. Methods that have been useful in the past can sometimes be replaced with newer, more efficient methods made possible by new technology.

General tips that may help you improve your efficiency include:

- At the beginning of each day, think about what you need to accomplish that day and set your priorities accordingly.
- Consider your energy levels during the day and match your energy level to your tasks. For example, if you think better in the morning, do tasks that require the most thought or brainpower in the morning.
- Organize your work environment. Identify items you need on a daily basis and keep them within easy reach. Not having to spend time searching for these items not only saves time, but also helps you retain your focus on the task at hand.

Keep the work moving across the wheel face to prevent the formation of grooves or ridges. Dress and retrue the wheel as necessary for maximum efficiency.

Never use a piece of cloth to hold the 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 especially designed for that purpose, **Figure 13-13**.

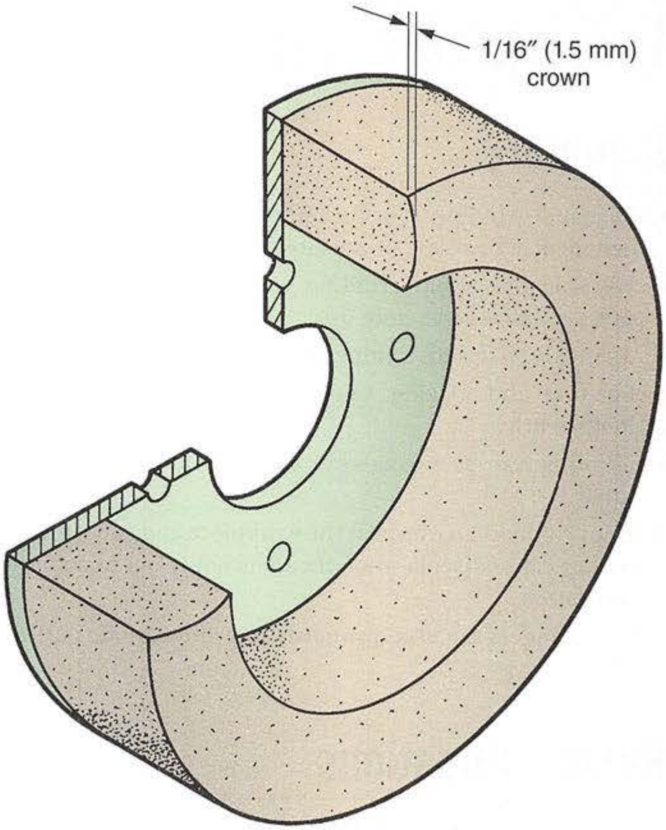
13.6 Using a Wet Grinder

A wet grinder is primarily used to grind carbide-tipped tools. Since carbide tools are 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. Silicon carbide or diamond-impregnated wheels should be used to grind the carbide tip.

Wet grinders should normally have a flat face, but a slightly crowned face should be used when grinding carbide-tipped tools, **Figure 13-14**. The crown minimizes the contact between the wheel and the work. This reduces the possibility of the tip being damaged by excessive heat.

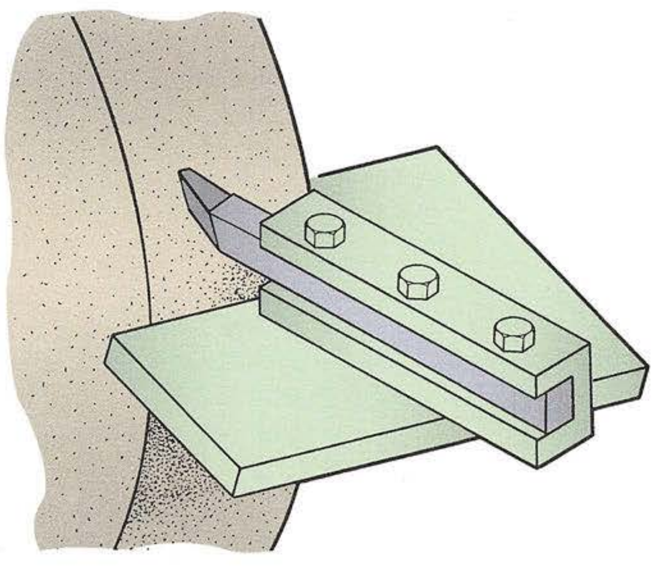
The coolant attachment must be adjusted to keep a full flow of liquid directed on the tool at all times. Adjust the tool rest to obtain the correct clearance angle, **Figure 13-15**. A protractor guide is helpful when compound clearance angles are required.

Use the entire face of the wheel. Keep the tool in continuous motion to minimize wheel wear. Dress and retrue the wheel as needed.



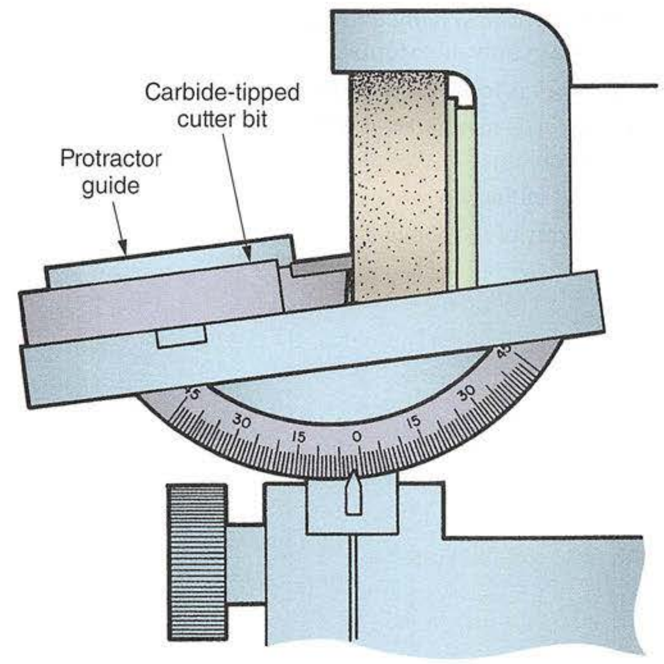
Goodheart-Willcox Publisher

Figure 13-14. A slight crown on the wheel face minimizes the amount of contact between the work and the wheel. The crown reduces the possibility of heat destroying the carbide tip of a tool.



Goodheart-Willcox Publisher

Figure 13-13. Use a hand vise to hold cutter bits while sharpening them.



Goodheart-Willcox Publisher

Figure 13-15. The tool rest can be adjusted to any desired angle using a table protractor and protractor guide.

Chapter Review

Summary

- Offhand grinding can be performed using an abrasive belt, bench, pedestal, or portable hand grinder.
- For safe operation, grinding wheels must be properly dressed and trued before use.
- A ring test helps determine whether a wheel is safe to use.
- Loading and glazing cause a grinding wheel to cut inefficiently.
- Grinding can be a dangerous operation if not performed safely.
- Dry grinders can overheat the workpiece and can ruin the tool or cause discoloration if care is not taken to cool the workpiece.
- Wet grinders are used primarily to grind carbide cutting tools.

Review Questions

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

1. Describe the grinding operation.
2. Bench and pedestal grinders are used to do ______ grinding that does not require great accuracy.
3. 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.
4. Name the two classifications of bench and pedestal grinders. How do they differ?
5. How far away should the tool rest be positioned from the grinding wheel? Why is this done?
6. How can grinding wheel soundness be checked?
7. Grinding soft metals, such as aluminum, brass, or copper, can ______ the grinding wheel.
8. A(n) ______ grinding wheel has a shiny surface and cuts inefficiently.
9. Never mount a grinding wheel on a grinder without checking it for ______.
10. What should you do with wheels that are not sound or that have a worn center hole?
11. List four safety precautions to be observed when operating a grinder.
12. Work may ______ or discolor if it is forced against the wheel with too much pressure.
13. Carbide-tipped tools are usually sharpened on a(n) ______ grinder.
14. When grinding carbide-tipped tools, the face of the wheel on a wet-type grinder should be slightly ______.

CHAPTER 14

The Lathe

Chapter Outline

Learning Objectives

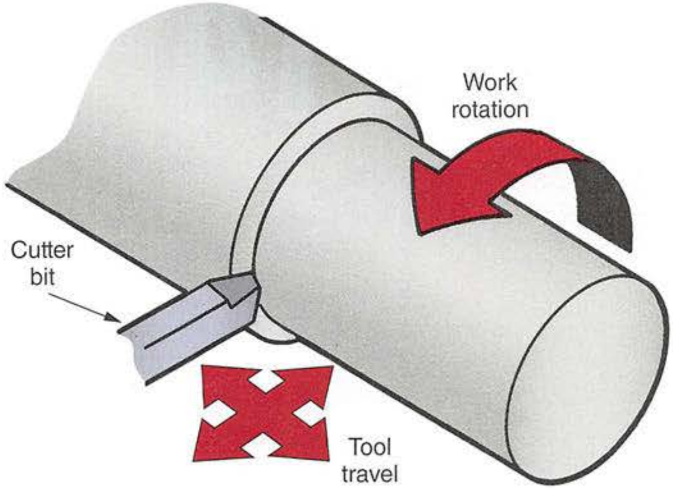
After studying this chapter, you will be able to:

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

Technical Terms

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

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



Goodheart-Willcox Publisher

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

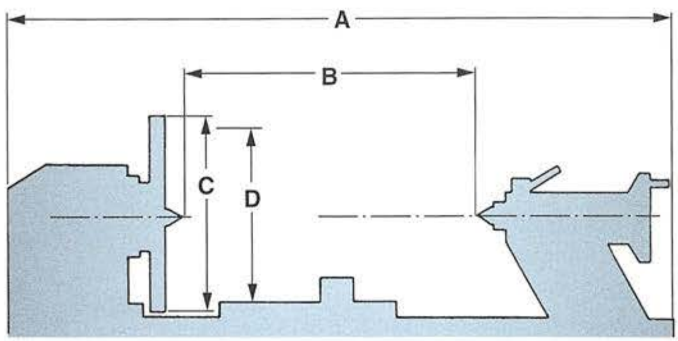
14.1 Lathe Size

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

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

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

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



Goodheart-Willcox Publisher

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

Jet Equipment & Tools

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

14.2 Major Parts of a Lathe

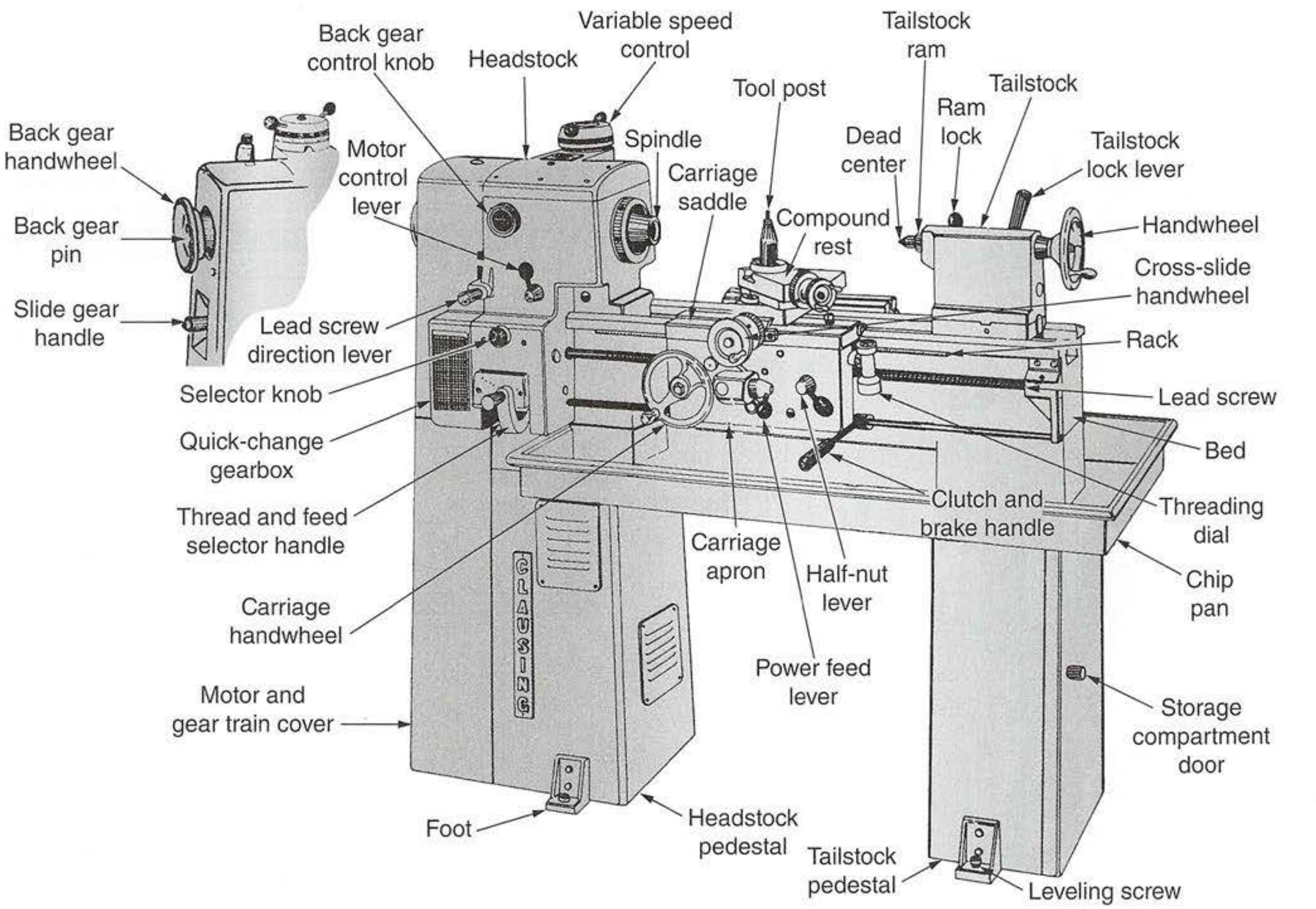
The chief function of any lathe, no matter how complex it may appear to be, is to rotate the work against a controllable cutting tool. Each of the lathe parts in **Figure 14-4** can be assigned to one of the following three functions:

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

14.2.1 Driving the Lathe

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

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



Clausing Industrial, Inc.

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

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

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

14.2.2 Holding and Rotating the Work

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

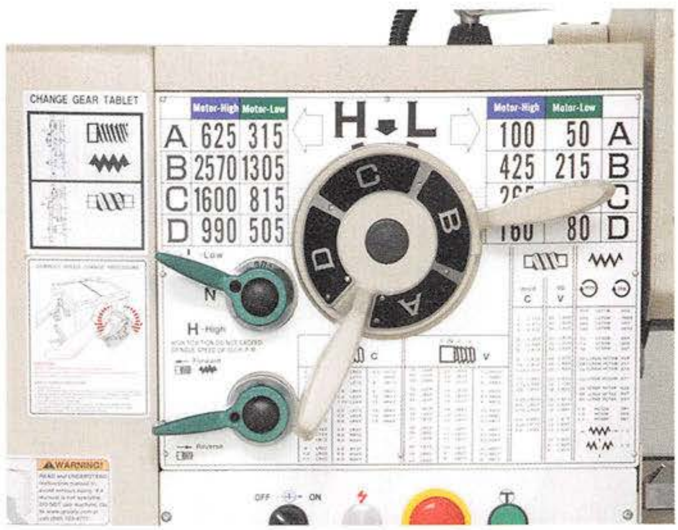
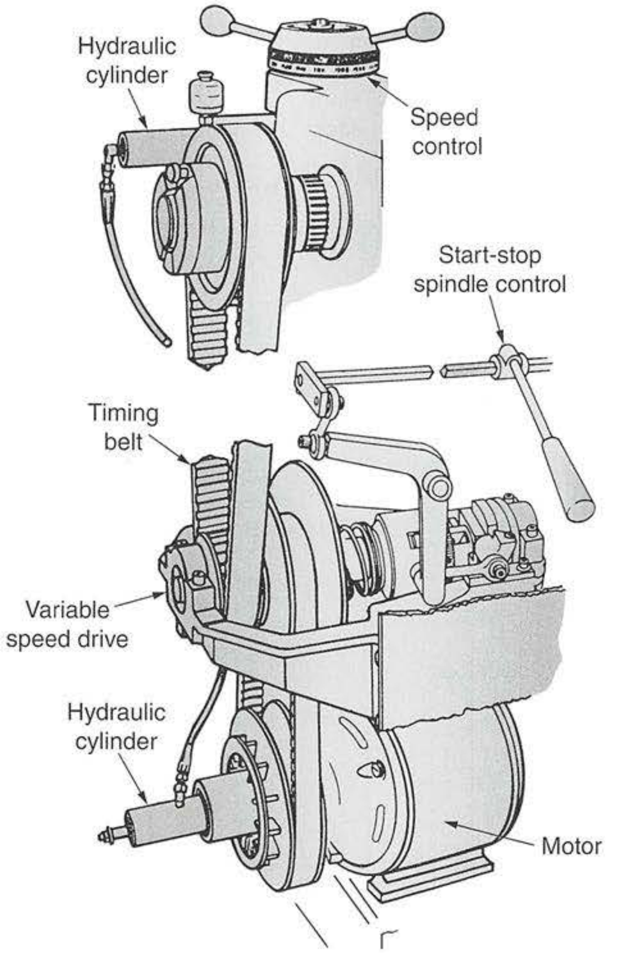


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

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

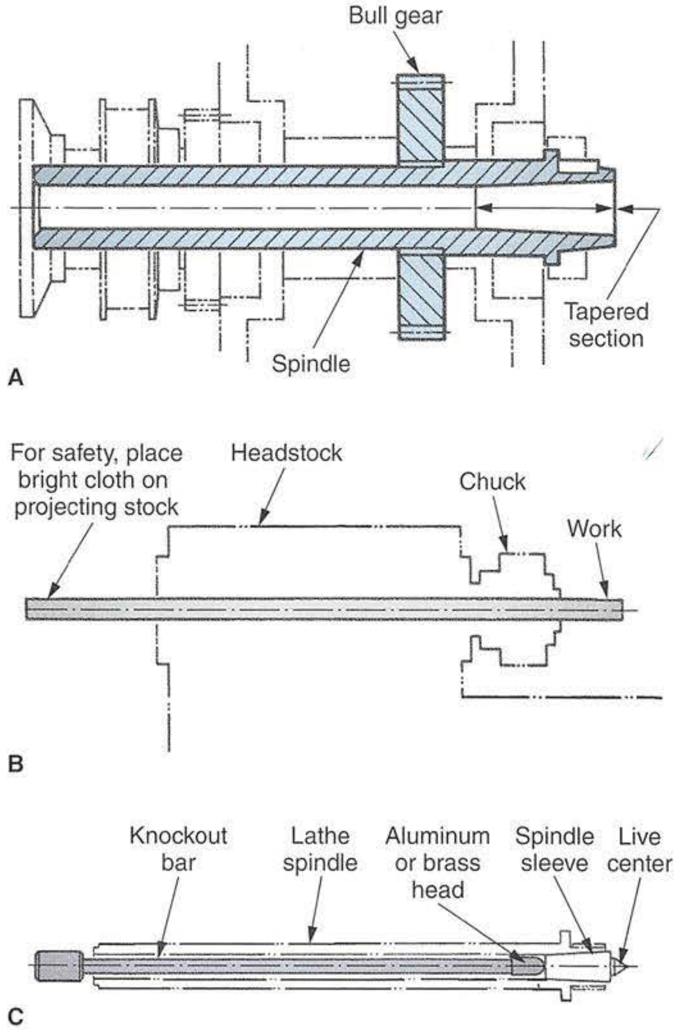


Clausing Industrial, Inc.

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

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

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



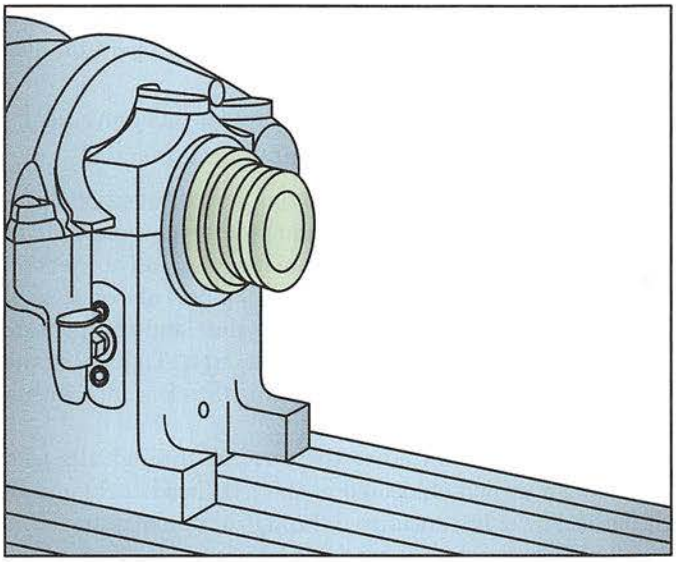
Goodheart-Willcox Publisher

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

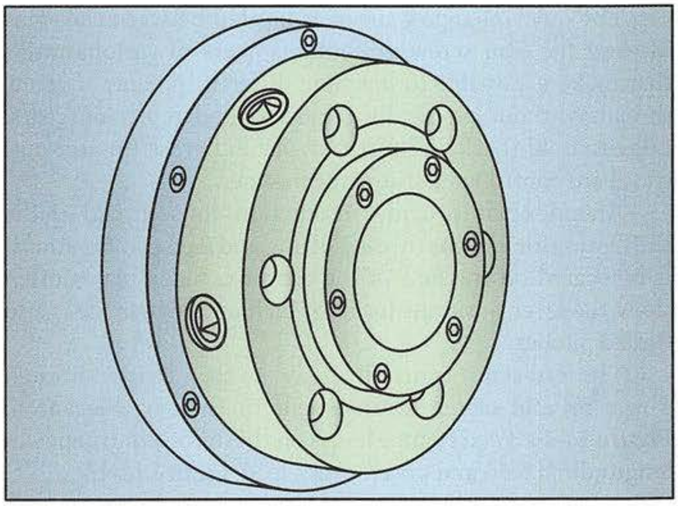
On the front end, a spindle may be threaded externally or fitted with a tapered spindle nose to receive work-holding attachments. See **Figure 14-9**. A threaded spindle nose is seldom used on modern lathes. It permits mounting an attachment by screwing it directly onto the threads until it seats on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

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

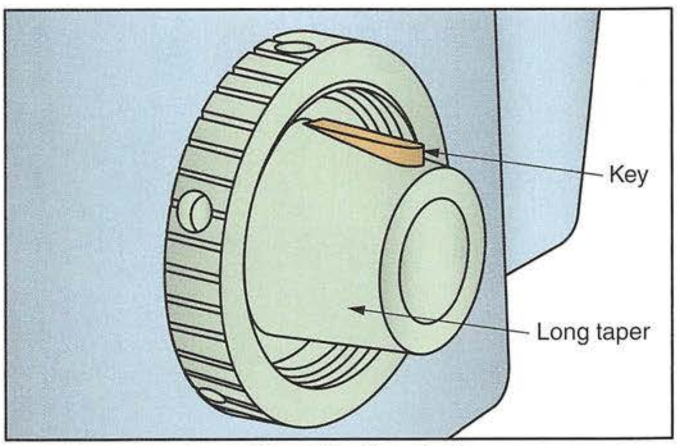
A long taper key spindle nose has a protruding long taper and key that fit into a corresponding taper and keyway in the back of the work-holding device. To mount a work-holding device (a chuck or faceplate), the spindle is rotated



Threaded Spindle Nose



Cam-Lock Spindle Nose



Long Taper Key Spindle Nose

Goodheart-Willcox Publisher

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

until the key is on top. The keyway in the back of the work-holding device is slid over the key to support the device until the threaded spindle collar can be engaged with the threaded section of the device and tightened.

SAFETY NOTE

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

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

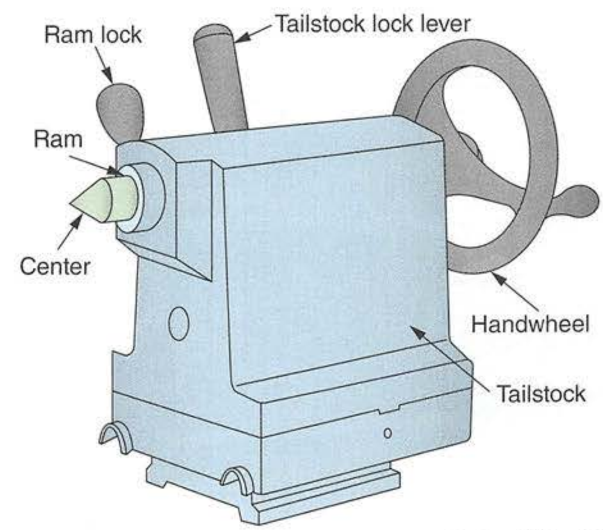
The outer end of the work is often supported by the lathe's ***tailstock***, **Figure 14-10**. The tailstock can be adjusted along the ways to accommodate different lengths of work.

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

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

14.2.3 Holding, Moving, and Guiding the Cutting Tool

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

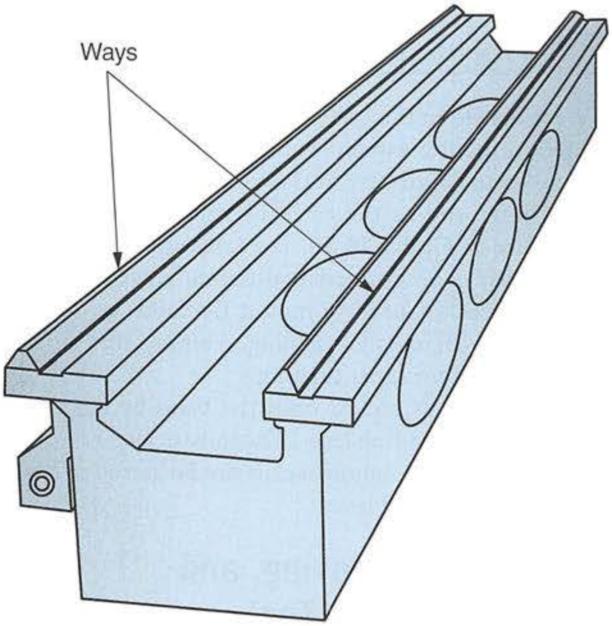


Goodheart-Willcox Publisher

Figure 14-10. Parts of a tailstock.

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

- The saddle is fitted to the ways and slides along them.
- The apron contains a drive mechanism to move the carriage along the ways, using hand or power feed.



Goodheart-Willcox Publisher

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

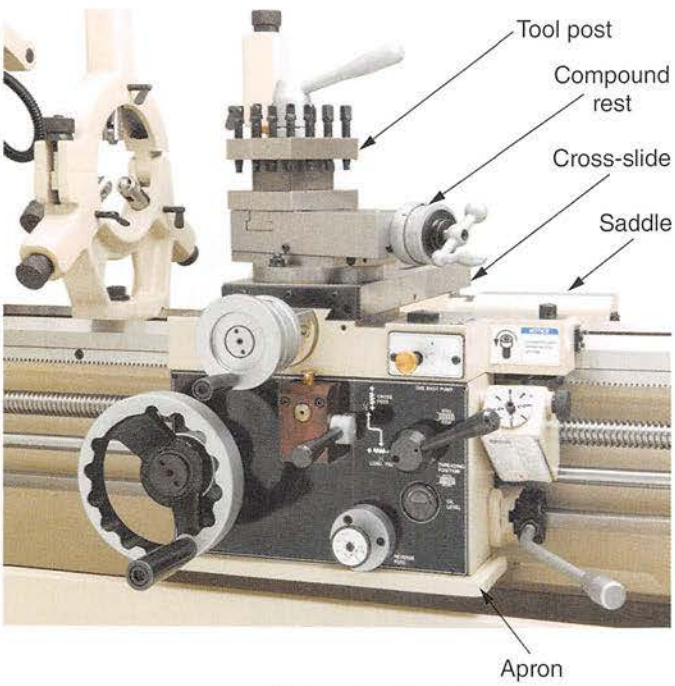


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

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

- The ***cross-slide*** permits transverse tool movement (movement toward or away from the operator, at a right angle to the axis of the lathe).
- The ***compound rest*** permits angular tool movement.
- The ***tool post*** is used to mount the cutting tool.

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

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

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

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

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

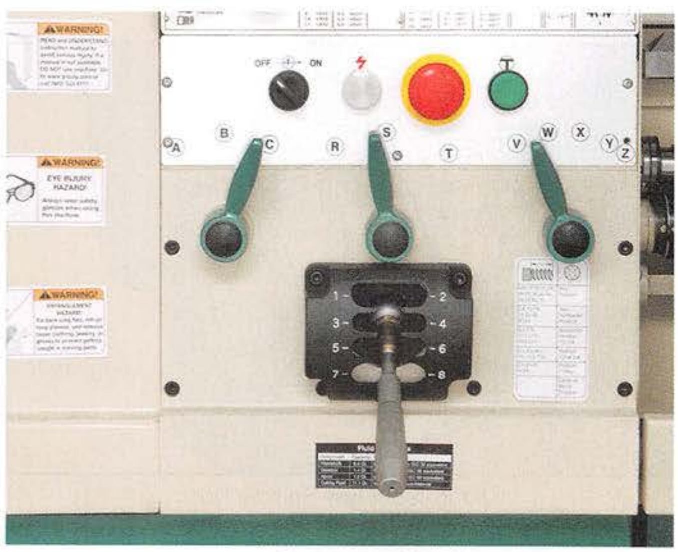
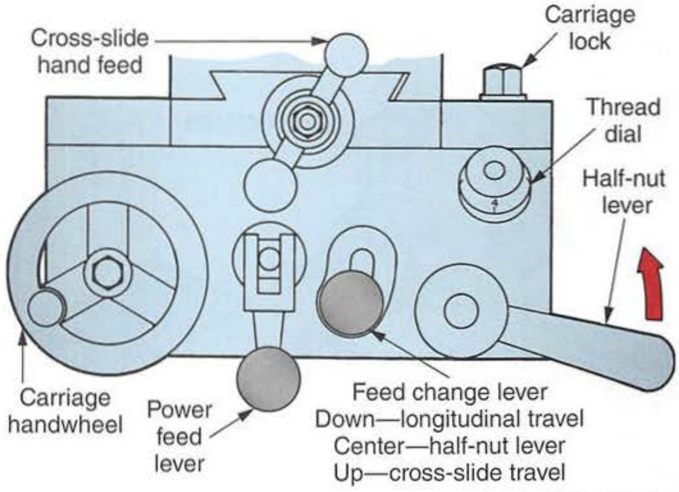


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

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

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

Figure 14-14. The lead screw.



Goodheart-Willcox Publisher

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

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

14.3 Work-Holding Attachments

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

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

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

14.3.1 Work-Holding between Centers

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

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

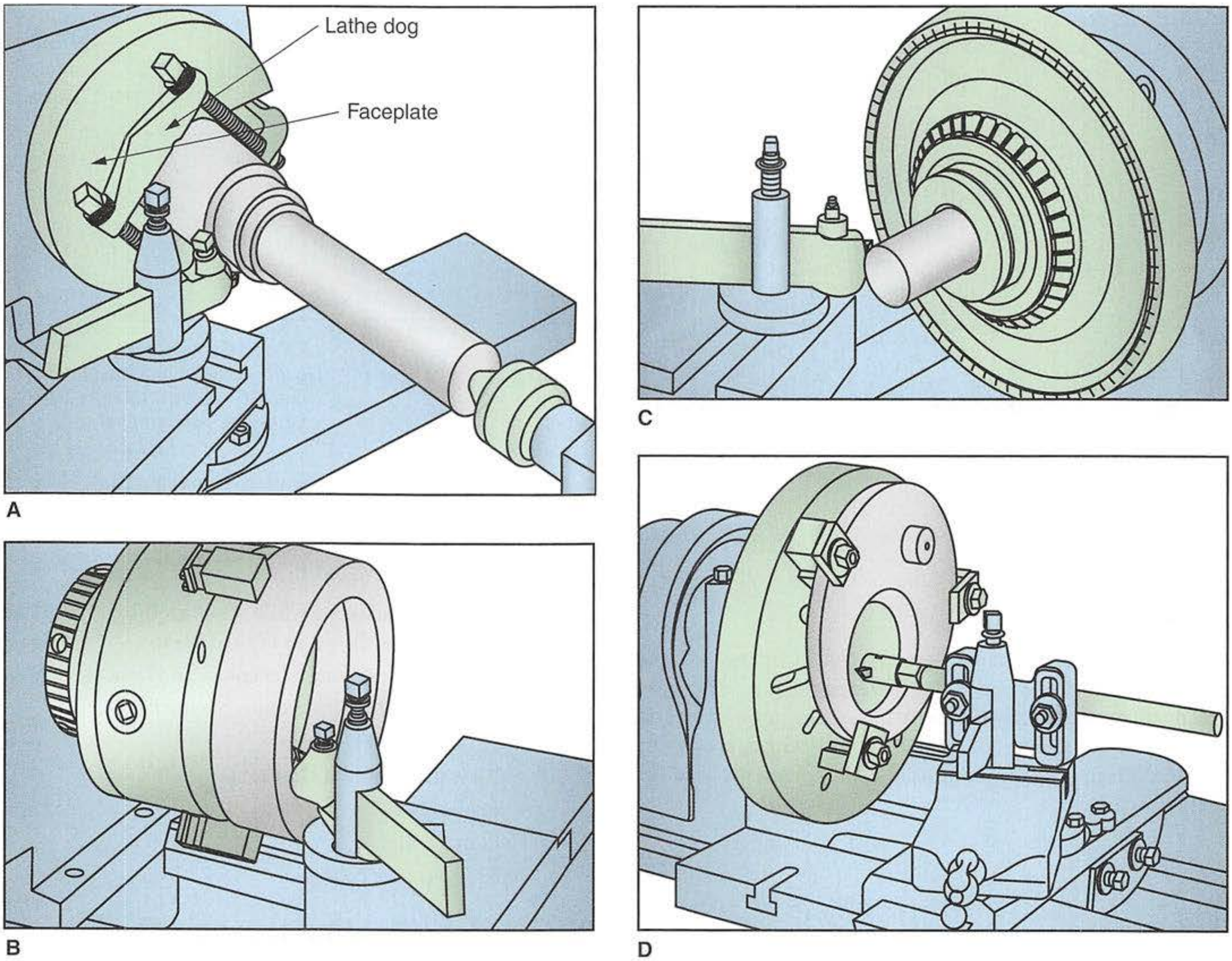
WORKPLACE SKILLS

Attitude on the Job

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

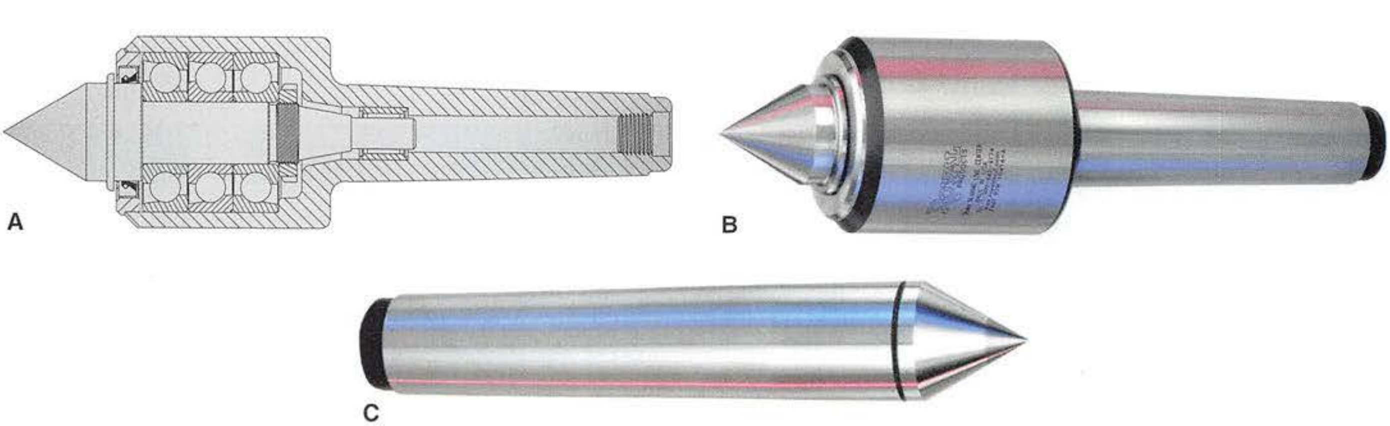
Enthusiasm spreads easily from one person to another. Usually, *enthusiasm* means a person enjoys what he or she is doing. In an office setting, enthusiasm builds a team spirit for working together.

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



Goodheart-Willcox Publisher

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



Royal Products, Division of Curran Manufacturing Corporation

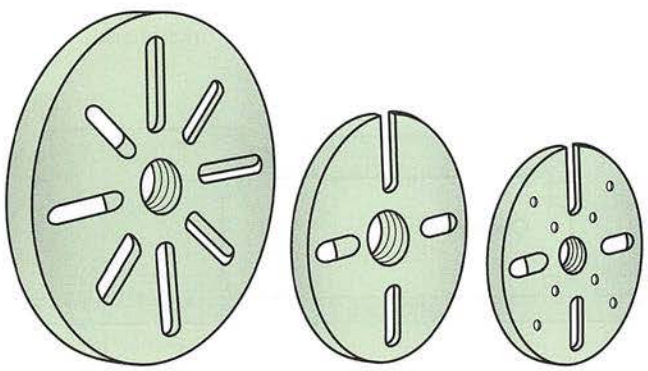
Figure 14-17. Lathe centers. A—Sectional view shows construction of a heavy-duty ball bearing live center. B—High-precision quad-bearing live center. C—A dead center does not rotate. The carbide tip provides great wear resistance.

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

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

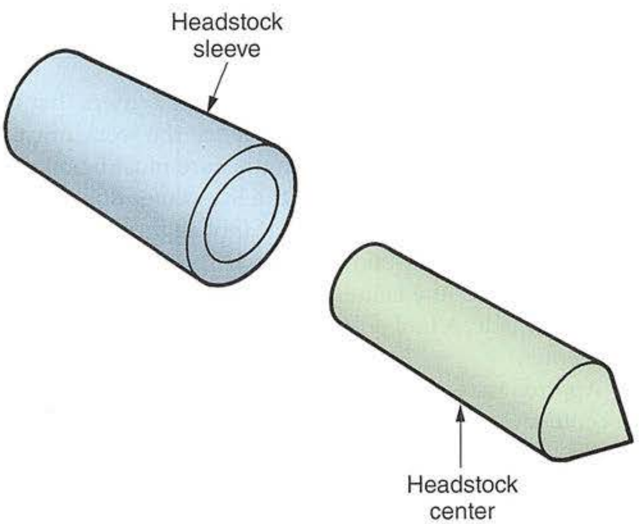
A ***lathe dog*** is a device clamped to one end of the material to drive the workpiece. The three types of lathe dogs, as shown in **Figure 14-20**, are as follows:

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



Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

Figure 14-19. Sleeve and headstock center.

Goodheart-Willcox Publisher

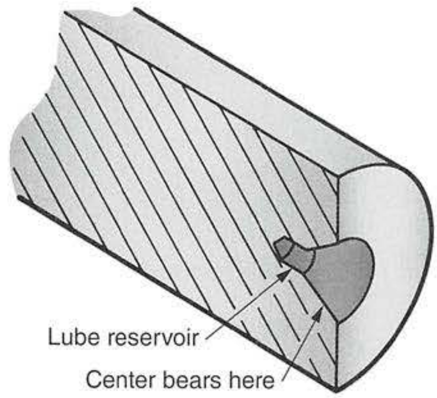
Figure 14-20. Lathe dogs.

Drilling Center Holes

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

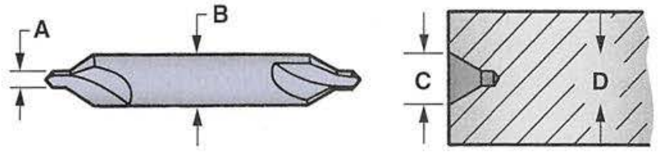
Center holes are usually drilled with a combination drill and countersink, **Figure 14-23**. The drill angle is identical to that of the center point. The straight drill provides clearance for the center point and serves as a reservoir for a lubricant. The chart provides the information needed to select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

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



Goodheart-Willcox Publisher

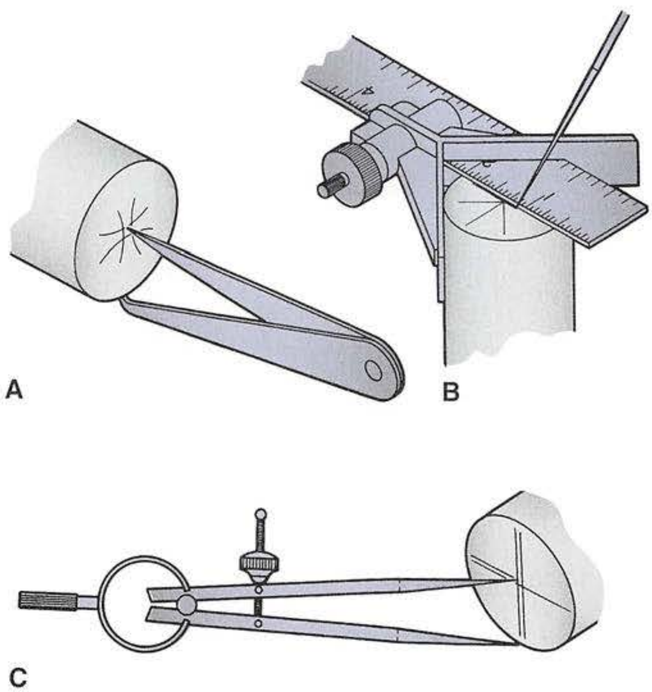
Figure 14-21. The tailstock center rides in the drilled and countersunk center hole. If a dead center is used, a supply of lubricant is placed in the reservoir. The lubricant will expand and lubricate the center as the metals heat up.



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

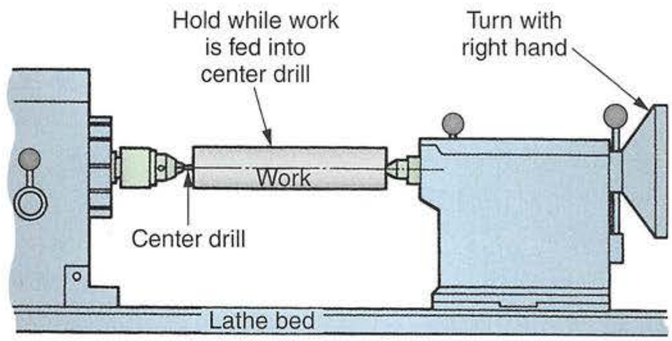
Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

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

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

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

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

Checking Center Alignment

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

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

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

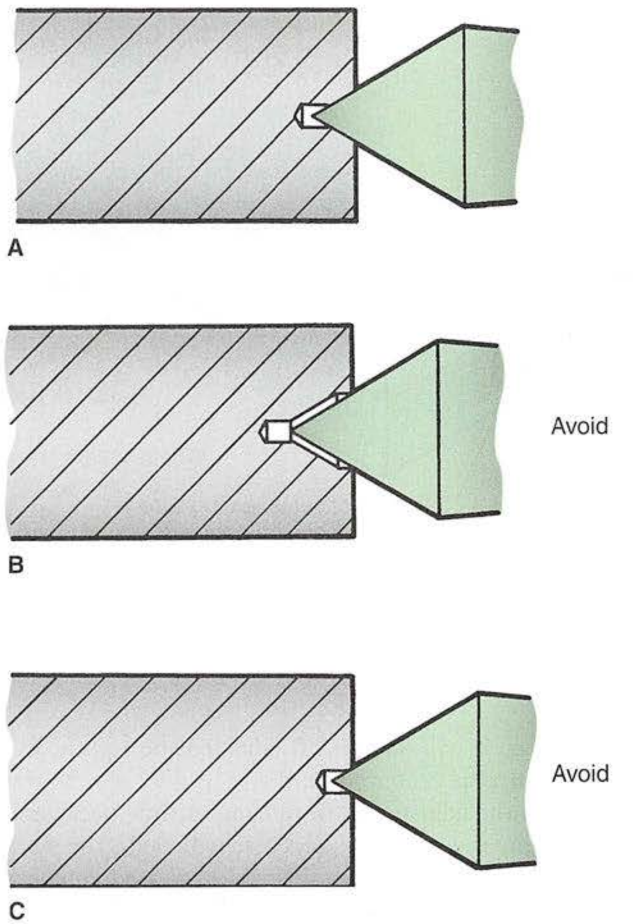


Figure 14-25. Correctly and incorrectly drilled center holes. A—Properly drilled center hole. B—Hole drilled too deep. C—Hole not drilled deep enough. Does not provide enough support. If used with a dead center, the center point will burn off.

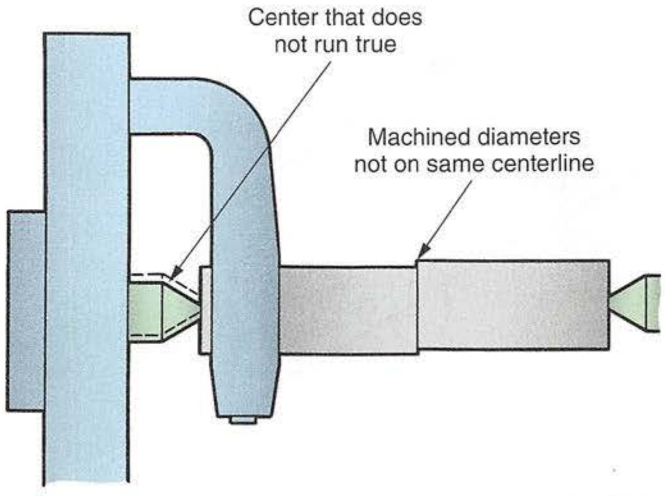


Figure 14-26. The workpiece must be reversed in the lathe dog so it can be machined for its entire length. If the live center does not run true, eccentric diameters will result.

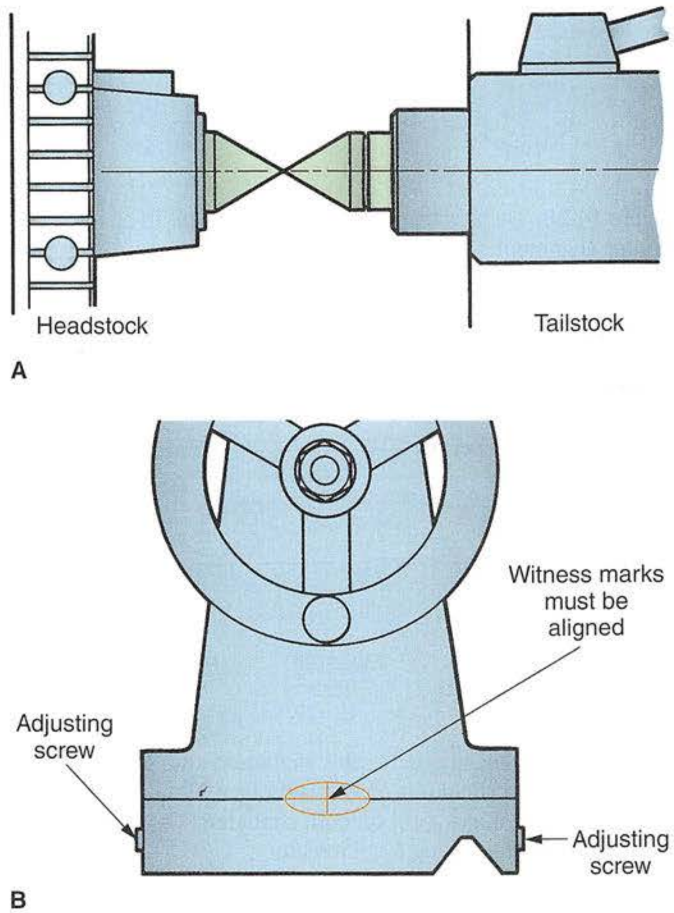
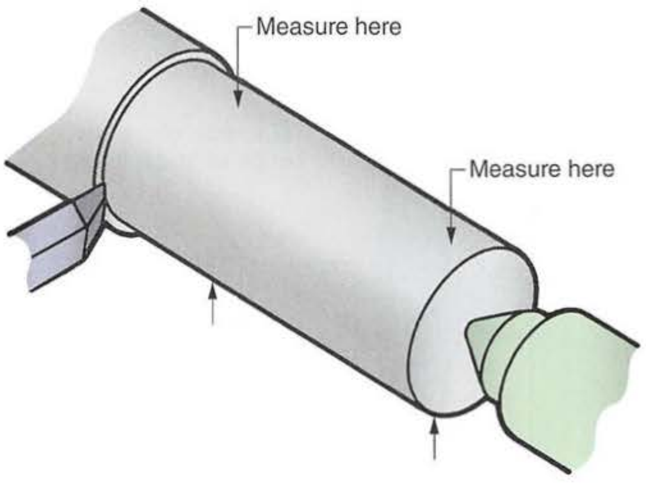


Figure 14-27. Checking center alignment. A—Checking alignment by bringing center points together. This view is looking down on the top of the centers. B—Alignment can be determined by checking witness lines on the base of the tailstock.

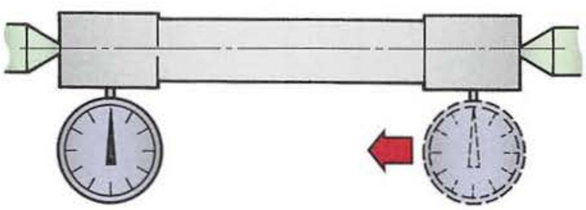
- Make a light trial cut across a few inches of the material. Check the diameter at each end with a micrometer, **Figure 14-28**. The centers are aligned if the readings are identical.
- Use a steel test bar and dial indicator, **Figure 14-29**. Mount the test bar between centers and position the dial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

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



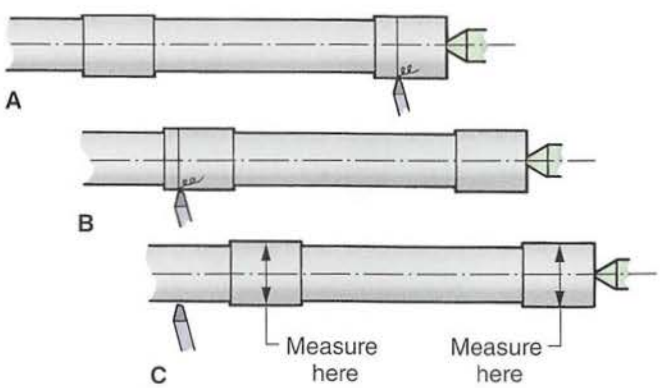
Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

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

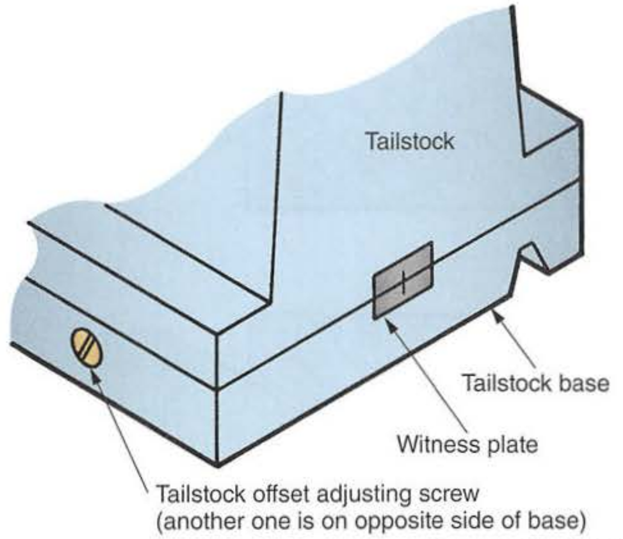


Goodheart-Willcox Publisher

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

Mounting Work between Centers

Clamp a dog to one end of the work. Place a lubricant (graphite and oil or a commercial center lubricant) in the center hole on the other end. Mount the piece on the centers and adjust the tailstock spindle until the work is snug. If the work



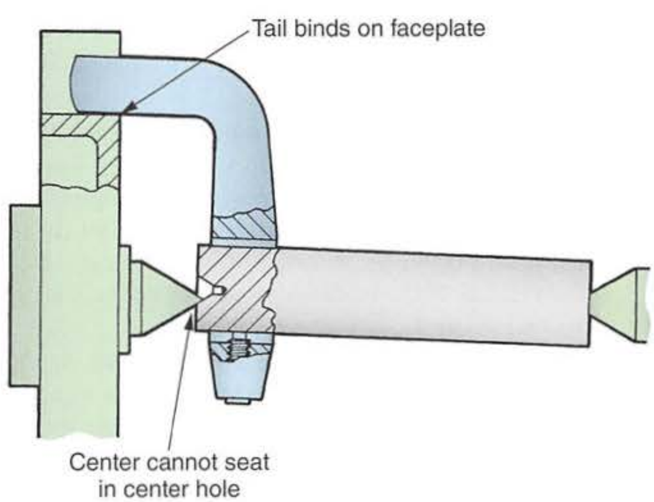
Goodheart-Willcox Publisher

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

is too loose, it will "clatter." If adjusted too tightly, it will score or burn the center point.

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

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



Goodheart-Willcox Publisher

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

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

14.3.2 Using Lathe Chucks

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

Three-Jaw Universal Chuck

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

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

Installing Chuck Jaws

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

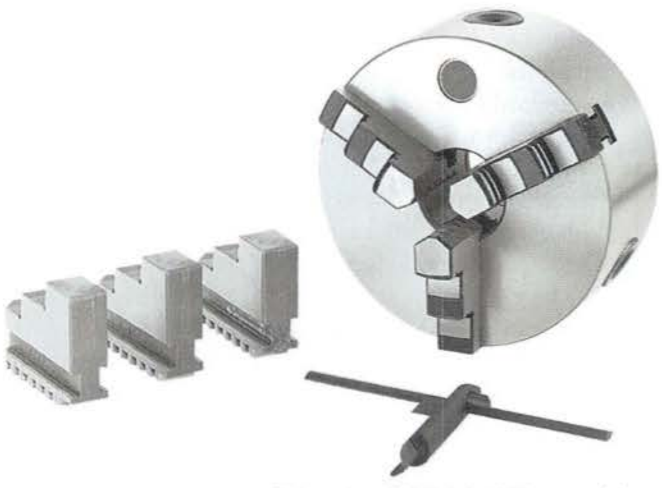
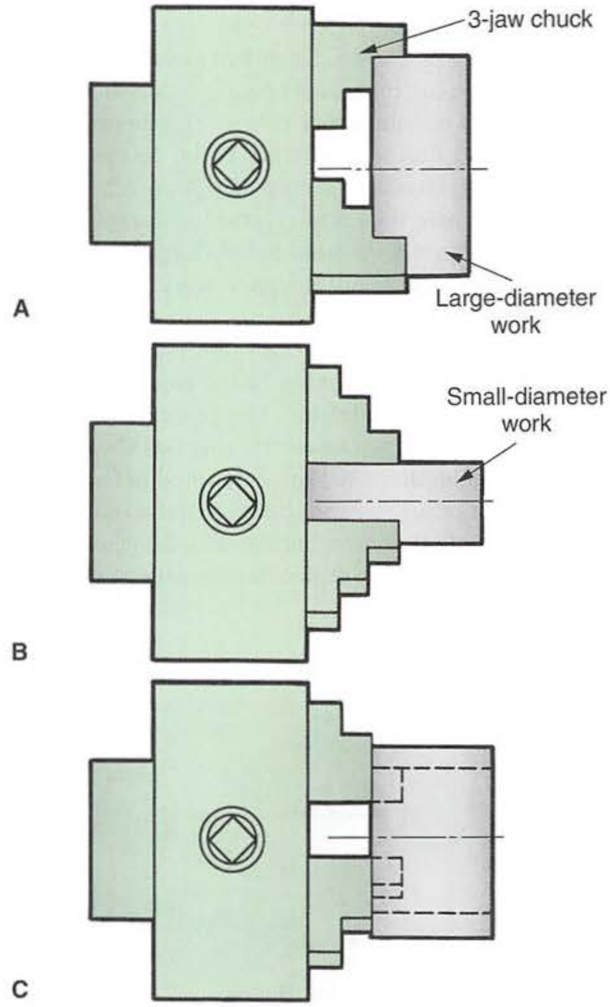


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

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



Goodheart-Willcox Publisher

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

SAFETY NOTE

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

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

SAFETY NOTE

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

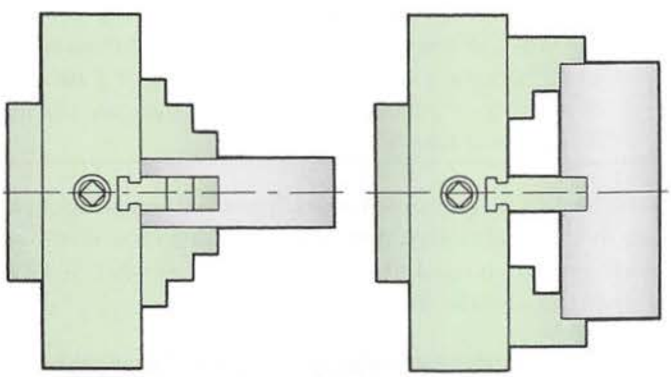
Four-Jaw Independent Chuck

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

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

AMT—The Association for Manufacturing Technology

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



Goodheart-Willcox Publisher

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

Goodheart-Willcox Publisher

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

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

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

Jacobs Chuck

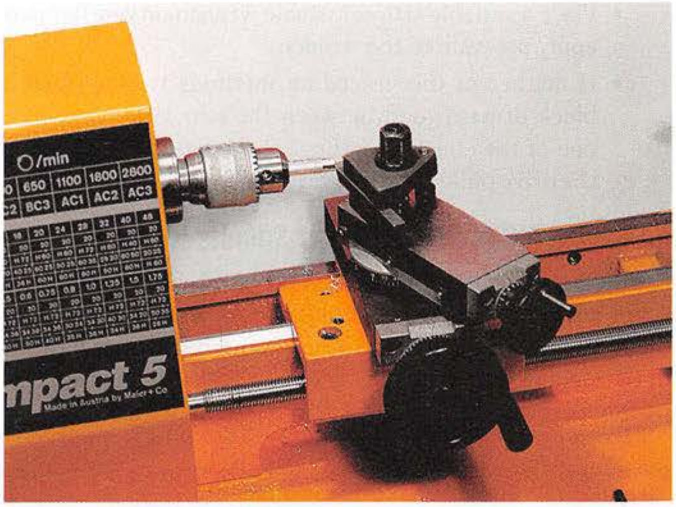
When turning small-diameter work, such as screws or pins, the Jacobs chuck can be used. This chuck, **Figure 14-38**, is better suited for such work than the larger universal or independent chuck.

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

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

Draw-In Collet Chuck

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

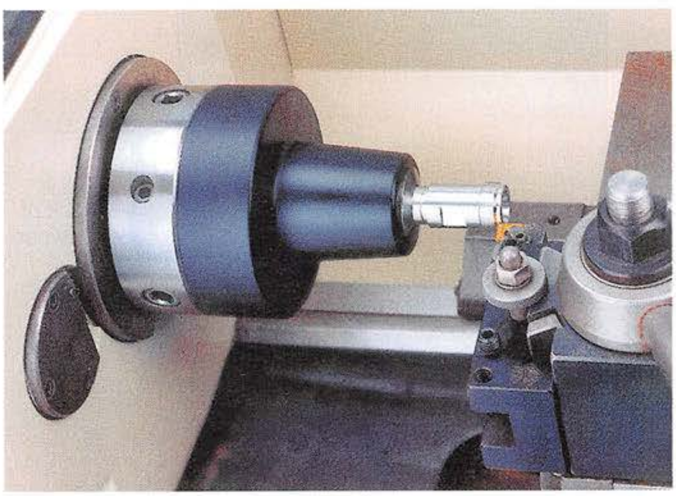


Goodheart-Willcox Publisher

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

Royal Products, Division of Curran Manufacturing Corporation

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



Royal Products, Division of Curran Manufacturing Corporation

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

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

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

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

Mounting Chucks

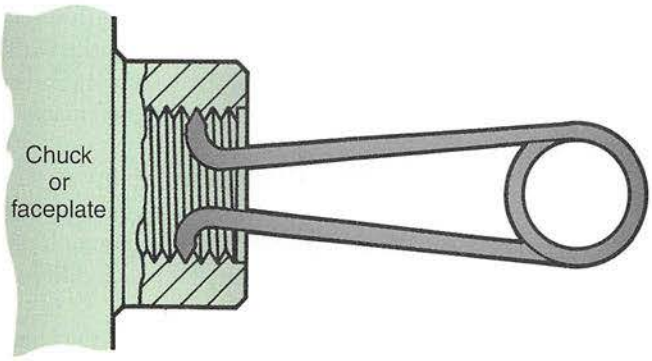
If a chuck is not installed on the spindle nose correctly, its accuracy is affected. To install a chuck, remove the center and sleeve, if they are in place. Hold the center and sleeve with one hand and tap them loose with a knockout bar. Carefully wipe the spindle end clean of chips and dirt. Apply a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

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

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

Maswerks, Inc.

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



Goodheart-Willcox Publisher

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

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

Removing a Chuck from Threaded Spindle

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

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

Removing a Chuck from Other Spindle Noses

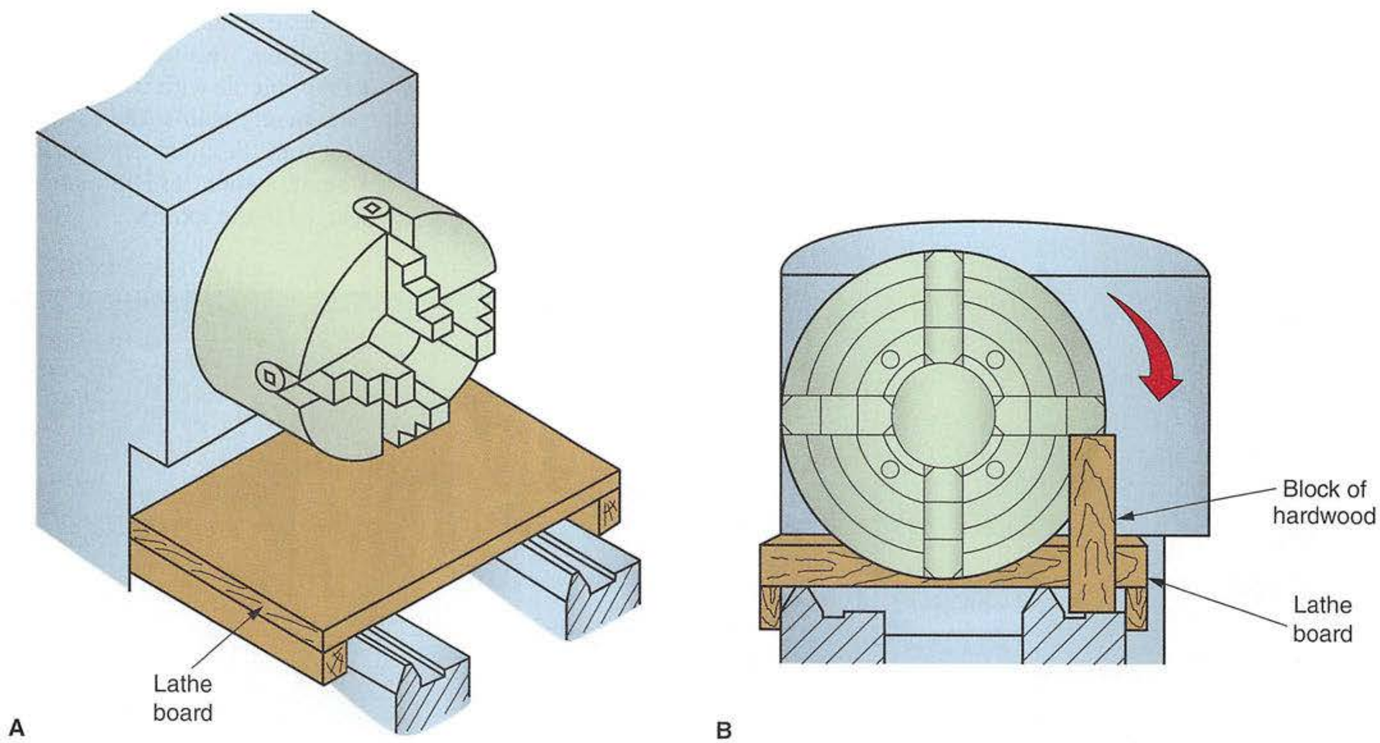
Removing a chuck from tapered and cam-lock spindle noses is not difficult. For tapered spindle noses, first lock the spindle in back gear, then place the appropriate spanner wrench in the locking ring. Give it a tap or two with a leather or plastic mallet. Turn the ring until the chuck is released.

SAFETY NOTE

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

14.4 Cutting Tools and Toolholders

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



Goodheart-Willcox Publisher

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

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

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

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

14.4.1 High-Speed Steel Cutting Tool Shapes

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

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

14.4.2 Types of Cutting Tools

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

Roughing Tools

The deep cuts made to remove considerable material from a workpiece are called ***roughing cuts***. Roughing tools have a tool shape (shape of cutting tip) that consists of a straight cutting edge with a small, rounded nose. This shape permits deep cuts at heavy 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, from right to left. See **Figure 14-47A**.

Finishing Tools

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

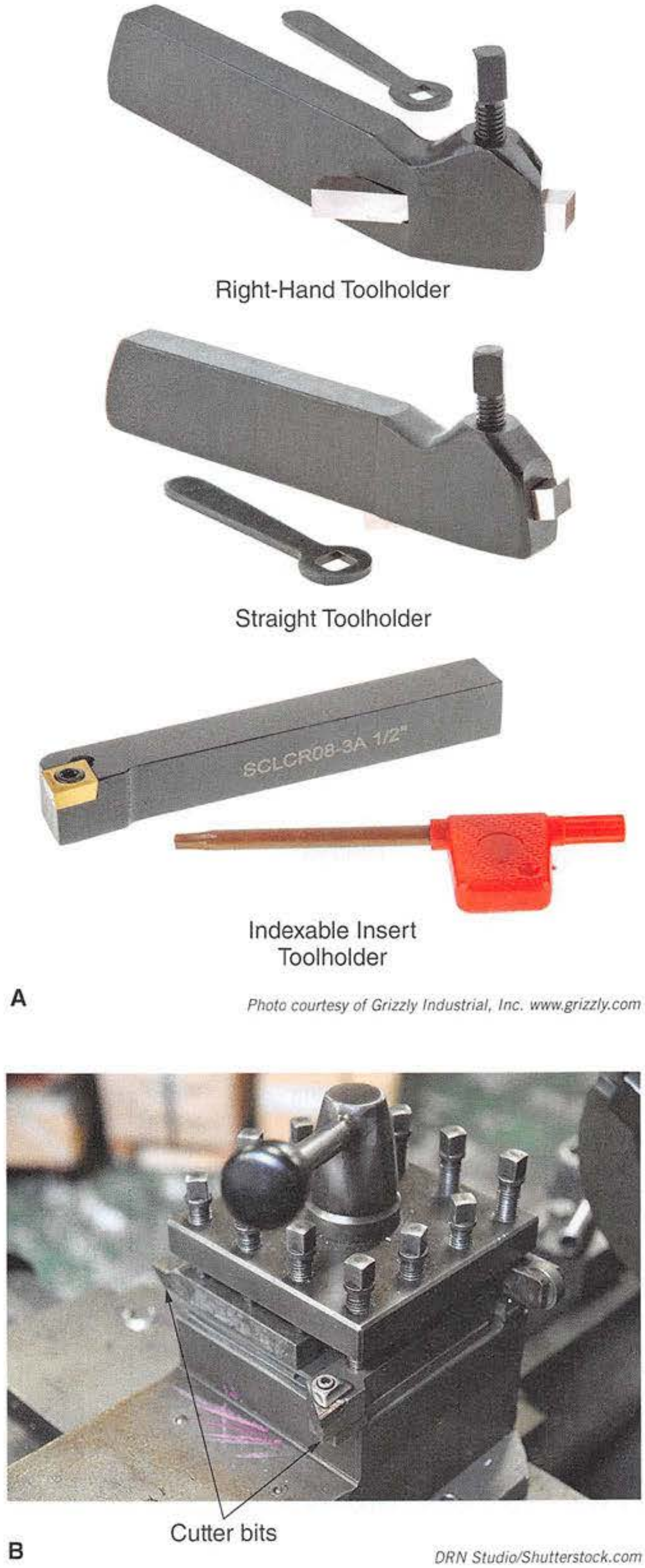


Figure 14-44. A—Examples of toolholders. B—A turret holder with four cutter bits.

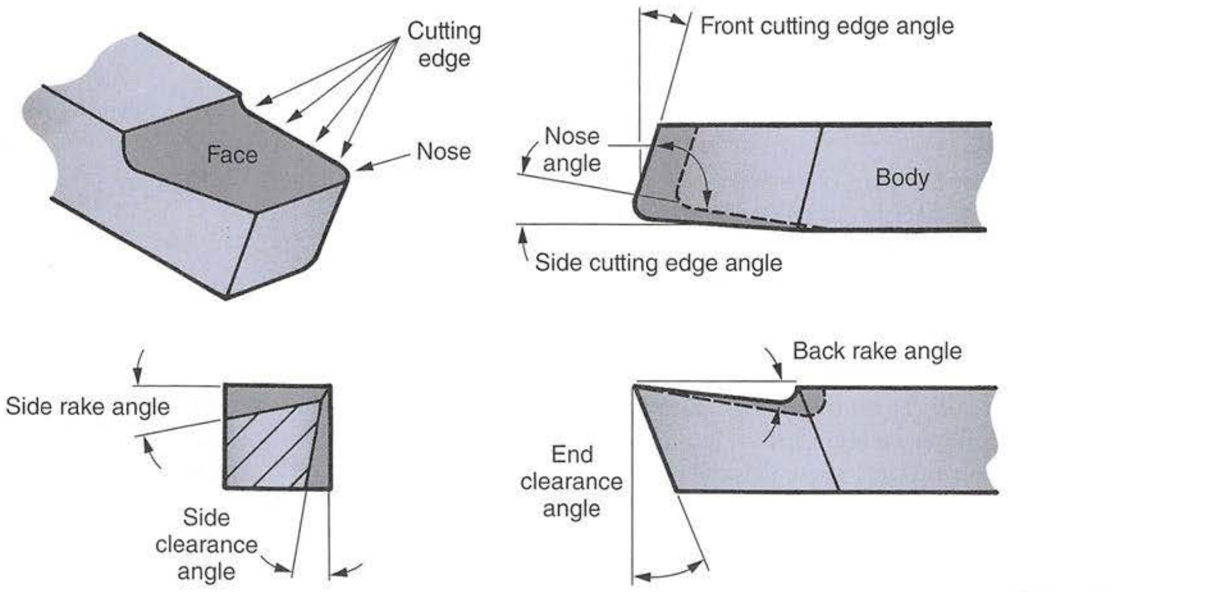


Figure 14-45. Cutter bit nomenclature.

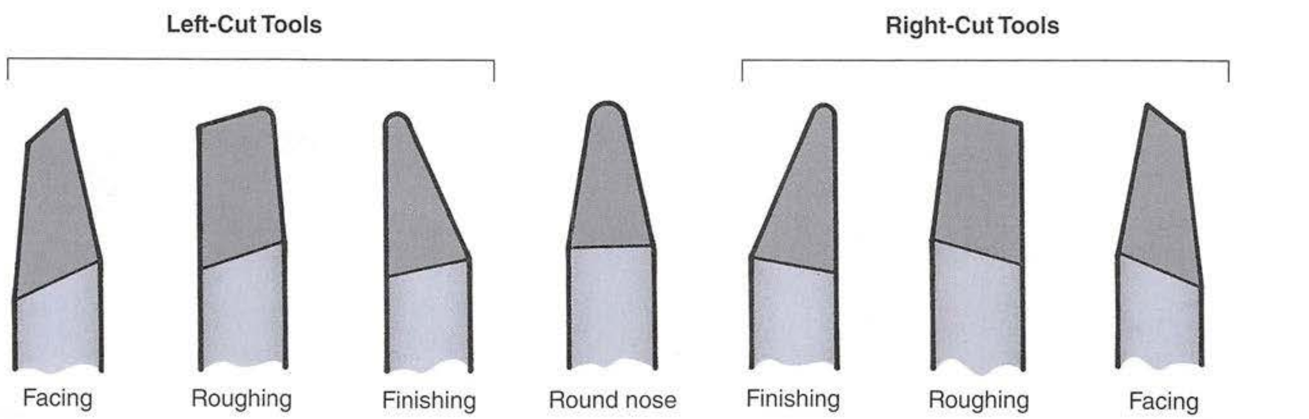


Figure 14-46. Standard HSS cutting tool shapes.

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

Facing Tool

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

Round-Nose Tool

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

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

Brazed-Tip Single-Point Cutting Tools

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

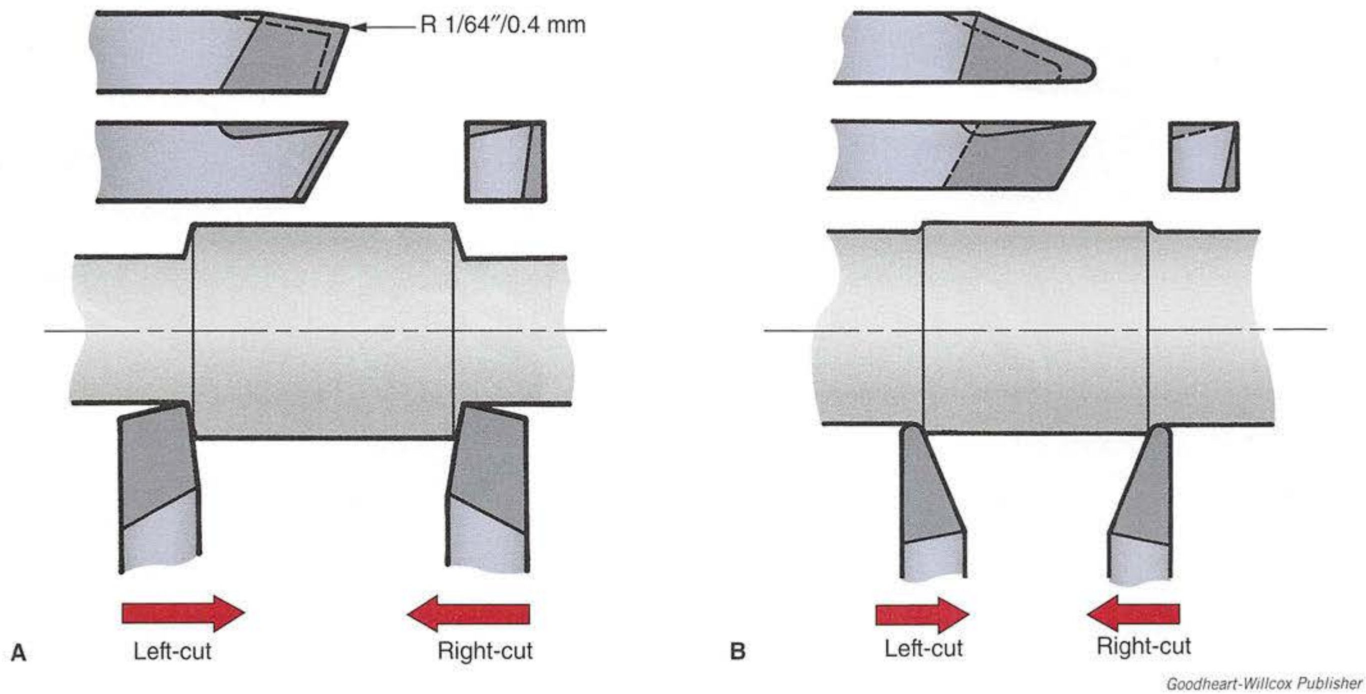


Figure 14-47. Lathe tools. A—The roughing tool is used for rapid material removal. B—The finishing tool produces a smooth surface.

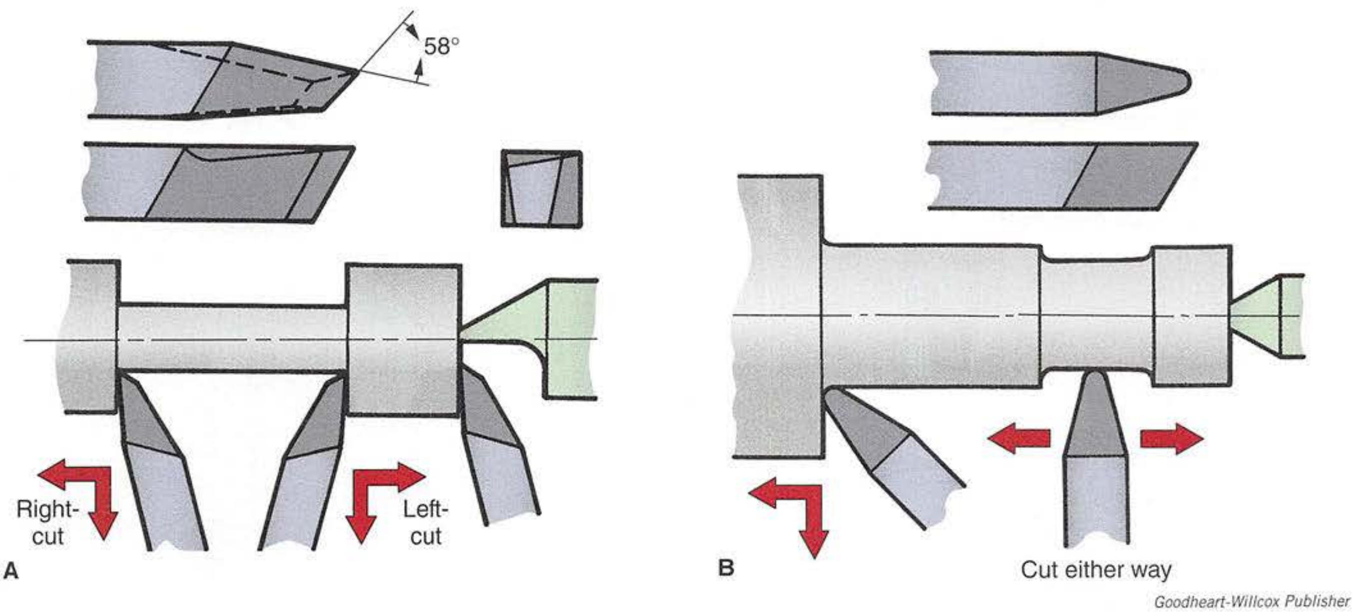
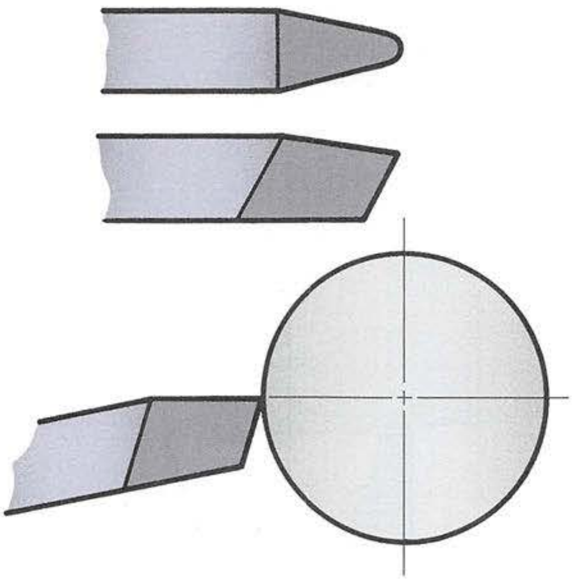


Figure 14-48. Lathe tools. A—A facing tool is used to machine surfaces perpendicular to the spindle centerline. B—A round-nose tool produces fillets. Its shape permits it to cut either left or right.

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

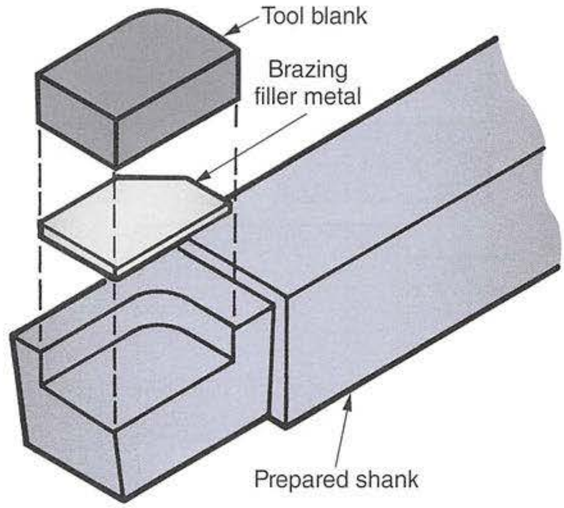
Cutting speeds can be increased by 300% to 400% when using carbide cutting tools. Powders of tungsten, carbon, and cobalt are molded into tool blanks and heated to extremely high temperatures. The hardness and strength of the blank can be controlled by varying the amount of cobalt that is used to cement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

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



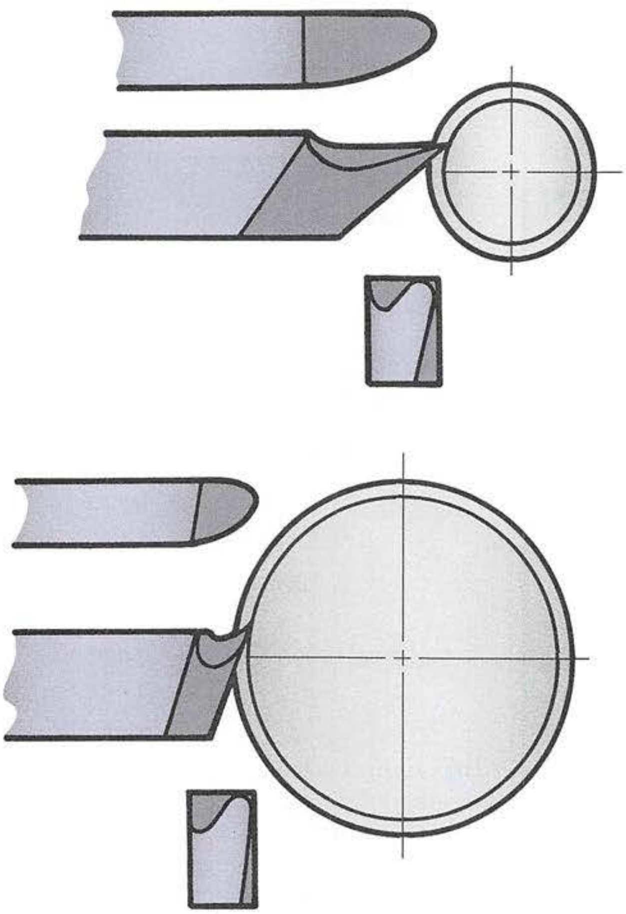
Goodheart-Willcox Publisher

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



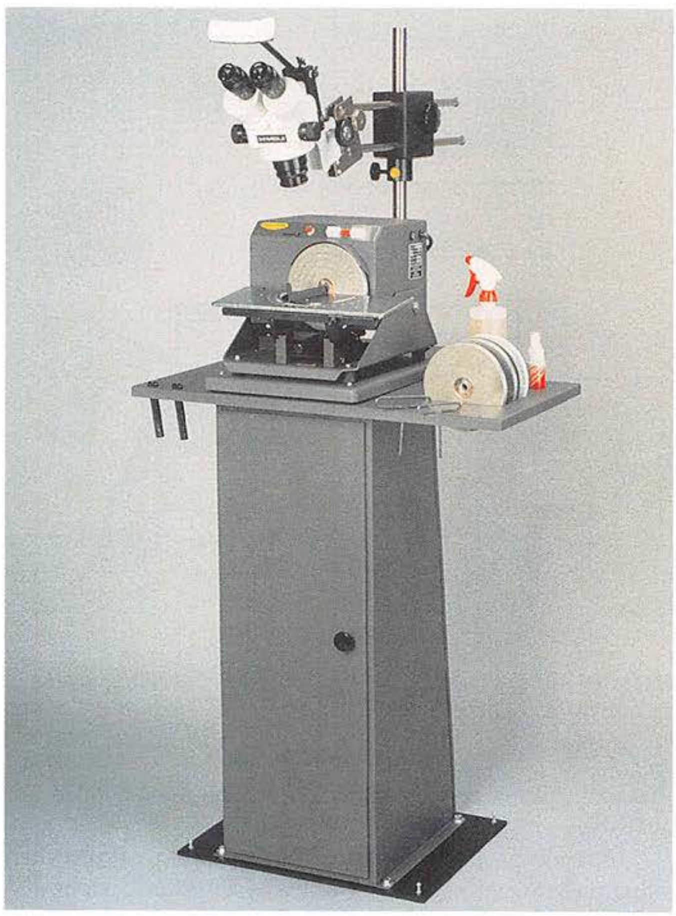
Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

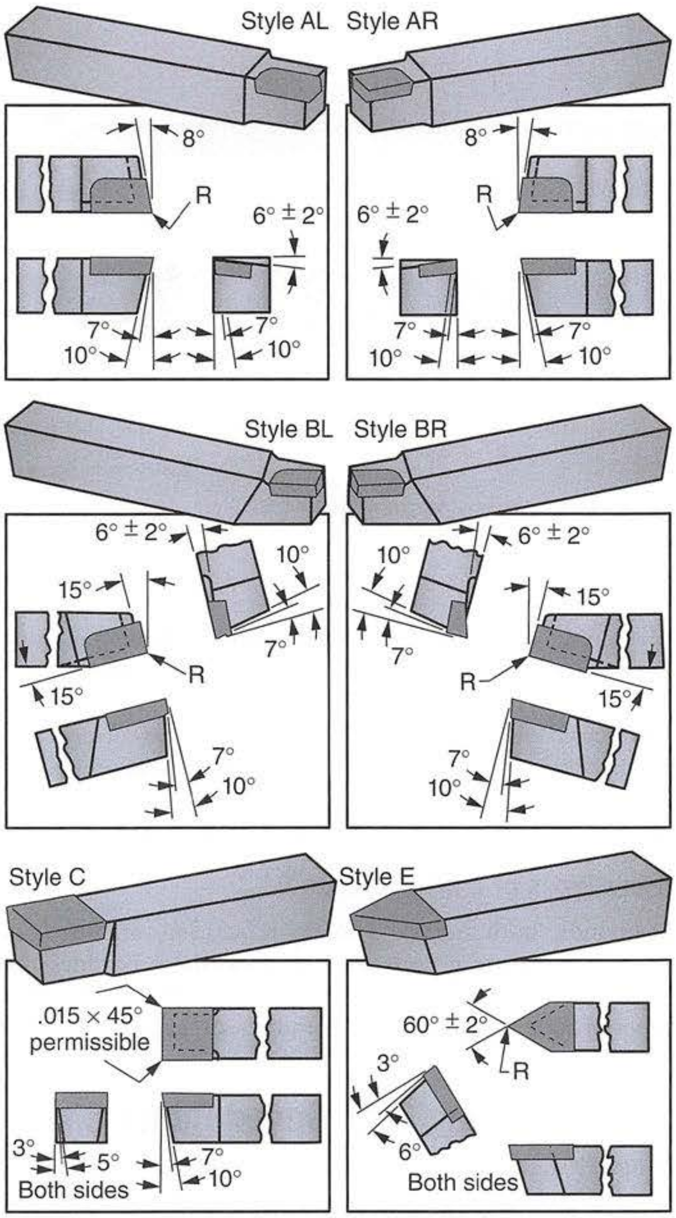
Figure 14-52. A grinder designed for sharpening carbide, cermet, cubic boron nitride, and polycrystalline diamond (PCD) cutting tools. It uses diamond-charged wheels and is fitted with a microscope inspection system.

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

Carbide-Tipped Straight Turning Tools

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

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



Carboloy

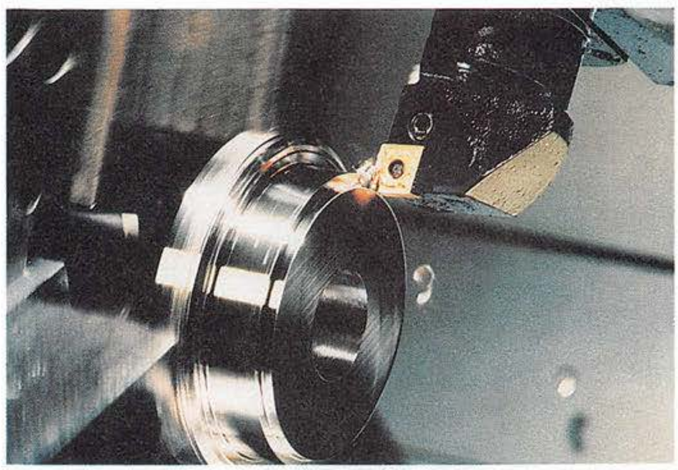
Figure 14-53. Typical standard cemented-carbide single-point tools. Style E is a carbide-tipped threading tool.

Indexable Insert Cutting Tools

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

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

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



Sandvik Coromant Co.

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

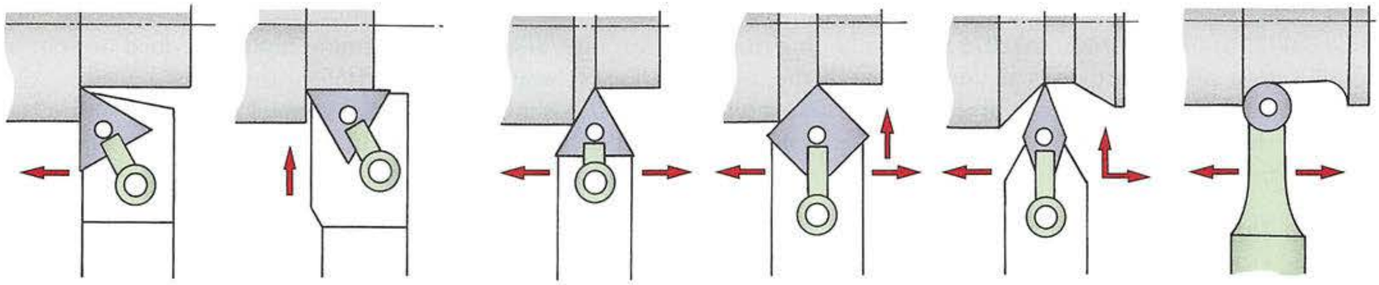


Figure 14-55. Indexable inserts are manufactured in a number of different shapes and sizes for different turning operations.

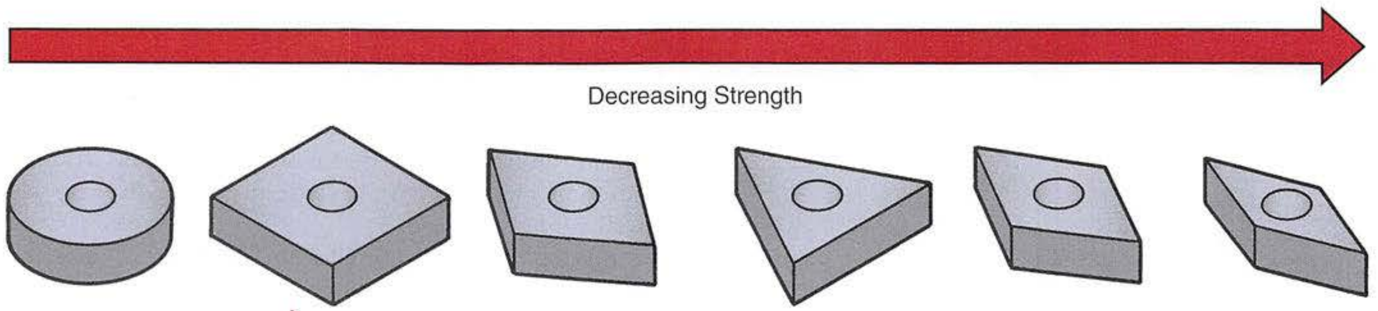


Figure 14-56. Most commonly used indexable insert shapes are shown in order of decreasing strength.


Carboloy

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

Chipbreakers

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

Other Types of Cutting Tools

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

14.4.3 Grinding High-Speed Cutter Bits

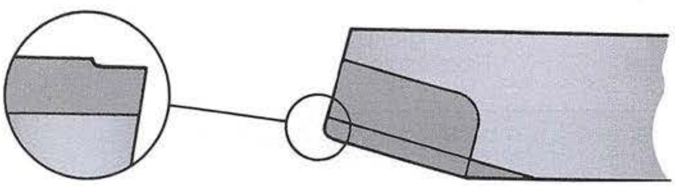
When first attempting to grind a cutter bit, it may be best if you first practice on sections of cold-finished steel square stock. You may also want to use chalk or bluing and draw the desired tool shape on the front portion of the blank. The lines will serve as guides for grinding.

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

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

Valenite, Inc.

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



Goodheart-Willcox Publisher

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

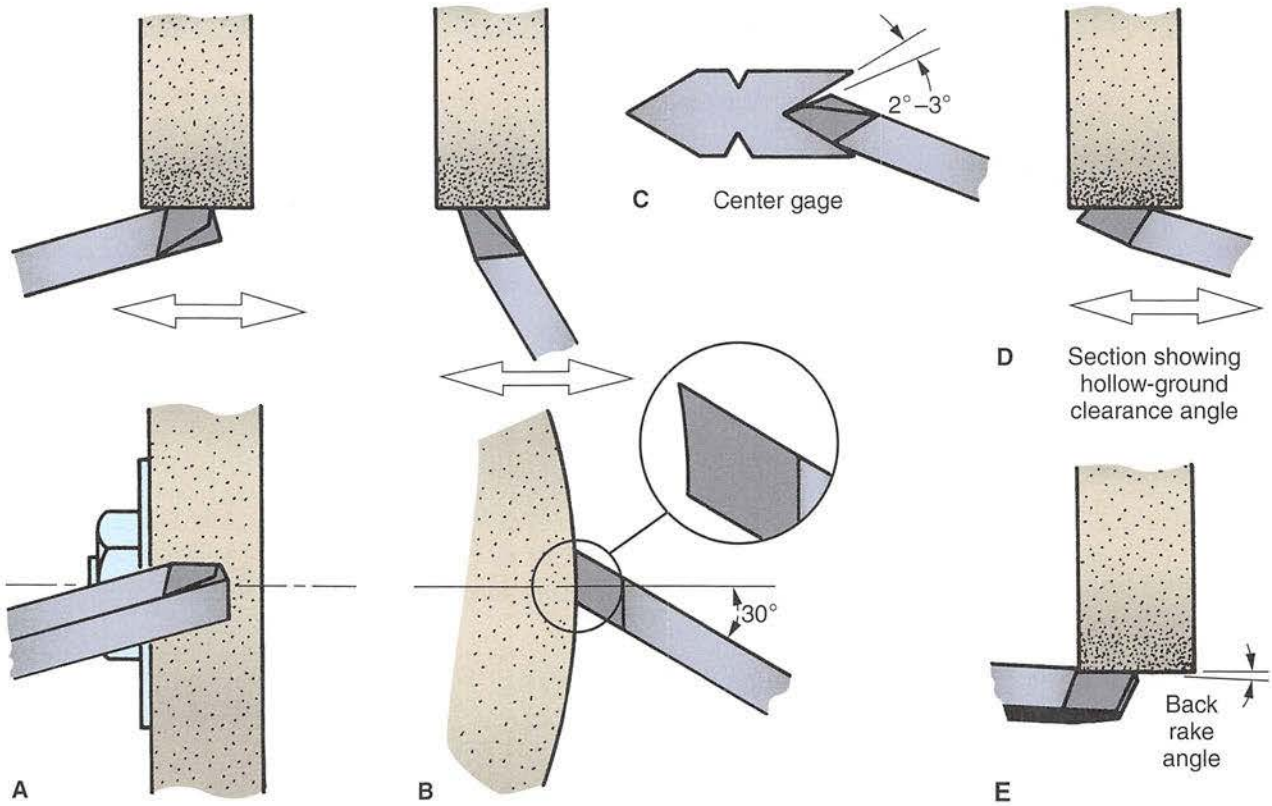
14.5 Cutting Speeds and Feeds

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

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

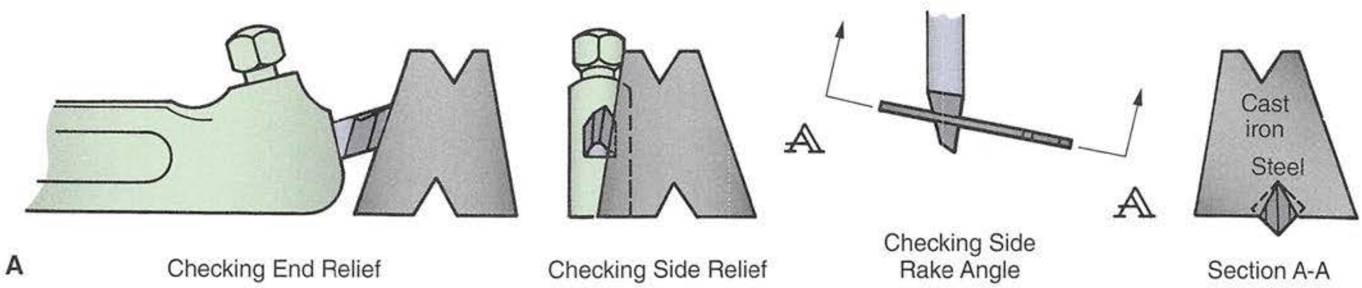
Feed is the distance that the cutter moves lengthwise along the lathe bed during a single revolution of the work. There are a number of factors that must be considered when determining the correct cutting speed and amount of feed, including the following:

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



Goodheart-Willcox Publisher

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



Rake and Clearance Angle for Lathe Tools (High-Speed Steel)

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

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

B

Goodheart-Willcox Publisher

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

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

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

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

14.5.1 Calculating Cutting Speeds

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

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

Goodheart-Willcox Publisher

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

Inch-based.

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

Where:

rpm = Revolutions per minute.
CS = Cutting in feet per minute.
D = Diameter of the work in inches.

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

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

D = 4

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

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

$$= 1000 \text{ rpm}$$

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

Metric-based.

$$\text{rpm} = \frac{\text{CS} \times 1000}{\text{D} \times \pi}$$

Where:

rpm = Revolutions per minute.
CS = Cutting speed 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 calculations.)

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

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

D = 100 mm

$\pi = 3$

$$\text{rpm} = \frac{\text{CS} \times 1000}{\text{D} \times 3}$$

$$\text{rpm} = \frac{300 \times 1000}{100 \times 3}$$

$$= 1000 \text{ rpm}$$

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

14.5.2 Roughing Cuts

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

14.5.3 Finishing Cuts

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

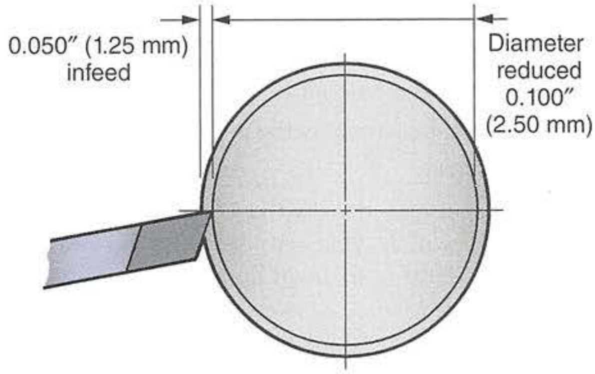
14.5.4 Depth of Cut

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

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

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



Goodheart-Willcox Publisher

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

Ed Phillips/Shutterstock.com

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

14.6 Preparing the Lathe for Operation

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

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

spindle nose, apply a drop of lubricating oil before attaching the chuck or faceplate.

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

SAFETY NOTE

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

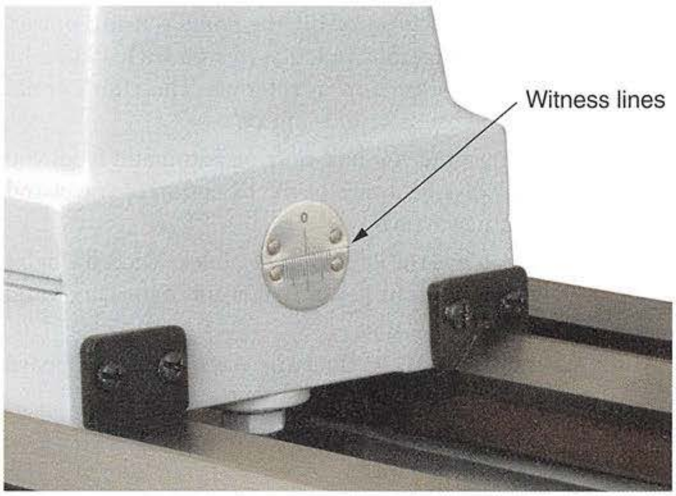
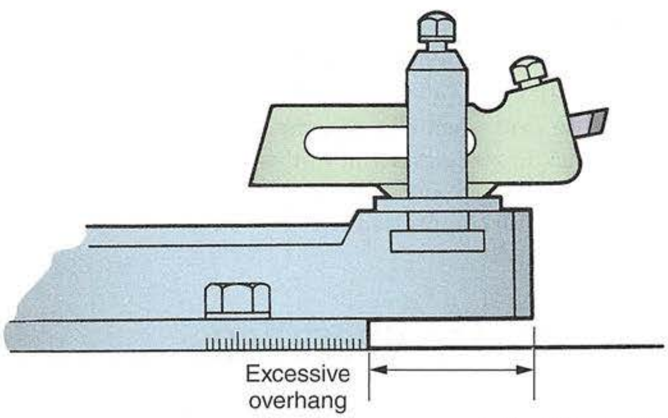


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

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



Goodheart-Willcox Publisher

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

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

14.7 Cleaning the Lathe

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

SAFETY NOTE

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

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

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

SAFETY NOTE

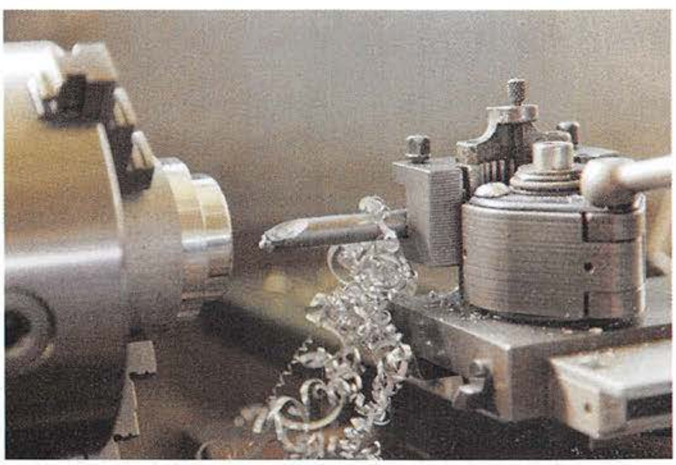
Never wrap the cord around your hand. The cord could catch and cause serious injury.

14.8 Lathe Safety

Follow these safety guidelines while operating a lathe:

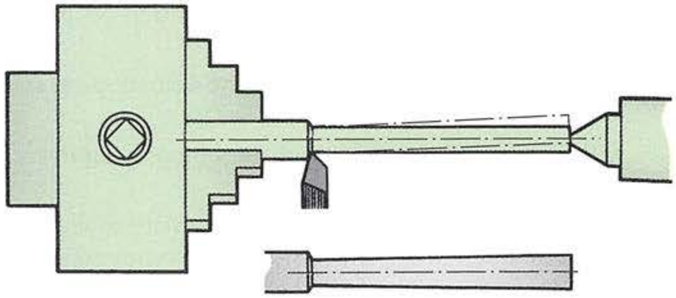
- Do not attempt to operate a lathe until you know the proper procedures and have been checked out on its safe operation by your instructor.
- Never attempt to operate a lathe while your senses are impaired by medication or other substances.
- Dress appropriately! Remove any necklaces or other dangling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

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



movit/Shutterstock.com

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



Goodheart-Willcox Publisher

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

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

14.9 Facing Operations

Facing is an operation that machines the end of the work square and reduces the work to a specific length. Facing is often the first operation performed in order to clean up the

face of the workpiece before other machining operations are performed.

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

14.9.1 Facing Work Held between Centers

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

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

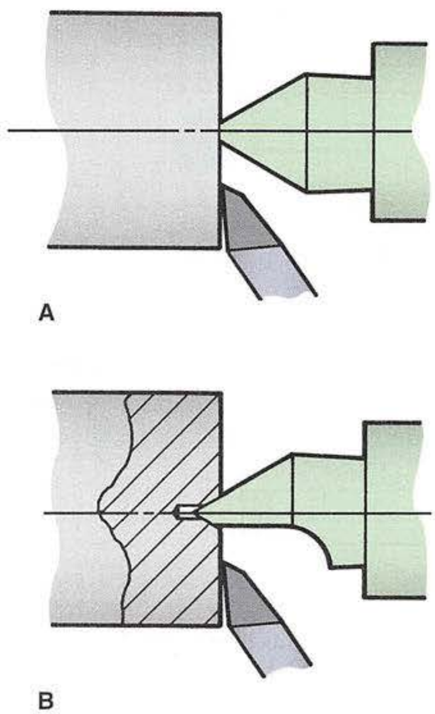


Figure 14-69. Facing stock. A—Relationship of the cutter bit to the work face when making a facing cut. B—Using a half center provides more clearance when facing the end of stock.

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

14.9.2 Facing Stock Held in a Chuck

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

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

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

14.10 Turning Operations

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

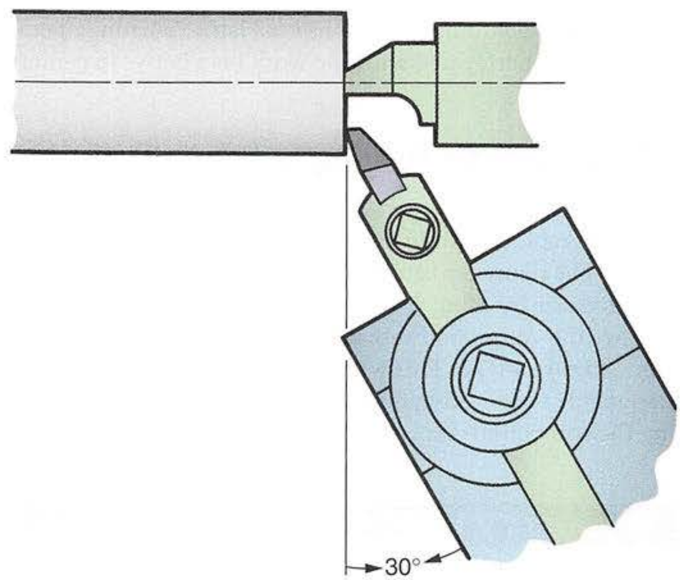
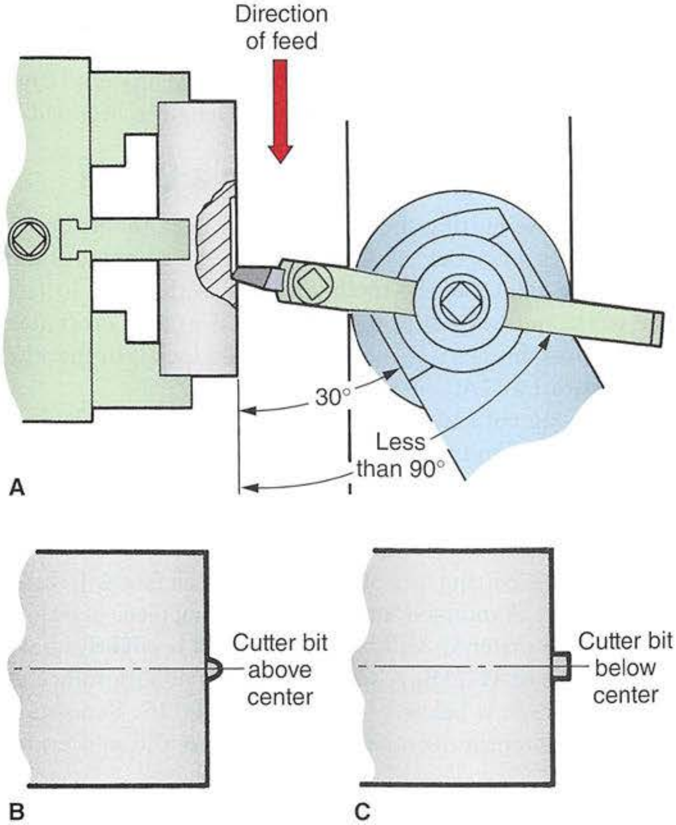


Figure 14-70. Recommended compound rest setting when facing stock to length.



Goodheart-Willcox Publisher

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

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

14.10.1 Rough Turning between Centers

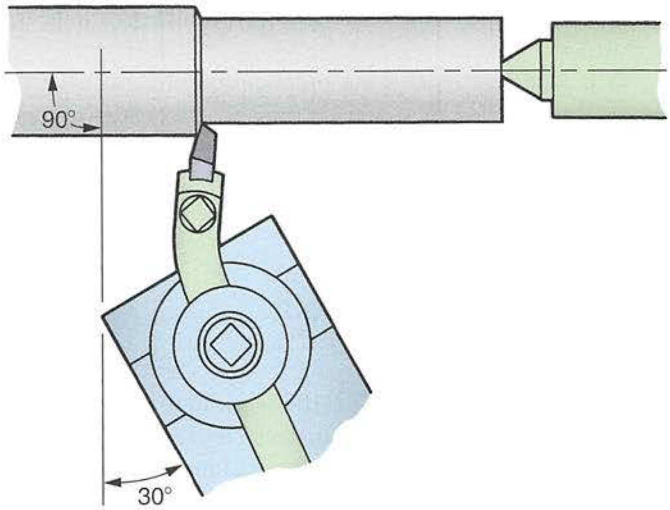
Rough turning is an operation in which excess material is cut away rapidly with little regard for the quality of surface finish. The diameter is reduced to within 1/32″ (0.8 mm) of required size by using deep cuts and coarse feeds.

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

SAFETY NOTE

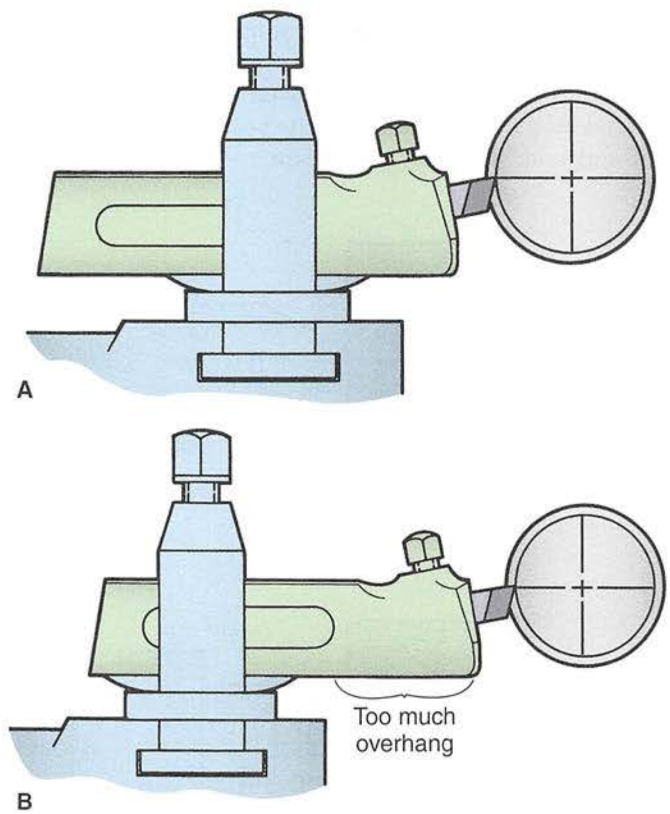
Caution: Always check the maximum distance the compound rest can be fed toward the dog or chuck without striking them before you start the lathe.

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



Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

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

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

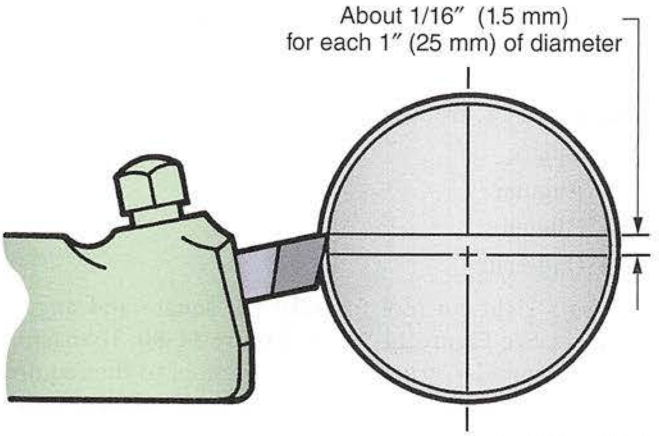


Figure 14-74. Set the tool slightly above center when rough turning.

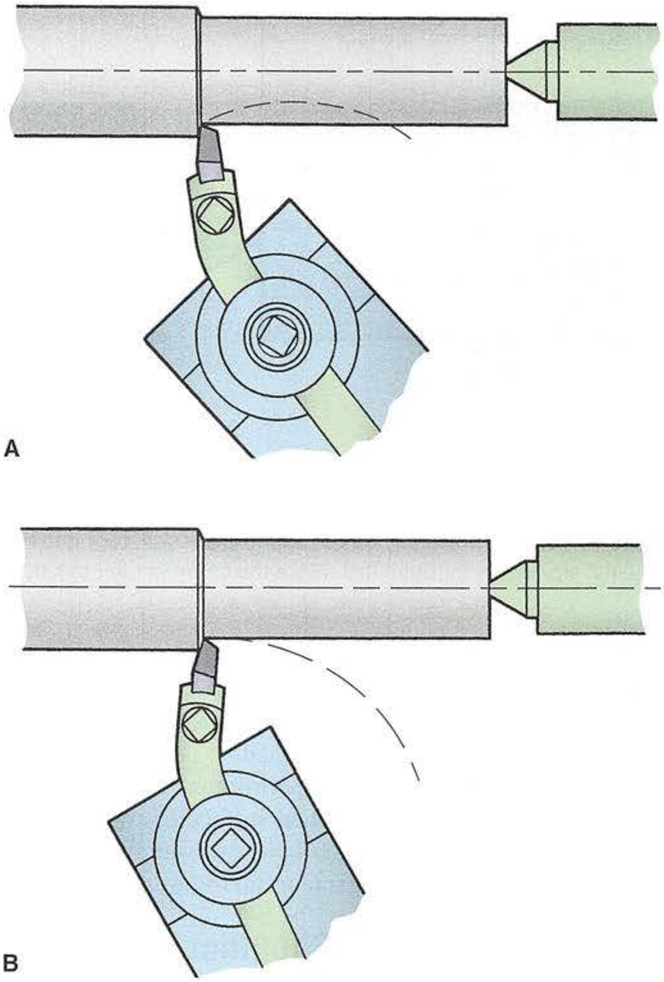


Figure 14-75. Toolholder positioning. A—An incorrectly positioned tool cuts deeper into work if the toolholder slips in the tool post. B—A correctly positioned tool swings clear of the work if the toolholder slips.

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

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

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

SAFETY NOTE

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

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

14.10.2 Finish Turning

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

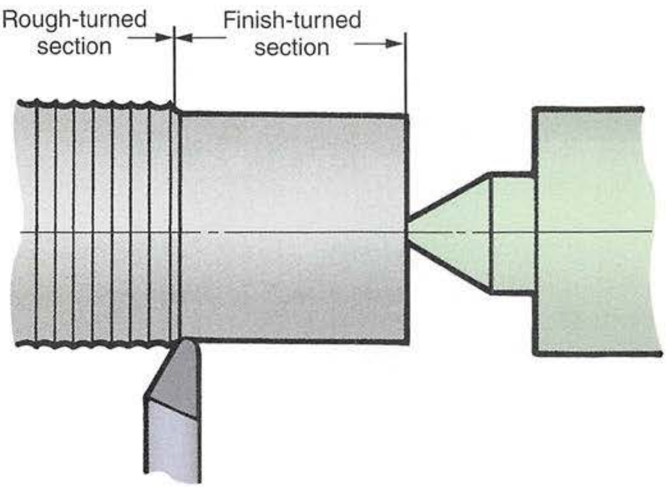
Fit a right-cut finishing tool into the toolholder. All rough and finish machining should be done toward the headstock (right to left) because the headstock offers a more solid base than the tailstock. Position the tool on center and check for adequate clearance between the compound rest and the revolving lathe dog.

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

SAFETY NOTE

Never reverse a lathe. Brake it to a stop.

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



Goodheart-Willcox Publisher

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

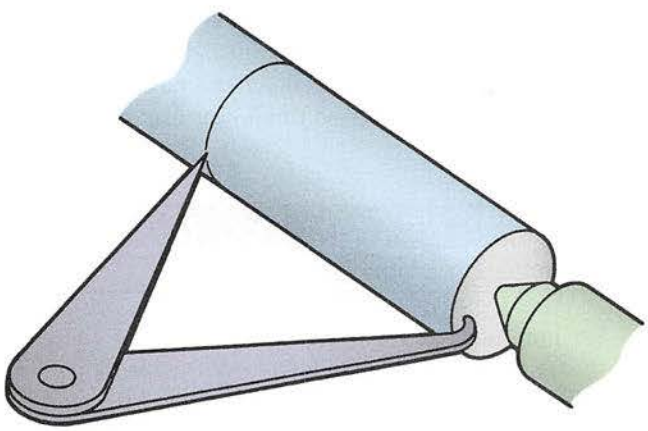
diameter is 0.008″ (0.20 mm) oversize, tool infeed should be 0.004″ (0.10 mm). Make another cut about 1/2″ (13 mm) in width at the new depth setting. Measure again to make sure the correct diameter will be machined.

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

14.10.3 Turning to a Shoulder

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

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



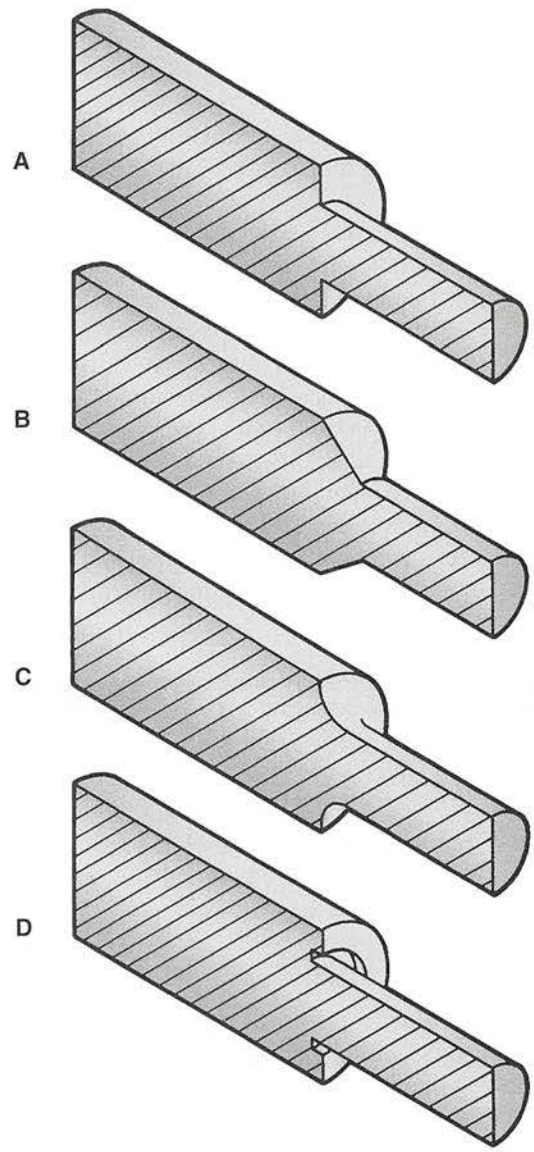
Goodheart-Willcox Publisher

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

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

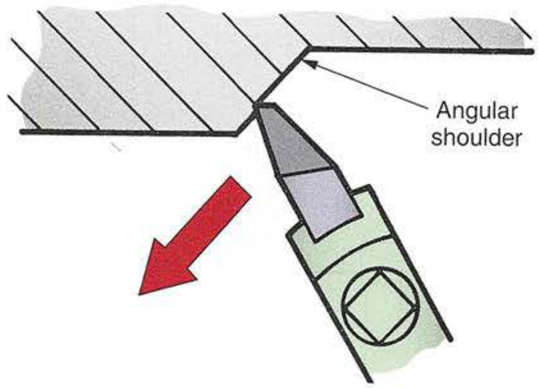
- Square.
- Angular.
- Filleted.
- Undercut.

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



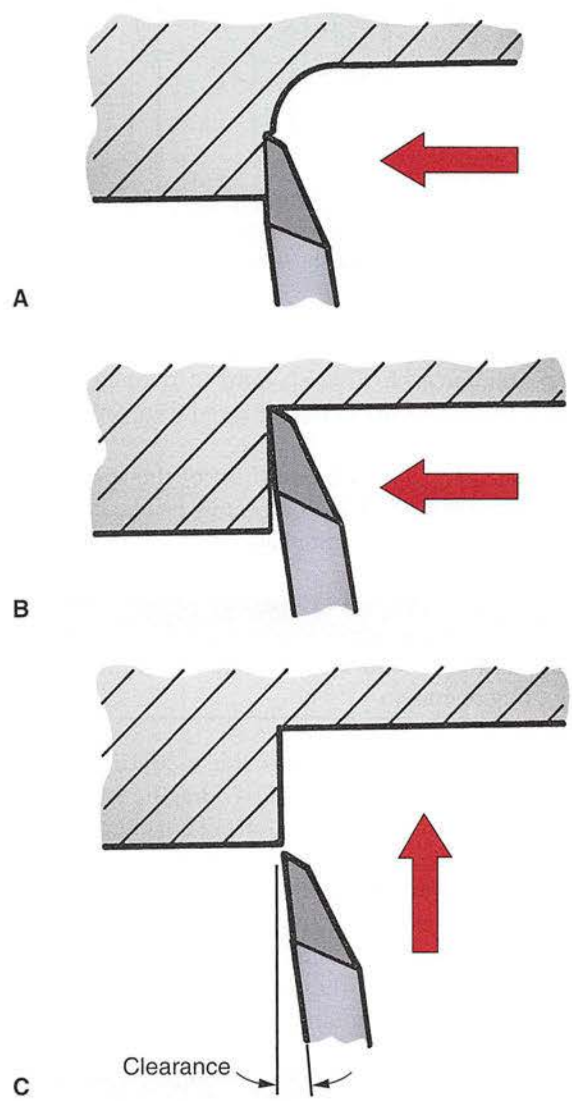
Goodheart-Willcox Publisher

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



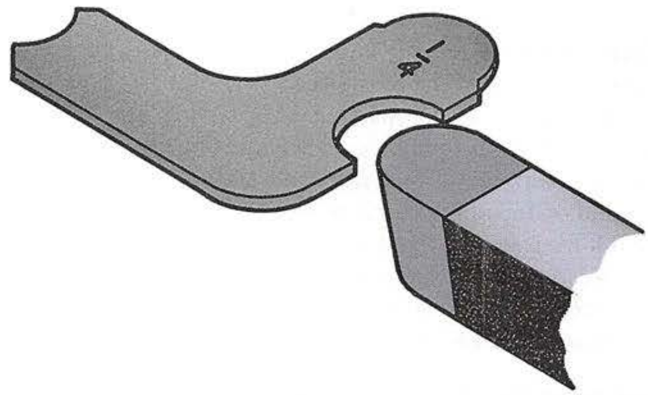
Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

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



Goodheart-Willcox Publisher

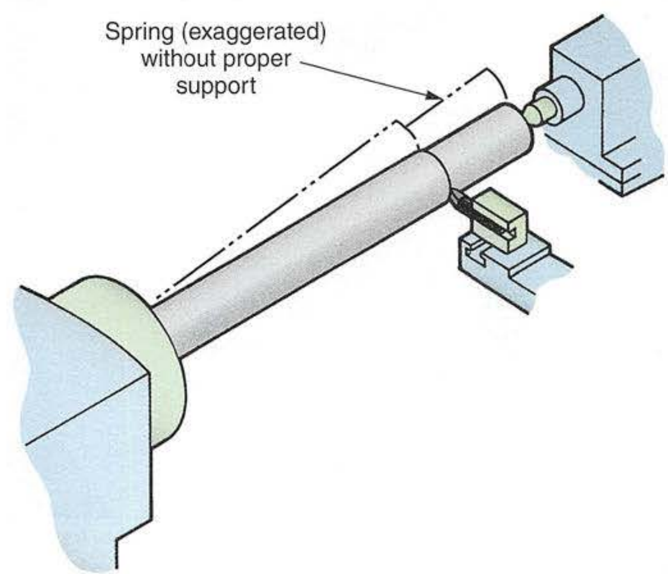
Figure 14-81. The radius on a cutter bit can be checked with a fillet gage.

14.10.4 Turning Work Held in a Chuck

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

14.11 Parting and Grooving Operations

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



Goodheart-Willcox Publisher

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

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

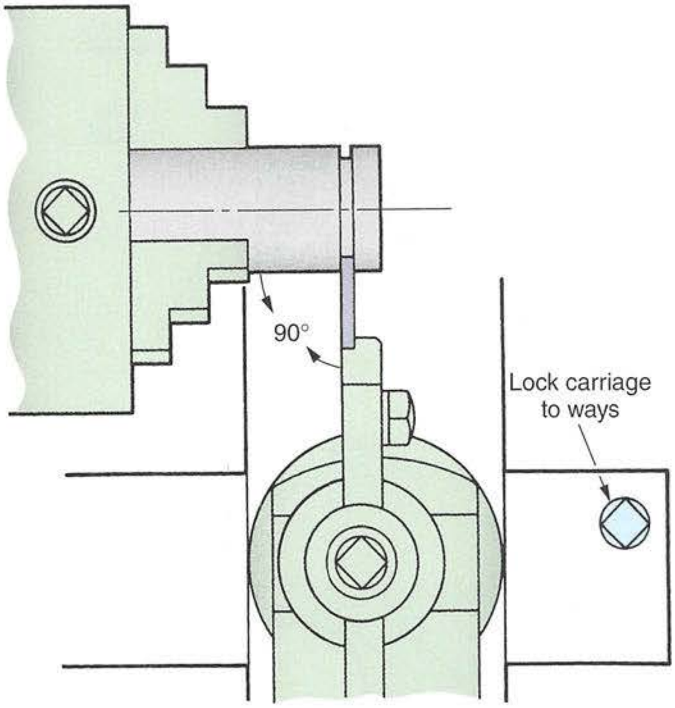
Cutting tools for parting or grooving are held in a straight or offset toolholder. They must be ground with the correct clearance (front, side, and end). A concave rake is ground on top of the cutter to reduce chip width and to prevent it from seizing (binding) in the groove.

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

14.11.1 Parting Operations

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

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



Goodheart-Willcox Publisher

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

Iscar Metals, Inc.

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

SAFETY NOTE

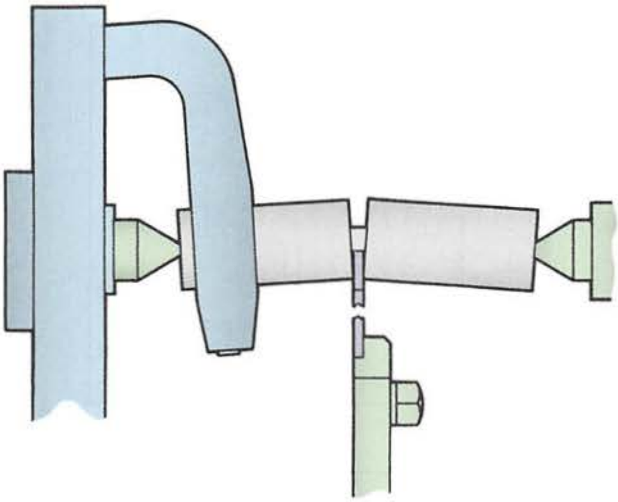
Do not forget to lock the carriage to the ways during a parting operation.

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

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

SAFETY NOTE

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



Goodheart-Willcox Publisher

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

14.11.2 Grooving or Necking Operations

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

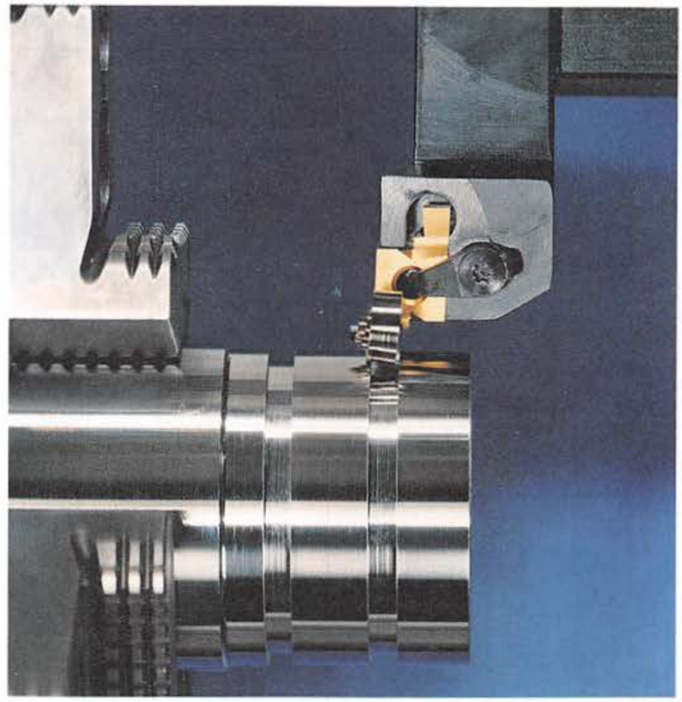
The tool is set on center and fed in until it just touches the work surface. Set the cross-feed micrometer dial to zero and feed in the tool to the specified depth. Square grooves can be machined with a parting tool or HSS tool ground to the correct width.

14.11.3 Cutting Grooves with a Parting Tool

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

14.12 Gathering Information from Chips

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



Carboloy

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

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

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

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

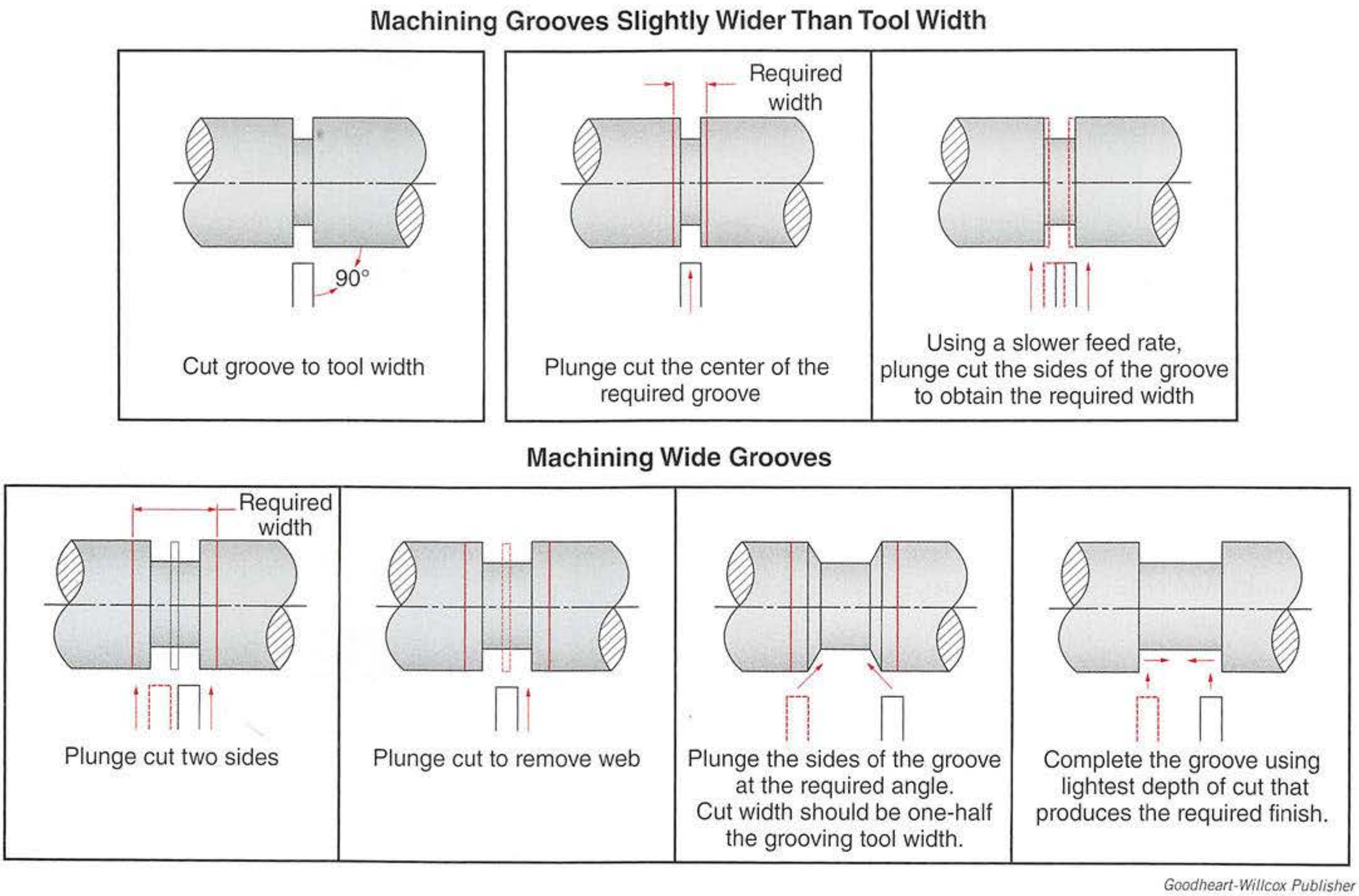


Figure 14-87. These two techniques can be used to machine grooves of various widths using a parting tool.

Chapter Review

Summary

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

Review Questions

1. What is the basic principle of lathe operation?
2. The size of a lathe is determined by the ______ and the length of the bed.
3. What determines the largest piece that can be turned between centers on a lathe?
4. Into what three functional categories do the various parts of the lathe fall?
5. Some lathes come equipped with a(n) ______, allowing for slower speeds with greater power.
6. Explain the purpose of ways on the lathe bed.
7. Most work is machined while supported by one of four methods. List them.
8. Before work can be mounted between centers, a(n) ______ must be drilled at each end of the stock.
9. When the work is turned between centers, a tapered piece will result if the centers are not aligned. Approximate alignment can be determined by two methods. What are they?
10. Briefly describe the three methods for checking center alignment if close tolerance work is to be done between centers.
11. What are the four most commonly used types of lathe chucks?
12. For most lathe operations, you will be using a single-point cutting tool made of ______.
13. What is cutting speed?
14. ______ is used to indicate the distance that the cutter moves in one revolution of the work.
15. Cutting speeds can be increased 300% to 400% by using ______ cutting tools.

For Questions 16 and 17, calculate the cutting speeds for the following metals using the information provided below.

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

16. 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.
17. 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.

For Question 18, calculate the cutting speed for the metric-size material using the information provided.

$$\text{Formula: rpm} = \frac{CS \times 1000}{D \times 3}$$

18. What spindle speed is required to finish-turn 200 mm diameter aluminum alloy? Recommended cutting speed for the material is 300 mpm.
19. What type of lubricant should you use to lubricate a lathe?
20. Accumulated metal chips and dirt are cleaned from the lathe with a(n) ______, *never* with your hands.
21. Which of the following actions are considered dangerous when operating a lathe?
    A. Wearing loose clothing and jewelry.
    B. Measuring with work rotating.
    C. Operating lathe with most guards in place.
    D Using compressed air to clean machine.
    E. All of the above.
22. Which operation machines the end of a workpiece square and reduces it to a specific length?
    A. Parting.
    B. Facing.
    C. Grooving.
    D. All of the above.
    E. None of the above.

23. List the four types of shoulders.
24. Why is a concave rake ground on top of the cutter when used for parting operations?
25. When using the parting tool, the spindle speed of the machine is about _____ the speed used for conventional turning.
26. When examining the chips created by a machining process, you note that the chips are thick and intertwining. What does this tell you about the parameters used for the operation?

The carriage supports and controls the cutting tool and is composed of a number of parts. For Questions 27–30, match each description with the correct term.

27. Fitted to the ways and slides along them.
28. Permits transverse tool movement.
29. Permits angular tool movement.
30. Used to mount the cutting tool.

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

CHAPTER 15

Other Lathe Operations

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Perform boring, drilling, reaming, and knurling operations on a lathe.
- Describe how filing and polishing operations are performed on a lathe.
- Properly set up steady and follower rests.
- Explain how and when to use mandrels.
- Perform grinding operations on a lathe.
- Describe other attachments for the lathe.
- Demonstrate familiarity with industrial applications of the lathe.

Technical Terms

automatic screw machine
boring
boring mill
follower rest
knurling
mandrel
steady rest
turret lathe

In addition to the basic lathe operations you learned about in the previous chapter, a number of other applications are possible on a lathe, such as boring, drilling, reaming, knurling, filing, polishing, grinding, and milling. Some of these operations require additional attachments, rests, or work-holding devices.

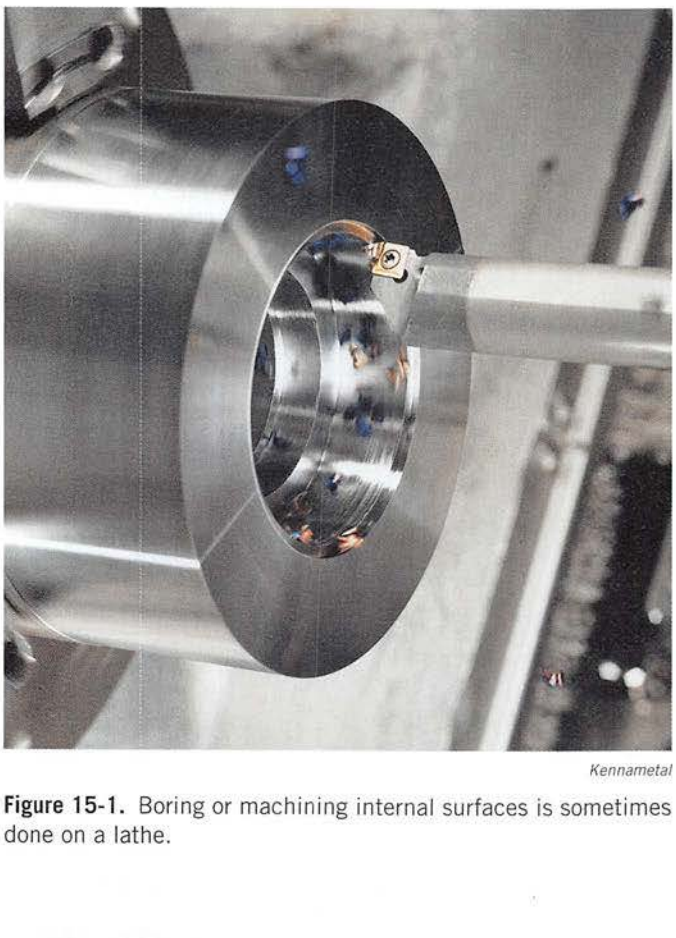
15.1 Boring on a Lathe

Boring is an internal machining operation in which a single-point cutting tool is used to enlarge a hole, **Figure 15-1**. Boring may be used to enlarge a hole to a specified size when a drill or reamer will not do the job. When properly set up, the lathe produces a hole that is concentric with the outside diameter of the work.

15.1.1 Boring Difficulties

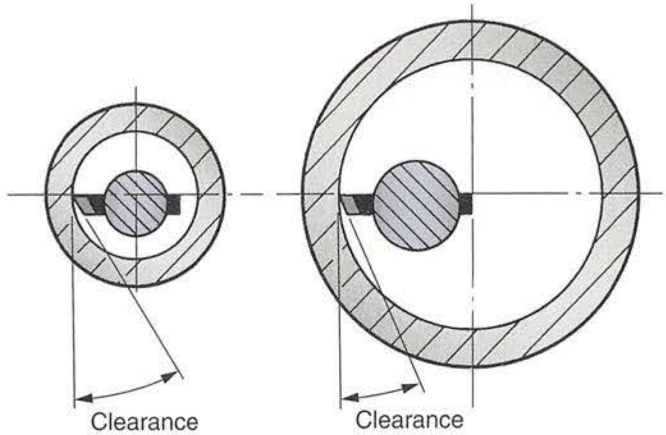
While the machining technique remains essentially the same as for external turning, you may encounter several conditions that could cause difficulty. When boring on a lathe, you must make allowances for the following:

- Movement of the cross-slide screw is reversed.
- The machinist must work "by feel" because the cutting action cannot always be observed.
- Additional front clearance must be ground on the cutting tool to avoid rubbing, **Figure 15-2**. Other than that, the shape of the cutting tool is identical to that used for external turning.
- Boring a deep or small-diameter hole requires a long, slender boring bar. The overhang makes the tool more likely to spring away from the surface being machined. It is also necessary to take several light cuts, instead of one heavy cut, to remove the same amount of material.
- Some people find that internal measuring tools are more difficult to use than those for making external measurements.



Kennametal

Figure 15-1. Boring or machining internal surfaces is sometimes done on a lathe.



Goodheart-Willcox Publisher

Figure 15-2. A cutting tool used to bore small-diameter holes requires greater front clearance to prevent rubbing.

15.1.2 Boring the Hole

The hole size to be bored determines the type and size of boring bar required. Always use the largest bar possible to give maximum tool support, **Figure 15-3**. The bar should extend from the holder only far enough to permit the tool to cut to the required hole depth, **Figure 15-4**.

The boring bar is set on center or slightly below center, with the bar parallel to tool travel, **Figure 15-5**. Check for adequate clearance when the tool is at maximum depth in the hole.

Begin by making a light cut in the same manner as you would for external machining. When the cut is complete, stop the machine. Set the cross-slide micrometer dial to zero and back the tool away from the work. Remove the boring bar from the hole.

Check the hole diameter with an inside micrometer or with a telescoping gage and micrometer. After checking hole accuracy, bring the cross-slide back to zero, and advance the tool to make another cut. The amount of infeed is determined by the boring bar being used and the material being bored. Make additional cuts, checking the hole size frequently, until the desired diameter is attained.

When making the final cut, it may be necessary to reverse tool travel after reaching the desired depth. Reverse the carriage feed, not the spindle rotation. Let the tool feed

out of the hole without changing the tool setting. This will compensate for any tool spring.

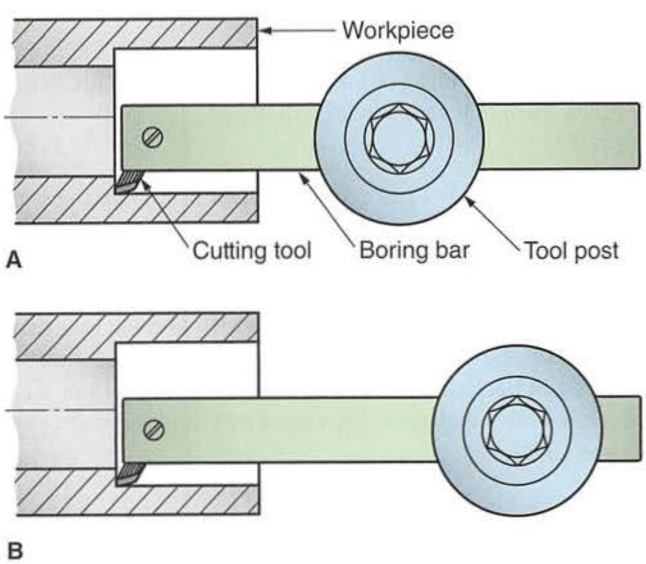
When boring holes with long, slender boring bars, it may be necessary to run the tool into the hole without changing its setting after every second or third cut to compensate for tool spring. With such long, slender boring bars, chatter is more likely to occur than when doing external work. Chatter can usually be eliminated by using one of the following methods:

- Using the largest boring bar practical for the hole size.
- Using a slower spindle speed.
- Reducing tool overhang.
- Grinding a smaller radius on the cutting tool nose.
- Placing a weight on the back overhang of the boring bar.
- Placing the tool slightly below center.
- Using a dampened boring bar.

Dampened boring bars are specifically designed to reduce vibration. The center of the bar is hollow and filled with lead shot or another material designed to absorb vibration.

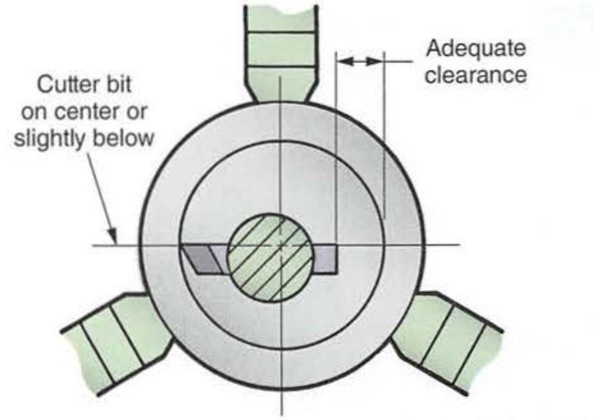
www.mini-lathe.com

Figure 15-3. A set of carbide-tipped boring bars with 1/2″ (12.5 mm) diameter shanks.



Goodheart-Willcox Publisher

Figure 15-4. Keep the cutting tool as close to the tool post as possible for maximum tool support. A—Properly positioned boring bar. B—Boring bar projecting too far from the tool post. The resulting vibration and chatter could produce a rough machined surface.



Goodheart-Willcox Publisher

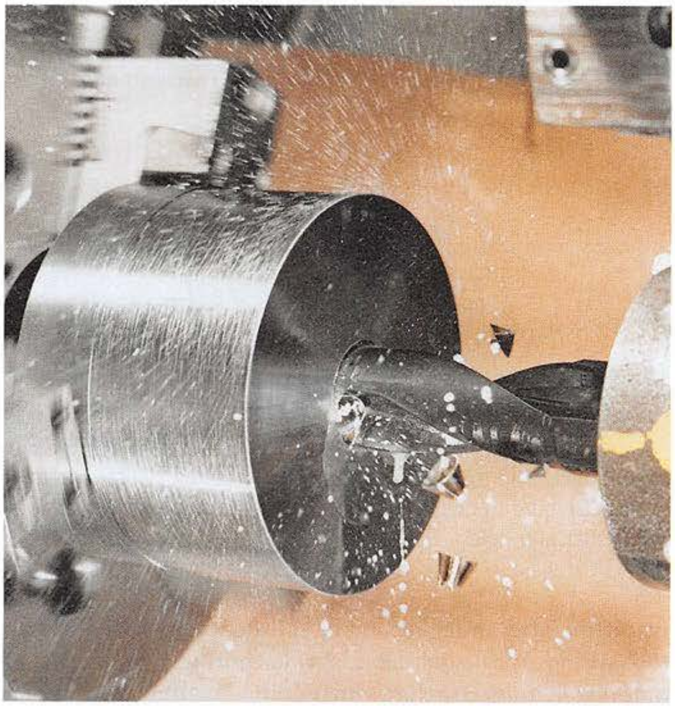
Figure 15-5. The tool is set on, or slightly below, center when boring. Be sure to check for adequate clearance between the boring bar and hole.

15.2 Drilling on a Lathe

When a hole is to be cut in solid stock, the usual practice is to hold it in a suitable chuck and mount the drill in the tailstock. Drilling is accomplished on a lathe by feeding the stationary drill into the rotating workpiece, **Figure 15-6**.

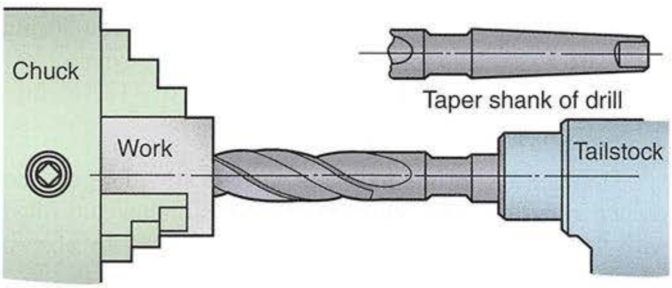
When using standard twist drills to drill holes that are 1/2″ (12.5 mm) or less in diameter, a straight shank drill is placed in a Jacobs chuck. The Jacobs chuck is then fitted into the tailstock spindle. Holes larger than 1/2″ (12.5 mm) in diameter are made with taper shank drills, **Figure 15-7**.

Drills that have taper shanks too large to be fitted in the tailstock can be used if mounted as shown in **Figure 15-8**. A dog is fitted to the neck of the drill. The tool is set up to permit the tailstock center to press into the center hole in the drill tang.



Kennametal

Figure 15-6. When drilling on a lathe, the workpiece is held in the headstock and rotated while the stationary drill, held in the tailstock, is advanced into the work. The cutting fluid helps to reduce heat and flush away chips.



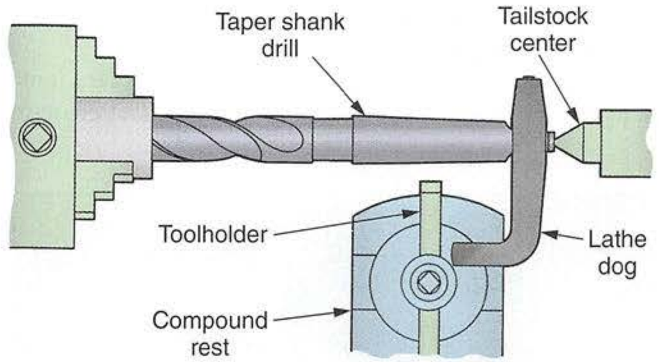
Goodheart-Willcox Publisher

Figure 15-7. Drills larger than 1/2" (12.5 mm) in diameter are usually fitted with a self-holding taper that fits into the tailstock spindle of the lathe.

The drill's cutting point bears against the rotating work. The drill is prevented from revolving by the dog bearing against the compound rest. The tailstock center keeps the drill aligned and enables it to be fed into the material by the tailstock handwheel. The makeshift lathe dog setup can be avoided by using a commercial drill holder, **Figure 15-9**.

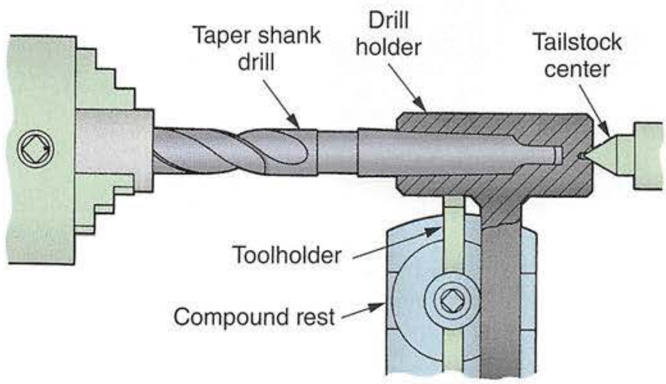
SAFETY NOTE

Extreme care must be used to prevent the drill from slipping off the tailstock center after it breaks through the work.



Goodheart-Willcox Publisher

Figure 15-8. When a drill shank is too large to be fitted into the tailstock, a lathe dog can be used to keep it from revolving. The tail of the lathe dog is supported by the compound rest. This type of drilling requires care to prevent the drill from slipping off the tailstock center when the full drill diameter breaks through the work.



Goodheart-Willcox Publisher

Figure 15-9. Large taper shank drills can be used on the lathe by fitting them in a commercial drill holder.

Accuracy in drilling requires a centered starting point for the drill. A starting point made with a combination drill and countersink is adequate for most jobs. Holes over 1/2" (12.5 mm) in diameter require a pilot hole. The pilot hole should have a diameter equal to the width of the larger drill's dead center. See **Figure 15-10**. Ample clearance must be provided in back of the work so that the drill will not strike the chuck or headstock spindle when it breaks through, **Figure 15-11**.

15.3 Reaming on a Lathe

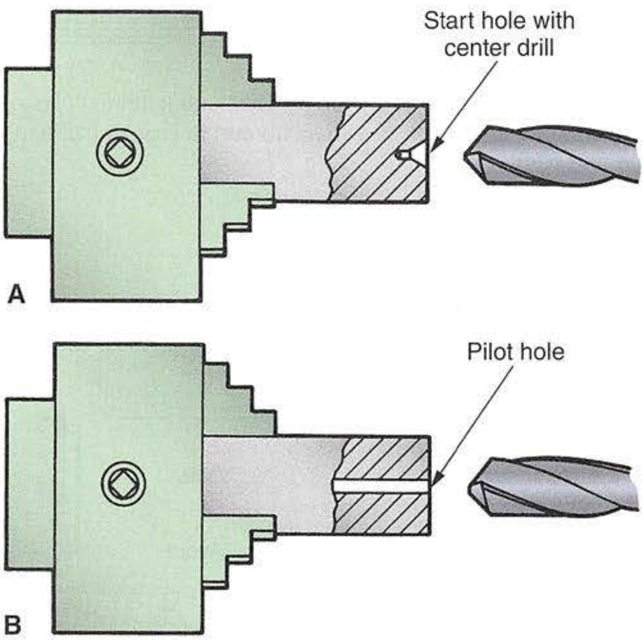
Reaming is an operation used to make a hole accurate in diameter and finish, **Figure 15-12**. The hole is drilled slightly undersized to allow sufficient stock for reaming. The allowance for reaming depends on the hole size, as follows:

- For hole sizes ranging up to 1/4″ (6.5 mm) in diameter, allow 0.010″ (0.25 mm) of material for reaming.
- For hole sizes from 1/4″ (6.5 mm) to 1/2″ (12.5 mm) in diameter, allow 0.015″ (0.4 mm).
- For hole sizes from 1/2″ (12.5 mm) to 1.0″ (25.0 mm) in diameter, allow 0.020″ (0.5 mm).
- For hole sizes from 1.0″ (25.0 mm) to 1.5″ (37.5 mm) in diameter, allow 0.025″ (0.6 mm).
- For hole sizes above 1.5″ (37.5 mm) in diameter, allow 0.030″ (0.8 mm) for reaming.

When reaming, use a cutting speed about two-thirds the speed you would use for a similar size drill with the material being machined. Also, use a slow, steady feed with an adequate supply of cutting fluid. Remove the reamer from the hole before stopping the machine.

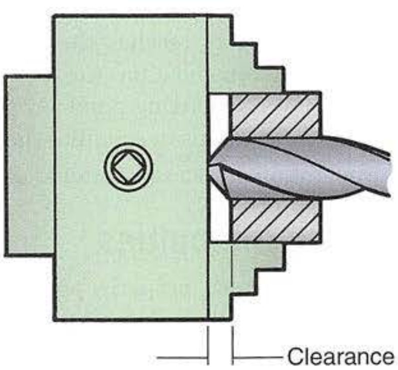
If you are using a hand reamer, do not apply power to the workpiece mounted in the chuck. Fit the reamer into the hole, supporting the shank end with the tailstock center. Use an adjustable wrench to turn the reamer in a clockwise direction, **Figure 15-13**.

When removing a reamer from the hole, continue to rotate the tool clockwise. Avoid turning it counterclockwise because that would ruin the tool's cutting edges.



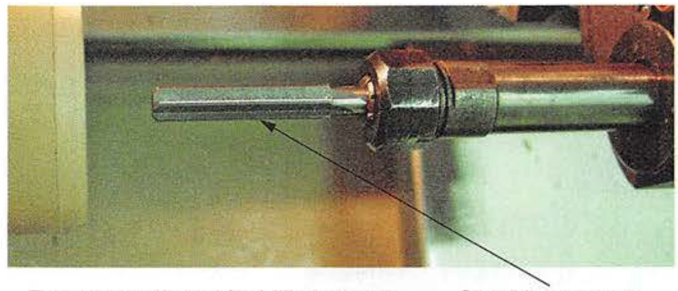
Goodheart-Willcox Publisher

Figure 15-10. Centering the drill. A—The drill will cut exactly on center if the hole is started with a center drill. B—Holes larger than 1/2″ (12.5 mm) in diameter require a pilot hole.



Goodheart-Willcox Publisher

Figure 15-11. There must be enough clearance between the back of the work and the chuck face to permit the drill to break through the work without damaging the chuck.



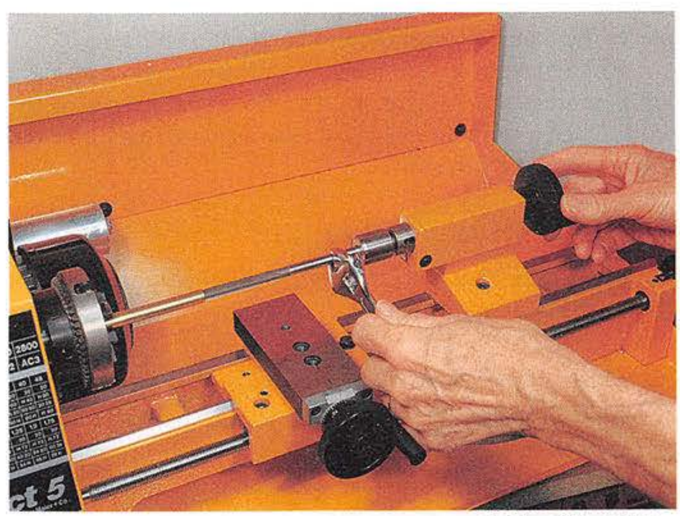
Reamer at 1/2 to 2/3 drilling speed   Chucking seamer

Goodheart-Willcox Publisher

Figure 15-12. A straight-flute reamer mounted in an extension holder being used to finish a hole in a plastic workpiece.

15.4 Knurling on a Lathe

Knurling is the process of forming horizontal or diamond-shaped serrations (raised grooves or teeth) on the circumference of the work, **Figure 15-14**. Knurling is used to provide a gripping surface, change the appearance of the work, or increase the work's diameter. It is done with a knurling tool mounted in the tool post, **Figure 15-15**. The knurled pattern



Goodheart-Willcox Publisher

Figure 15-13. Using a hand reamer on the lathe. Never turn on the power when performing hand reaming operations.

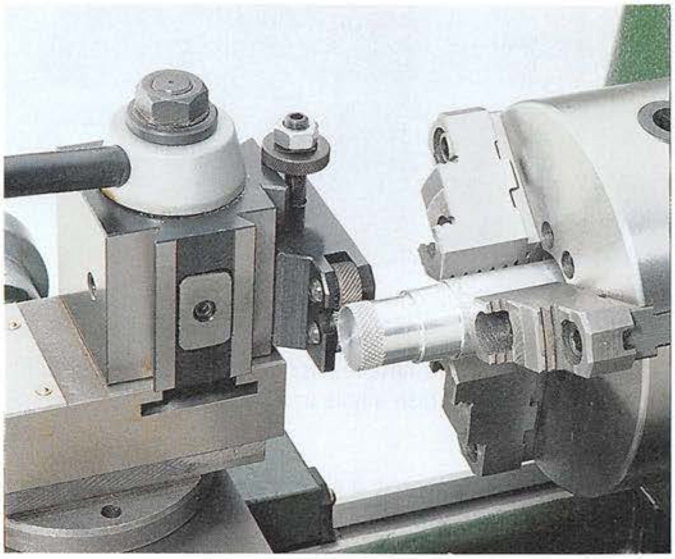


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 15-14. Knurling rollers are being used to form serrations on a part.

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

Figure 15-15. One type of knurling tool.

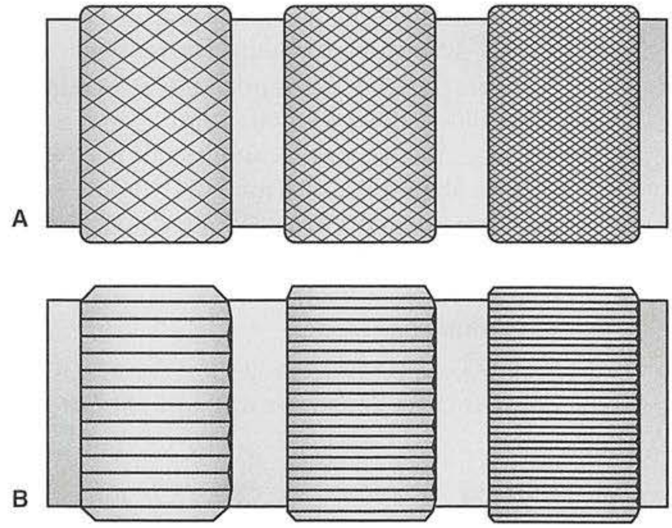
is raised by rolling the knurls against the metal. This displaces the metal into the required pattern.

Angular knurls raise a diamond pattern, and a straight knurl produces a straight pattern along the length of the work. The patterns can be produced in coarse, medium, and fine pitch. See **Figure 15-16**.

15.4.1 Knurling Procedure

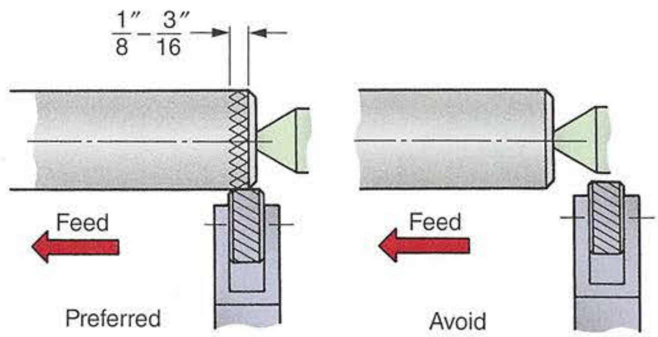
If a knurling tool is not properly set up, the knurls will not track and will quickly dull. The following procedure is recommended:

1. Mark off the section to be knurled.
2. Adjust the lathe to a slow back-geared speed and a fairly rapid feed.
3. Place the knurling tool in the tool post. Bring it up to the work. Both wheels must bear evenly on the work with their faces parallel with the centerline of the piece, **Figure 15-17**.



Goodheart-Willcox Publisher

Figure 15-16. Knurling patterns. A—Diamond knurl in coarse, medium, and fine pitch. B—Straight knurl in coarse, medium, and fine pitch.



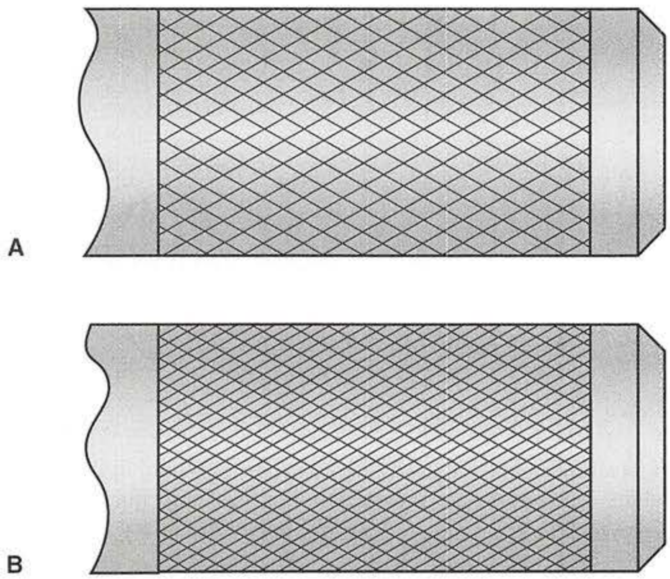
Goodheart-Willcox Publisher

Figure 15-17. Always start the knurl on the work.

4. Start the lathe and slowly force the knurls into the work surface until a pattern begins to form. Tool travel should be toward the headstock whenever possible. Engage the automatic feed and let the tool travel across the work. Flood the work with cutting fluid.
5. When the knurling tool reaches the proper position, reverse spindle rotation and allow the tool to move back across the work to the starting point. Apply additional pressure to force the knurls deeper into the work.
6. Repeat the operation until a satisfactory knurl is formed.

15.4.2 Knurling Difficulties

If knurling is not performed properly, problems can arise and destroy the work. A common problem is the double-cut knurl, **Figure 15-18**, in which one wheel of the knurling tool makes twice as many ridges as the other. A double-cut knurl is usually caused by one wheel being dull. Raising or lowering the knurling tool to put more pressure on the dull wheel frequently eliminates the trouble. Pivoting the tool slightly



Goodheart-Willcox Publisher

Figure 15-18. Double-cut knurl. A—This is a correctly made diamond knurl pattern. B—A double-cut diamond knurl results when one knurl wheel is slightly above or below center.

to allow the right side of the wheels to apply more pressure may also help.

Considerable side pressures are developed during knurling operations. Watch the tool carefully. Do not permit the work to slip in the chuck or loosen on the tailstock center. If a ball-bearing center is not used, keep the tailstock center well lubricated.

Perform knurling before turning a shaft to a smaller diameter. If knurling is done after the smaller diameter has been machined, the work will spring away from the tool, giving the surface a superficial (light, nonpenetrating) knurl. It may also cause a permanent bend in the workpiece. See **Figure 15-19**.

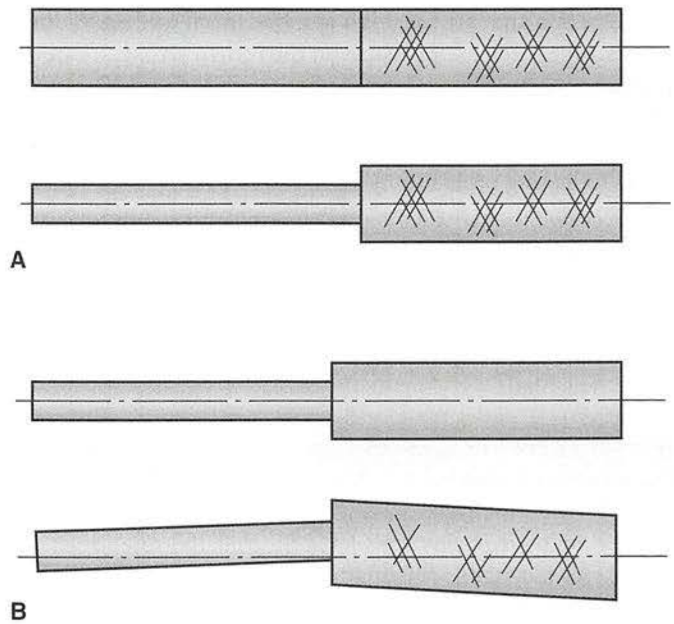
Avoid applying too much pressure to the knurling tool. The work surface becomes hardened during the operation and the knurled section could "flake off." High pressure also tends to bend the shaft.

SAFETY NOTE

Never stop the lathe with the knurls engaged in the work. The piece will take on a permanent bend.

15.5 Filing and Polishing on a Lathe

Lathe filing is done to remove burrs, round off sharp edges, and blend in form-cut outlines. A file is not intended to replace a properly sharpened cutting tool and should not be used to improve the surface finish on a turned section.

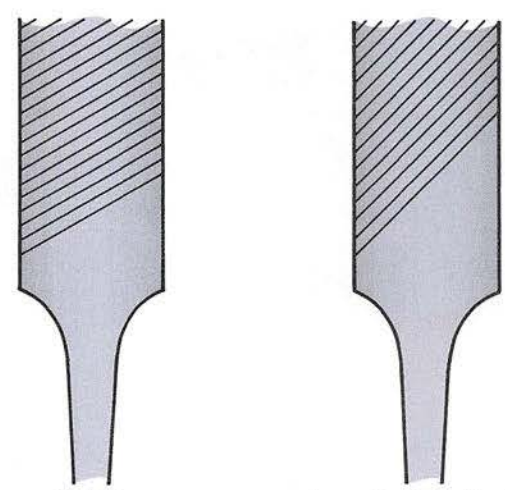


Goodheart-Willcox Publisher

Figure 15-19. Knurling problems. A—Do knurling before turning a shaft to a smaller diameter. B—If knurled after being turned to a smaller diameter, the shaft may take on a permanent bend and receive only a superficial (very light) pattern.

15.5.1 Filing on a Lathe

When filing on a lathe, avoid holding the tool stationary against the work. Keep the tool moving across the area being filed. If the file is held in one position, it will load with metal particles and score the surface of the work. An ordinary mill file produces satisfactory results. However, a long-angle lathe file produces a superior cutting action. See **Figure 15-20**.



Goodheart-Willcox Publisher

Figure 15-20. The cutting edges on a long-angle lathe file are at a different angle from those on a standard mill file.

Operate the lathe at high spindle speed and apply long, even strokes. Release pressure on the return stroke. If uneven pressure is applied, out-of-round work will result. Clean the file often.

As simple as filing may appear, it can be quite dangerous if the following precautions are not observed:

- Move the carriage out of the way and remove the tool post.
- Use the left-hand method of filing, **Figure 15-21**. This involves holding the file handle in the left hand. The right hand is then clear of the revolving chuck or faceplate.

SAFETY NOTE

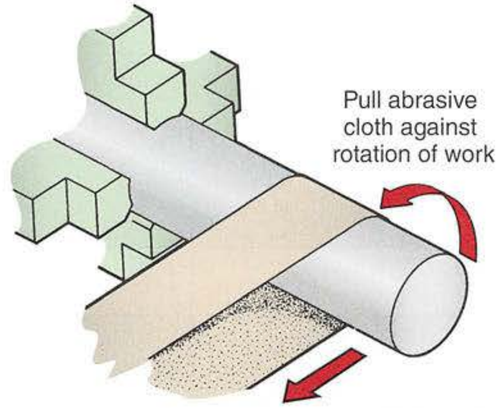
1. Avoid the right-hand method of filing. The right-hand method places your left arm over the rotating chuck or faceplate. This method can result in serious injury.
2. Never use a file without a handle. It is too easy to drive the unprotected tang into your hand.

15.5.2 Polishing on a Lathe

Polishing is sometimes done on a lathe using a strip of abrasive cloth suitable for the material to be polished. Grasp the strip between your fingers and hold it across the work, **Figure 15-22**. If more pressure is required, mount the abrasive cloth on a strip of wood or on a file, **Figure 15-23**. A high spindle speed is used for polishing.

The finer the abrasive used, the finer the resulting finish. A few drops of machine oil on the abrasive will improve the finish. For the final polish, reverse the cloth so the cloth backing, rather than the abrasive, is in contact with the work surface. Like filing, polishing is not a substitute for a properly sharpened tool bit.

Carefully and thoroughly clean the lathe after polishing operations. If not removed, abrasive particles from the cloth will cause rapid wear of the machine's moving parts.

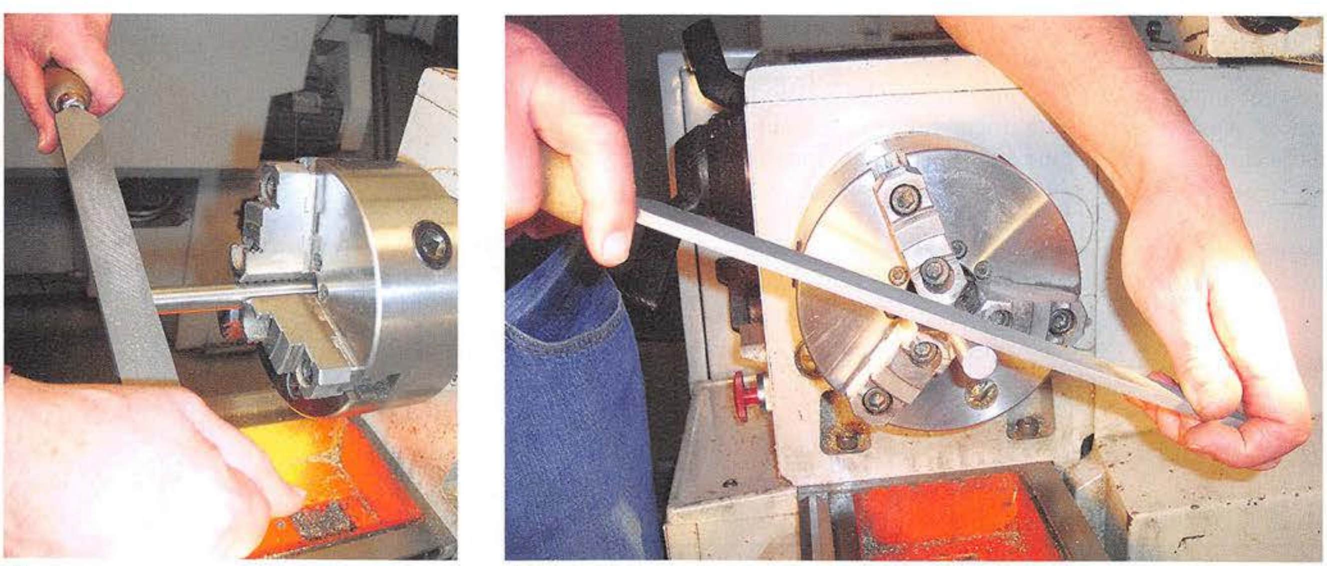


Goodheart-Willcox Publisher

Figure 15-22. Polishing with an abrasive cloth held in the hands. Keep your hands away from the revolving chuck or dog.

15.6 Using Steady and Follower Rests

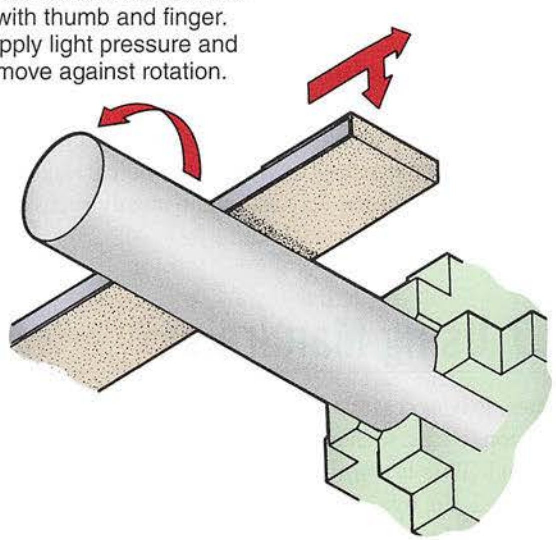
The steady rest and the follower rest are needed to provide additional support when the workpiece is long and thin. This additional support keeps the work from springing or bending



A B

Goodheart-Willcox Publisher

Figure 15-21. A—The left-hand method of filing on the lathe is preferred. B—Avoid using the right-hand method as shown here. Notice that the machinist's left hand and arm are extended over the revolving chuck, which can cause injury.



Goodheart-Willcox Publisher

Figure 15-23. More pressure can be applied if the abrasive cloth is supported by a file or block of wood.

away from the cutting tool. The support is also needed to reduce chattering when long shafts are machined. See **Figure 15-24**.

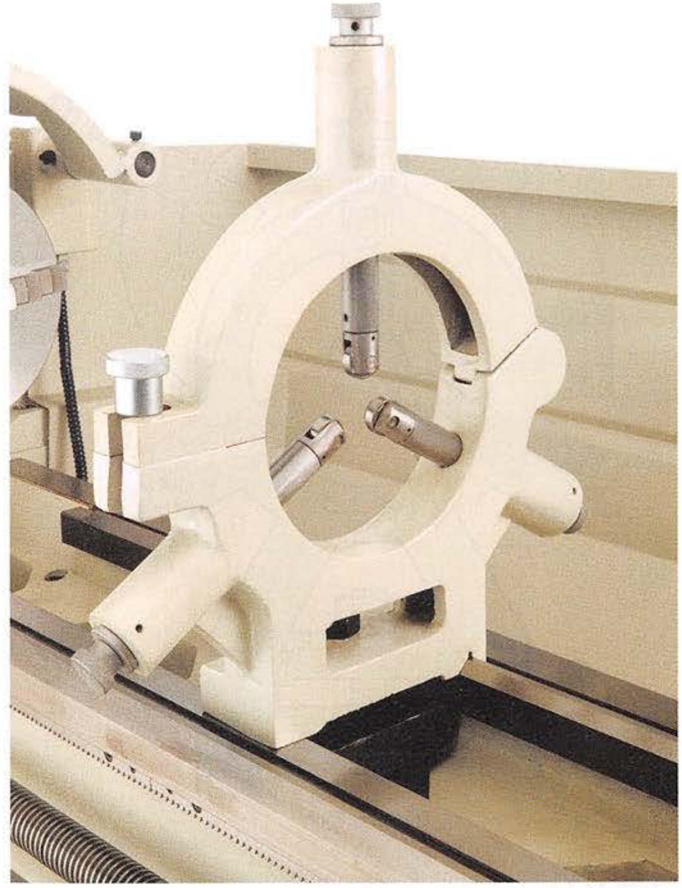


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 15-24. When the nature or shape of work prevents it from being mounted between centers, devices called "rests" are used to provide support. This lathe is equipped with a steady rest.

GREEN MACHINING

Reducing Chips

Managing waste from chips is critical for eco-conscious machine shops. Optimum cutting produces long, curling chips. Leaving these chips intact makes collection more difficult and wasteful. Cut the chips at intervals to make gathering and recycling easier. Chips can then be repurposed into new objects, like wire brushes and scouring pads, or pressed into bricks for pick-up by industry recyclers.

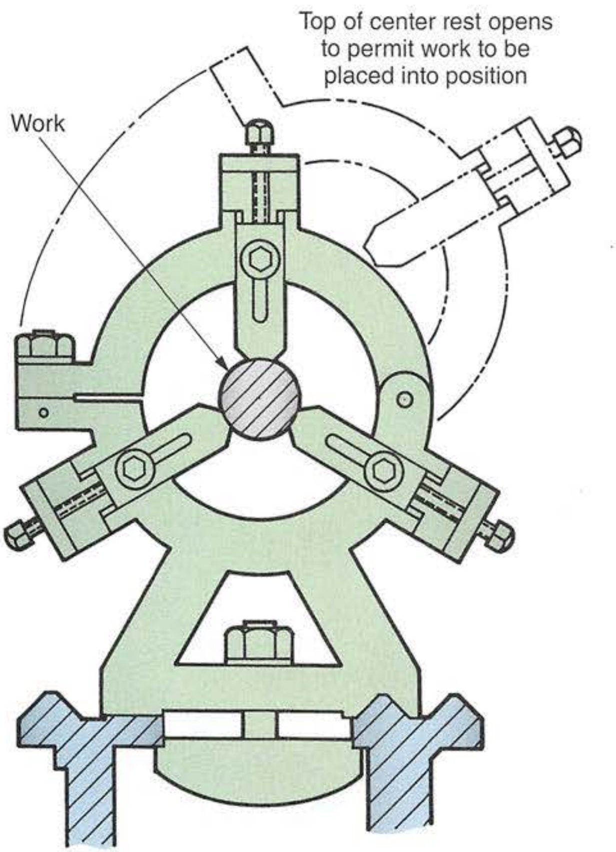
Before collection or bricking, treat all chips to remove excess cutting fluid. To remove cutting fluid, drain the chips or use centrifugal force to separate the fluid from the chips. Careful removal of cutting fluid from chips may allow the fluid to be reused as well.

15.6.1 Steady Rest Setup

The ***steady rest***, sometimes called a *center rest*, is bolted directly to the ways. It has three adjustable jaws, each with individual locking screws. The upper portion of the attachment is fitted with a single jaw that can be opened to place the work in position, **Figure 15-25**.

To set up a steady rest, use the following procedure:

1. Bolt the attachment to the ways at the desired position.
2. Back off all jaws and open the upper section.
3. Mount the work between centers or in a chuck. Support the free end with the tailstock center.
4. Lower and lock the upper segment in place.
5. Adjust the jaws up to the work and lock them into position. The jaws act as bearing surfaces where they contact the work. They must be well lubricated.
6. If the shaft being machined is unsuitable as a bearing surface (rough surface, out-of-round, square), use a cat head (a device that provides a bearing surface). Take care to center the shaft within the cat head. See **Figure 15-26**.
7. When machining at the end of a long shaft that cannot be supported with the tailstock, center the work in the chuck. Adjust the center rest to the work as close to the chuck as possible, then move it to a point where the support is needed. You can use the same technique when performing drilling, reaming, tapping, or other operations on the end of a long shaft.



Goodheart-Willcox Publisher

Figure 15-25. To permit easy installation of work, the top of the steady or center rest swings open. Take care to accurately center the work.

15.6.2 Follower Rest Setup

The ***follower rest*** operates on the same principle as the steady rest and is used in a similar manner. The follower rest differs slightly in that it provides support directly in back of the cutting tool and follows along during the cut. See **Figure 15-27.**

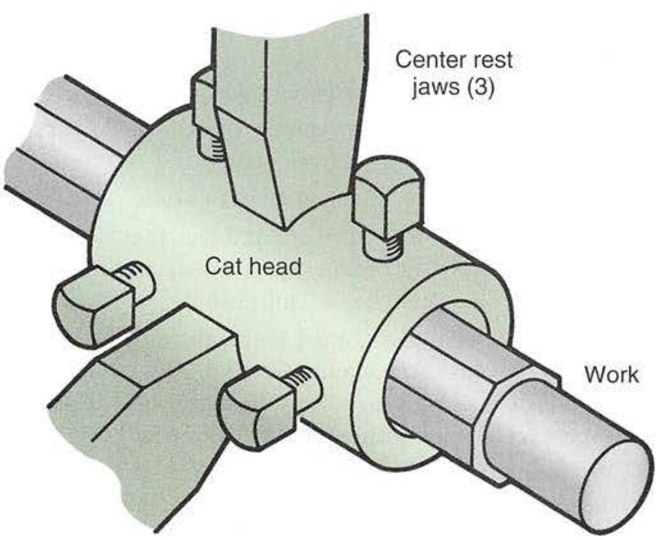
The follower rest bolts directly to the carriage and the jaws adjust in the same way as on the steady rest. The jaws must be readjusted after each cut.

15.7 Using Mandrels

At times, it is necessary to machine the outside diameter of a piece concentric with a hole that has been previously bored or reamed. This is a simple operation if the material can be held in the lathe by conventional means. There are, however, times when the material cannot be gripped solidly to permit accurate machining. In such cases, the work is mounted on a ***mandrel,*** (a slightly tapered, hardened steel shaft), and turned between centers, **Figure 15-28.**

A solid mandrel is made from a section of hardened steel that has been machined with a slight taper (0.0005″ per inch), **Figure 15-29A.** These mandrels are made in standard sizes starting at 1/8″ in diameter. The size is stamped on the large end. The other end is slightly smaller than the specified size to permit easy installation in the work.

An expansion mandrel permits turning work in which the opening is not a standard size. See **Figure 15-29B.** The shaft and sleeve have corresponding tapers and are machined



Goodheart-Willcox Publisher

Figure 15-26. A cat head provides a bearing surface when needed.

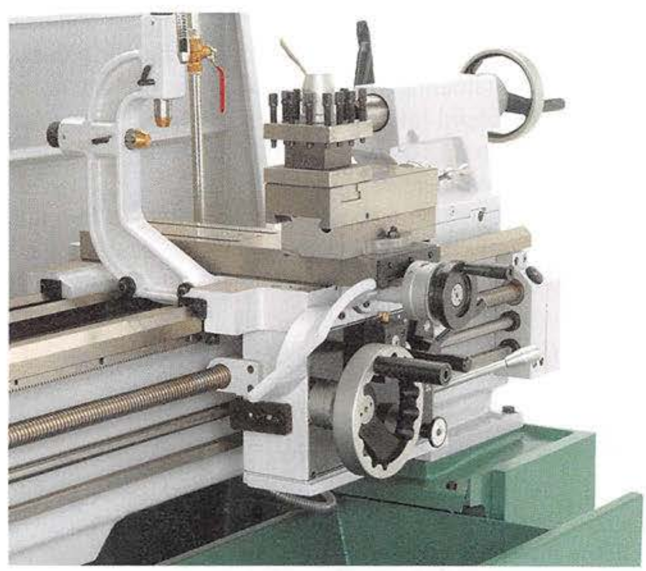
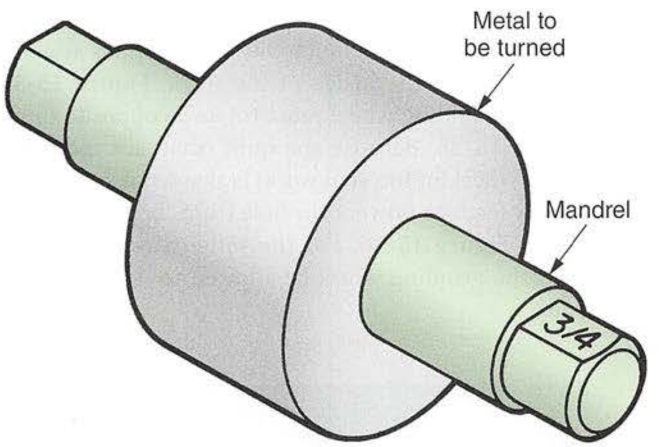


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 15-27. A follower rest is used to support long slender workpieces during machining operations.

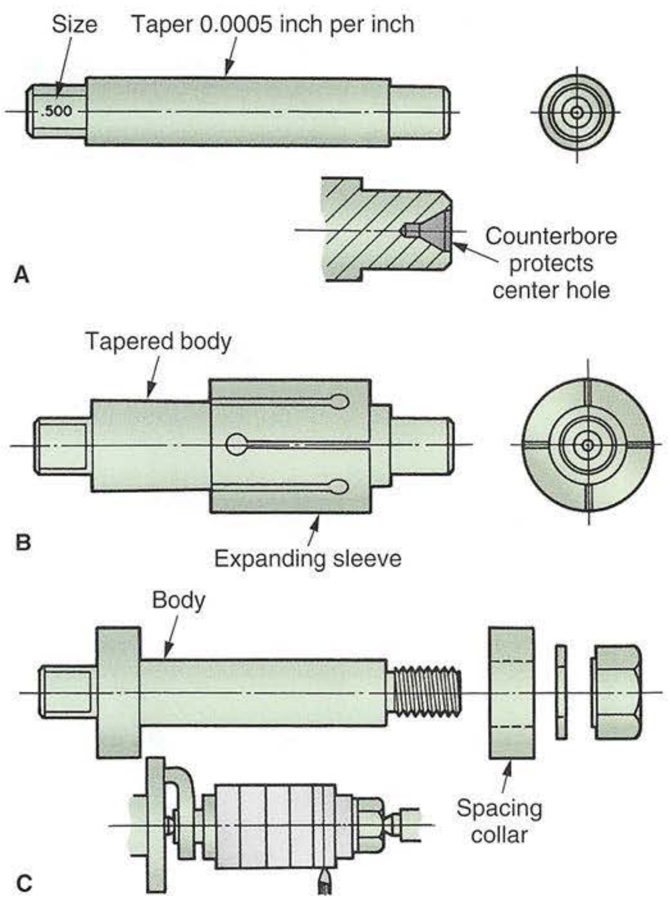
from hardened steel. The sleeve is slotted so it can expand when forced onto the tapered shaft.

A gang mandrel, **Figure 15-29C**, is helpful when many pieces of the same configuration must be turned. Several pieces are mounted on the mandrel and separated with spacing collars. They are locked in place by tightening a nut.



Goodheart-Willcox Publisher

Figure 15-28. Work mounted on a lathe mandrel.



Goodheart-Willcox Publisher

Figure 15-29. Different kinds of mandrels. A—Solid mandrel. B—Expansion mandrel. C—Gang mandrel.

Work is pressed on a mandrel with a mechanical arbor press. The work must first be checked for burrs and cleaned. Lubricate the work with a light oil to prevent it from "freezing" on the mandrel.

The mandrel is mounted between centers and driven by a lathe dog. Use care so the tool does not come into contact with the mandrel during the machining operation. In an emergency, a mandrel can be machined from a section of mild steel.

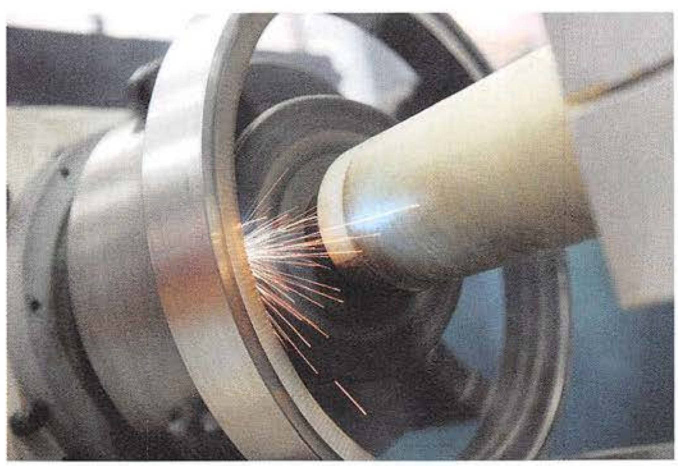
15.8 Grinding on the Lathe

The tool post grinder permits the lathe to be used for internal and external grinding, **Figure 15-30**. With a few simple attachments, it is possible to sharpen reamers and milling cutters on the lathe. You can also grind shafts and true lathe centers.

Since steel parts sometimes warp during heat treatment, it is common to machine the piece to within 0.010″ to 0.015″ (0.2 mm to 0.3 mm) of finished size. After heat treatment, the metal is mounted on the lathe and ground to finished size. A light grinding cut is made on each pass. When done properly, grinding produces a very smooth finish.

15.8.1 Preparing a Lathe for Grinding

Particles of the grinding wheel wear away during the grinding operation. Abrasive particles can cause excessive wear if they get into moving parts, so it is important to protect the lathe from them. When preparing to grind, cover the lathe bed, cross-slide, and other parts with canvas or heavy kraft paper to protect them from abrasive dust and grit. It is also good practice to place a small tray of water or oil just below the grinding wheel to collect as much grit and dust as possible.



AMT—The Association for Manufacturing Technology

Figure 15-30. A tool post grinder used for internal grinding on a lathe.

SAFETY NOTE

When placing the protective covering on the lathe, be sure the covering material cannot become entangled in the lead screw or other moving parts.

15.8.2 Preparing the Grinder

Select the grinding wheel best suited for the job. It must be balanced and run true in order to obtain a smooth, accurately sized job.

Use a diamond wheel dresser, consisting of an industrial diamond tip mounted on a steel shank, for the truing operation, **Figure 15-31**. Mount it solidly to the lathe, on center or slightly below the center of the grinding wheel. Move the rotating wheel back and forth across the diamond, removing about 0.001″ (0.02 mm) on each pass. Remove only enough material to true the wheel.

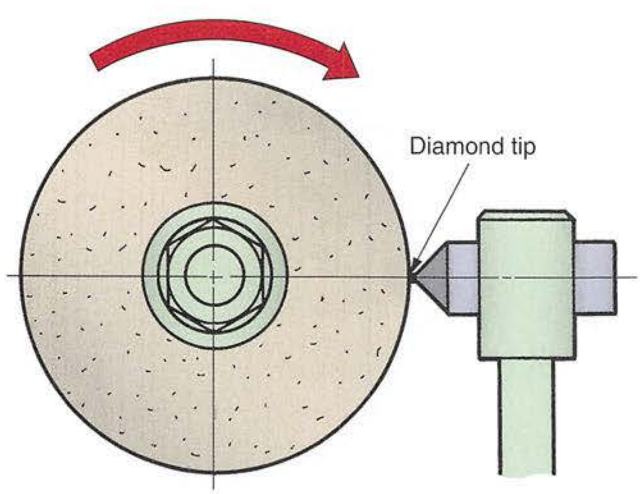
15.8.3 External Grinding

External grinding, **Figure 15-32**, is done to finish the exterior surface of the piece. The following steps are recommended to complete the job with the least amount of difficulty:

1. Mount the work solidly in the lathe. Provide adequate clearance.
2. Adjust lathe spindle speed to 80–100 rpm, and set a feed of 0.005″–0.007″ (0.12 mm–0.17 mm).
3. Turn on power for lathe and grinder. The work turns into the grinding wheel, **Figure 15-33**.
4. Feed the grinding wheel into the work until it just begins to "spark."
5. Engage the automatic longitudinal feed.
6. Check work diameter frequently with a micrometer. Use light cuts to avoid overheating and warping the workpiece.
7. Dress the grinding wheel again before making the final pass over the work. Allow the work to "spark out" (reach the point where the grinding wheel no longer cuts).

15.8.4 Internal Grinding

Internal grinding is done in much the same manner as external grinding but on the inside of the work, **Figure 15-34**. The work and grinding wheel must rotate in opposite directions, **Figure 15-35**. Because the quill (shaft for mounting the grinding wheel for internal work) is slender, use very light cuts and slow feeds to prevent the hole from "bell-mouthing," as shown in **Figure 15-36**. For the same reason, it is suggested that the grinding wheel be allowed to "spark out" on the last cut.

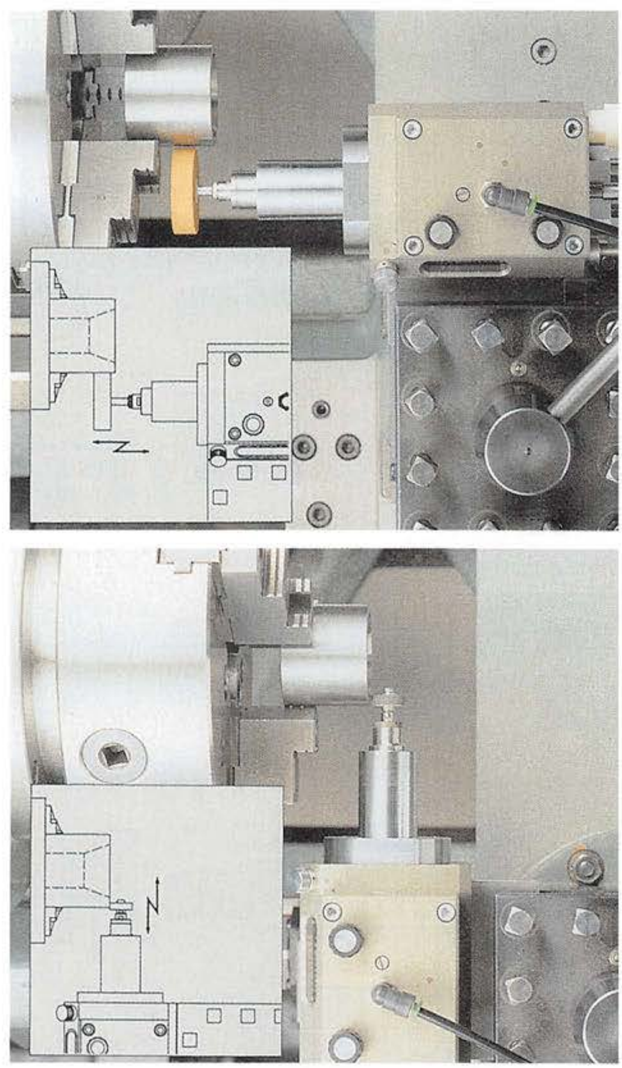


Goodheart-Willcox Publisher

Figure 15-31. Using a diamond wheel dresser to true the wheel on a tool post grinder before grinding on a lathe.

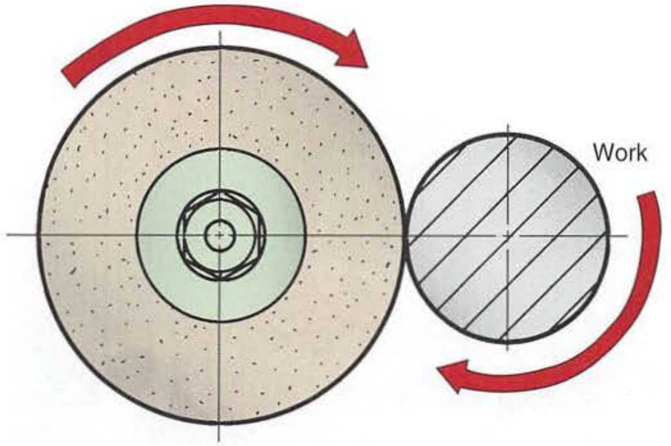
A

B



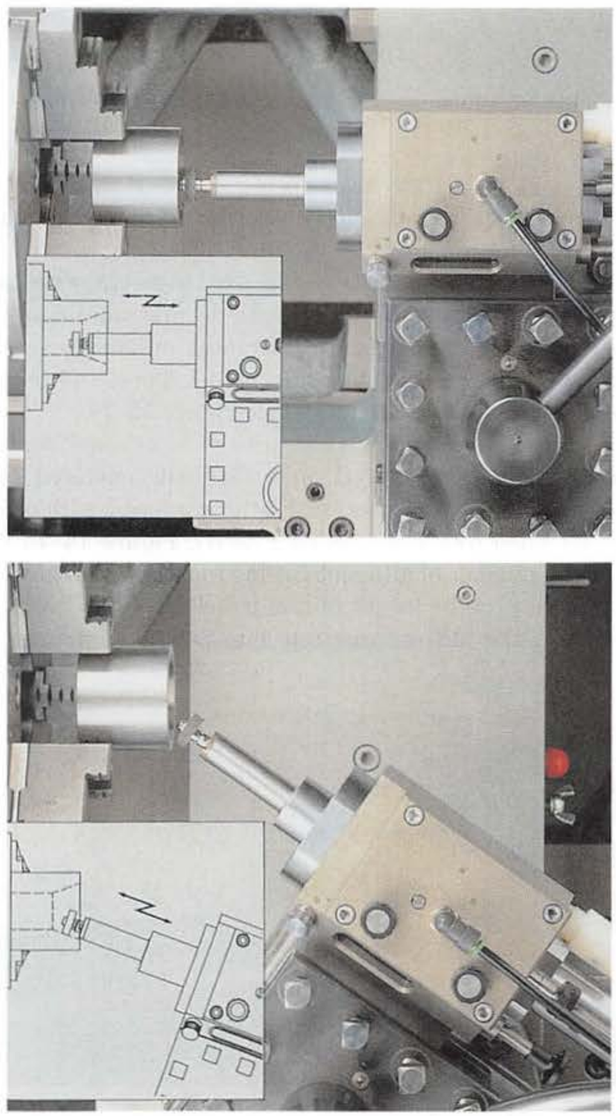
NSK America

Figure 15-32. External grinding on a lathe. A—Grinding the circumference of a workpiece. B—Grinding the end of a workpiece.



Goodheart-Willcox Publisher

Figure 15-33. For external grinding, the work turns into the grinding wheel.

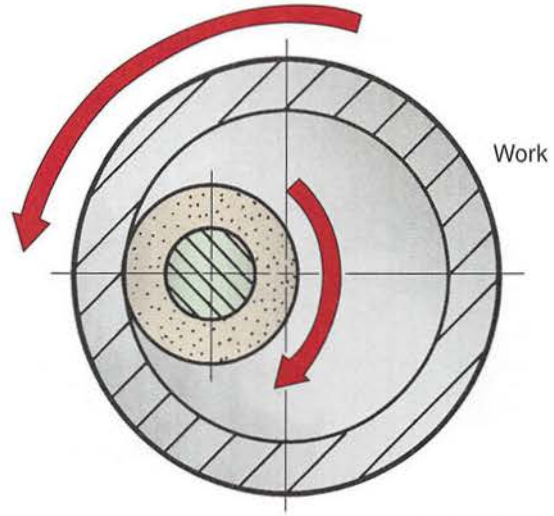


A

B

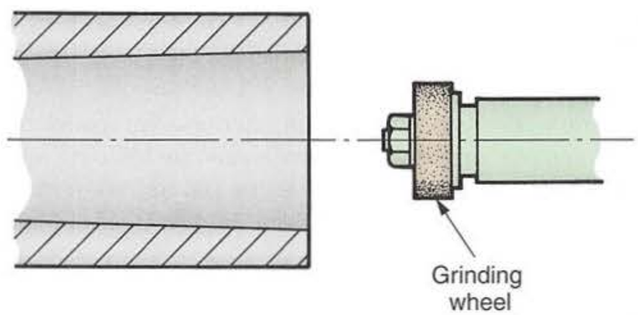
NSK America

Figure 15-34. Internal grinding operations on the lathe. A—Internal grinding. B—Taper grinding.



Goodheart-Willcox Publisher

Figure 15-35. For internal grinding, the work and grinding wheel turn in opposite directions.



Goodheart-Willcox Publisher

Figure 15-36. Bell-mouthing (grinding a hole larger at its mouth) is caused by taking too deep a cut with the grinder or by grinding with a feed that is too rapid.

SAFETY NOTE

When cleaning the lathe after grinding, use a brush to sweep particles off the lathe. Never use your hand or compressed air. Using compressed air can force particles into the joints of the machine and cause premature wear.

15.9 Other Lathe Attachments

Other attachments for the lathe further extend its capabilities. Among these are milling attachments and duplicating or tracing units.

15.9.1 Milling on a Lathe

Some lathes can be fitted with a vertical milling attachment, **Figure 15-37**. Such machines are designed primarily

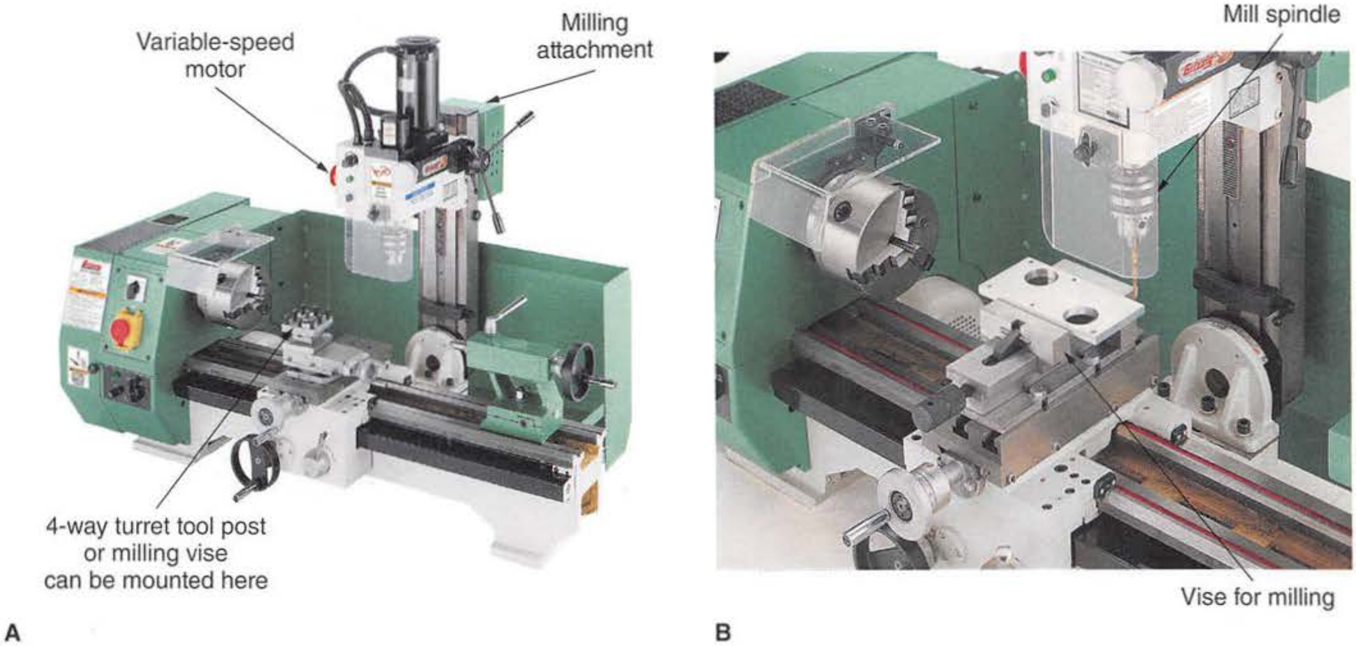


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 15-37. Milling attachments on a lathe. A—A lathe with a milling attachment mounted on the back of the bed. B—Work-holding devices are fitted on the cross-slide.

for home workshops, but they are often used in model and experimental shops as well. A vise is mounted to the cross-slide, which also provides traverse (in-and-out) movement. Longitudinal (back-and-forth) feed is furnished by the carriage.

A horizontal milling attachment is available for some lathes that permits limited milling operations to be performed. The cutter is mounted on an arbor or fitted into the headstock. Cutter depth is controlled by the adjusting screw on the device. Cutter movement is controlled by carriage and cross-slide movements.

15.9.2 Duplicating and Tracing

A tracing or duplicating unit can be used when several identical pieces must be produced. Several types are available. One type uses flat templates. Another type uses a three-dimensional template or pattern. Most units are hydraulically operated.

The duplicating unit improves quality because each part is an exact duplicate of the master template. However, computer-controlled lathes now do the work formerly done by these units.

15.10 Industrial Applications of the Lathe

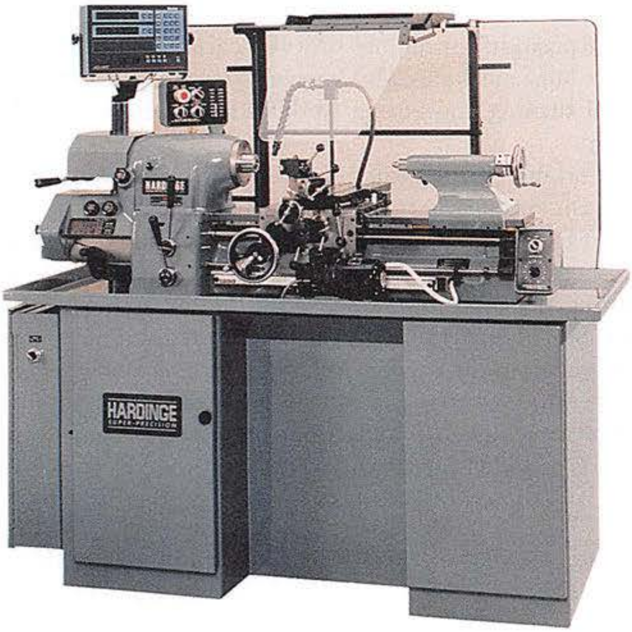
Industry makes wide use of variations of the basic lathe. Conventional metalworking lathes are manufactured in a large range of sizes from the tiny jeweler's lathe to large machines that turn forming rolls for the steel industry.

Heavy-duty engine lathes are used to machine very large workpieces, **Figure 15-38**. A highly accurate toolroom lathe is required to meet the close tolerances and fine surface finish specifications of toolrooms, model shops, and research and development laboratories. See **Figure 15-39**.

Limited production runs (usually less than 250 pieces) are sometimes produced on a manually operated ***turret lathe***. This is a conventional lathe equipped with a four- to six-sided toolholder called a *turret*. **Figure 15-40** illustrates a number of different cutting tools fitted to the turret. Stops control the length of tool travel and rotate the turret to bring the next cutting tool into position automatically.

ID1974/Shutterstock.com

Figure 15-38. A heavy-duty engine lathe on display at a military technologies exposition.



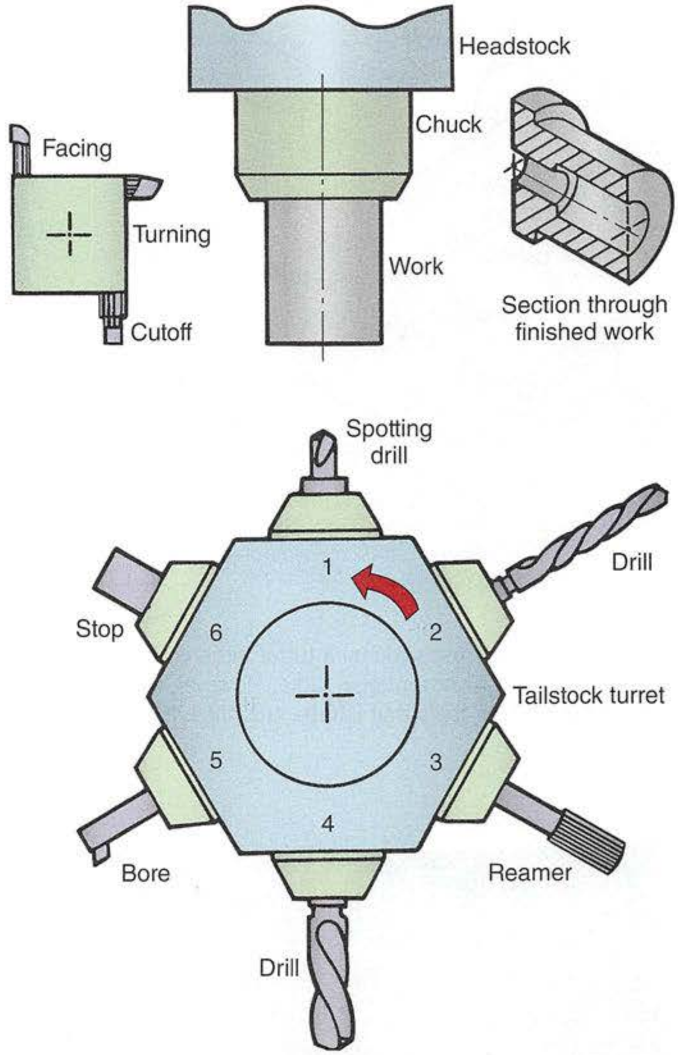
Hardinge Super Precision HLV-DR is a registered trademark of Hardinge, Inc.

Figure 15-39. The Hardinge Super Precision HLV-DR toolroom lathe.

A cross-slide unit is fitted for turning, facing, forming, and cutoff operations, **Figure 15-41**.

The ***automatic screw machine*** is a variation of the lathe that was developed for high-speed production of large numbers of small parts. The machine performs a large number of operations either simultaneously or in a very rapid sequence.

Increasingly, industry is relying on automatic turning centers to produce tiny precision parts in quantity. These centers, referred to as "Swiss-type" machines because they were originally used in the Swiss watchmaking industry, use computer control to perform a number of operations in sequence, producing a finished part. See **Figure 15-42**.



Goodheart-Willcox Publisher

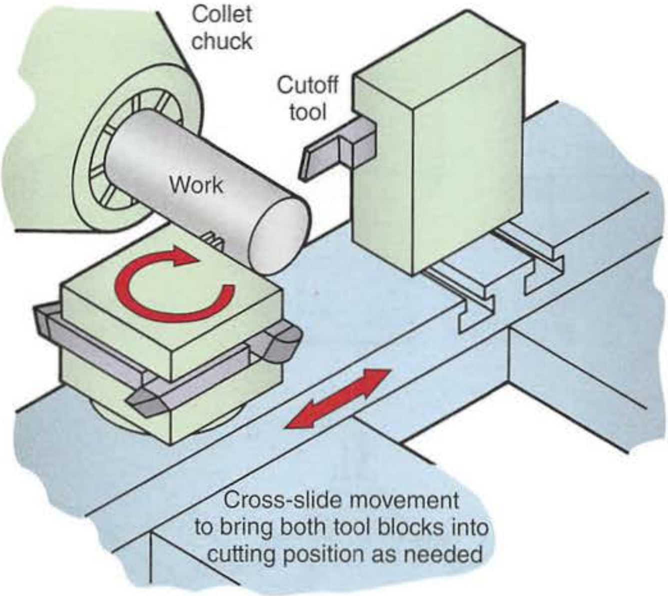
Figure 15-40. The turret in relation to other parts of a lathe. The turret rotates to bring each cutting tool into position. Stops control the depth of tool cuts.

WORKPLACE SKILLS

Creativity and Brainstorming

The ability to "think outside the box" to come up with workable design solutions is an important skill for machinists, machine designers, and most other professionals involved in machining. Creativity is therefore an important employability skill.

Some people are creative by nature. Even if you are not one of these people, you can learn to be more creative. One method is to practice *brainstorming*. Choose an issue that interests you—machining related or not—and write down as many solutions as you can think of. Do not worry at first about whether your solutions are probable or even possible. There are no right or wrong answers when brainstorming. Just list everything that comes into your head. Give yourself about 10 or 15 minutes to create the list. Then go back over your list and evaluate all of your ideas. By practicing brainstorming, you will become a more creative thinker.



Goodheart-Willcox Publisher

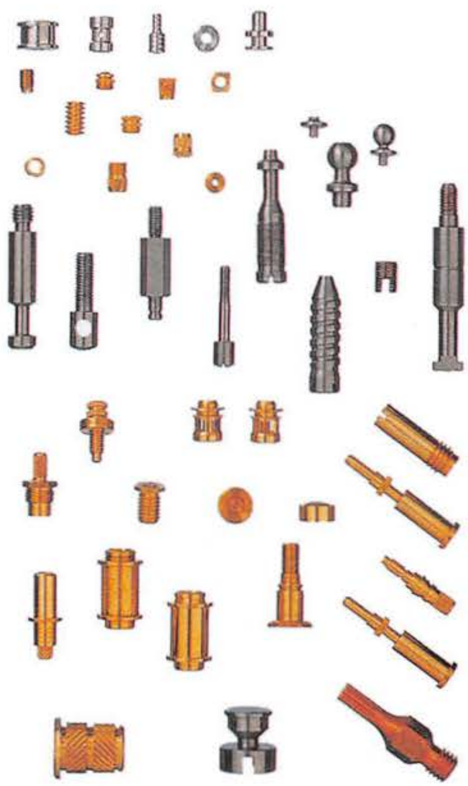
Figure 15-41. The cross-slide on a turret lathe is similar to the cross-slide on a conventional lathe. However, it is fitted with several cutting tools that can be brought into position as needed.

Work that is too large or too heavy to be turned in a horizontal position is machined on a vertical lathe, **Figure 15-43**. These huge machines, also known as ***boring mills***, are capable of turning and boring work with diameters up to 40′ (12 m).

Portable turning equipment is available for work in the field. For example, portable equipment is often used to chamfer the ends of large pipe prior to welding. See **Figure 15-44**.

Computer numerically controlled (CNC) lathes and turning machines are widely used for industrial production. See **Figure 15-45**. With proper programming, these machine tools are capable of producing complex work with great accuracy and repeatability. A more detailed description of CNC machine tools and automated manufacturing operations is provided later in this book.

A



B

Tornos-Bechler S.A.

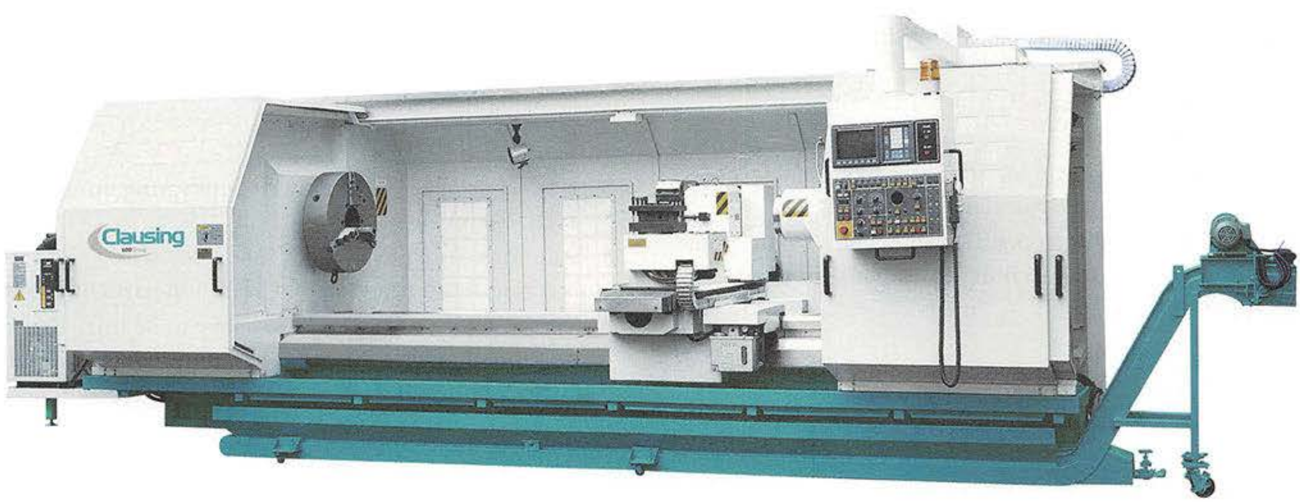
Figure 15-42. Swiss-type automatic turning center. A—This high-precision turning center machines small parts from bar stock ranging from 1 mm to 10 mm in diameter at high production rates. B—These tiny components, produced by a Swiss-type automatic turning center, are used in precision instruments.

AMT—The Association for Manufacturing Technology

Figure 15-43. A large boring mill. Workpieces are mounted on the large turntable and rotated into position for machining.

Tri-Tool, Inc.

Figure 15-44. A portable lathe that can be taken into the field. This worker is shown turning the end of a high-pressure gas pipeline to prepare it for being welded.



Clausing Industrial, Inc.

Figure 15-45. A large industrial CNC lathe that can remove a large amount of stock quickly.

Chapter Review

Summary

- Boring is a machining operation performed on a lathe to enlarge a hole.
- Drilling can be accomplished on a lathe by feeding a stationary drill into the revolving workpiece.
- Reaming is used to make a hole accurate in diameter and finish.
- Knurling forms horizontal or diamond-shaped serrations on the circumference of the work.
- Filing and polishing can be performed on a lathe to remove burrs, round off sharp edges, and blend in form-cut lines.
- Steady rests and follower rests are work-holding devices that offer additional support to securely hold long, thin work during lathe operations.
- Mandrels allow an outside diameter to be machined concentric with an inside diameter; some mandrels also allow multiple workpieces to be machined at once.
- A tool post grinder allows the lathe to be used for both internal and external grinding.
- Vertical milling attachments and tracing or duplicating units are also available to extend the capability of the lathe.
- Industrial applications of the lathe make use of several lathe attachments and variations of the basic lathe machine.

Review Questions

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

1. Boring is a(n) _____.
   A. drilling operation
   B. internal machining operation in which a single-point cutting tool is used to enlarge a hole
   C. external machining operation in which a single-point cutting tool is used to reduce the diameter of a hole
   D. All of the above.
   E. None of the above.
2. Drills that are used on the lathe are fitted with _____ shanks.
   A. straight
   B. taper
   C. fluted
   D. Both A and B.
   E. Both B and C.
3. What is the purpose of reaming?
4. The process of forming horizontal or diamond-shaped serrations on the circumference of the work is called _____.
5. Knurling is commonly done to _____.
   A. provide a gripping surface
   B. change the appearance of the work
   C. increase the work's diameter
   D. All of the above.
   E. None of the above.
6. For what three purposes is filing usually done on the lathe?
7. Polishing can be performed on the lathe using a strip of _____.
8. When is a steady rest used?
9. There are times when a shaft is unsuitable as a bearing surface and cannot be used with a steady rest. When this occurs, a(n) _____ can be used so the shaft can be supported with the steady rest.
10. What is the difference between a steady rest and a follower rest?
11. What is a mandrel and when is it used?
12. A mandrel is usually pressed into the work with a(n) _____.
13. Internal and external grinding can be done on a lathe with a(n) _____.
14. What should be done to protect the lathe from the abrasive dust and grit created during grinding operations?
    A. Use a nonabrasive grinding wheel.
    B. Cover the bed and moving parts with a heavy cloth.
    C. Use a soft abrasive grinding wheel.
    D. All of the above.
    E. None of the above.
15. Name two types of tracing or duplicating units available for the lathe.
16. Automatic turning centers, also known as _____ machines, can produce tiny precision parts in quantity.
17. Work that is too large or too heavy to be turned in the horizontal position is turned on huge machines called vertical lathes or _____.

CHAPTER 16

Cutting Tapers and Screw Threads on the Lathe

Chapter Outline

16.1 Turning Tapers
- **16.1.1** Turning Tapers with a Compound Rest
- **16.1.2** Turning Tapers by the Offset Tailstock Method
- **16.1.3** Turning a Taper with a Taper Attachment
- **16.1.4** Turning a Taper with a Square-Nose Tool

16.2 Measuring Tapers
- **16.2.1** Measuring Tapers by Comparison
- **16.2.2** Measuring Tapers Directly

16.3 Cutting Screw Threads on the Lathe
- **16.3.1** Screw Thread Forms
- **16.3.2** Preparing to Cut 60° Threads on a Lathe
- **16.3.3** Making the Cut
- **16.3.4** Resetting a Tool in the Thread
- **16.3.5** Cutting Threads with Threading Inserts
- **16.3.6** Measuring Threads
- **16.3.7** Cutting Left-Hand Threads
- **16.3.8** Cutting Square Threads
- **16.3.9** Cutting Acme Threads
- **16.3.10** Thread Rolling
- **16.3.11** Cutting Internal Threads
- **16.3.12** Cutting Threads on a Tapered Surface

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.
- Explain two methods for measuring tapers.
- Describe the various forms of screw threads.
- Cut screw threads on a lathe.

Technical Terms

external thread
internal thread
lead
major diameter
minor diameter
offset tailstock method
pitch
pitch diameter
setover
taper
taper attachment
threadcutting stop
thread dial
thread rolling
three-wire method

A variety of methods and tools can be used to turn tapers and cut threads on a lathe. Always measure tapers and threads accurately to ensure high-quality work.

16.1 Turning Tapers

A *taper* is a slope that increases or decreases at a uniform rate. Bell-shaped pieces are not tapered. See **Figure 16-1**. A cone is an example of a taper. The "wedging" action of a taper makes it ideal for driving drills, milling arbors, end mills, and centers. In addition, it will automatically align itself in a similarly tapered hole. The rate of taper can be stated as taper per inch, taper per foot, or millimeters per 25 mm of length. It can also be stated in degrees or as a ratio, **Figure 16-2**.

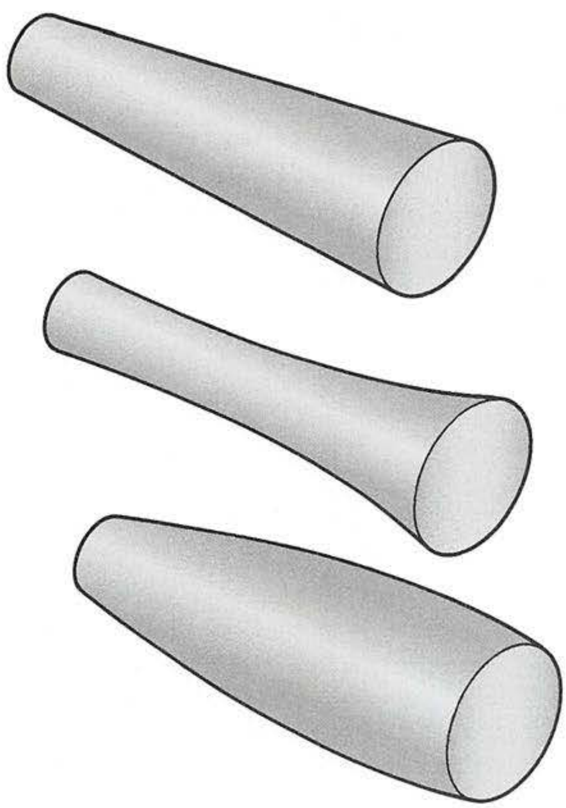
There are five principal methods of machining tapers on a lathe. Each has its advantages and disadvantages. The five methods, as listed in **Figure 16-3**, are as follows:

- The compound rest method.
- The offset tailstock method.
- Using a taper attachment.
- Using a square-nose tool.
- Using a reamer.

A

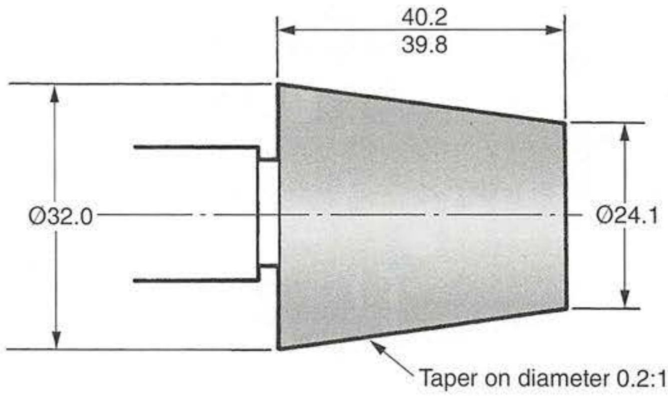
B

C



Goodheart-Willcox Publisher

Figure 16-1. 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.



Goodheart-Willcox Publisher

Figure 16-2. Taper may be stated as a ratio (0.2:1 in this example), in degrees, or as taper per inch, taper per foot, degrees, or millimeters per 25 mm.

Ways of Machining Tapers

Method	Advantages and Disadvantages	Information Needed
Compound rest	Length of taper limited. Cuts external and internal taper.	Must know the taper angle.
Offset tailstock	External taper only. Must work between centers.	Taper per inch or taper per foot.
Taper attachment	Best method to use.	Angle or taper per inch or foot.
Square-nose tool	Very short taper.	Taper angle.
Reamer	Internal only.	Taper number.

Goodheart-Willcox Publisher

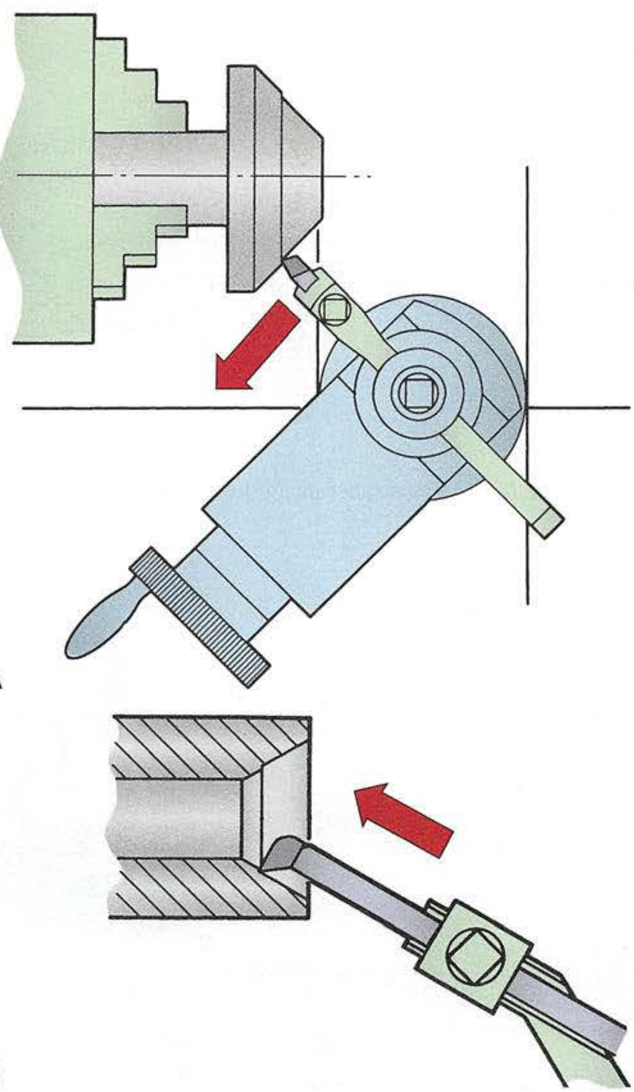
Figure 16-3. Methods by which tapers can be turned on a lathe.

16.1.1 Turning Tapers with a 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 16-4**. Taper length is limited, however, by the movement of the compound rest. 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.

Because the compound rest base is graduated in degrees, **Figure 16-5**, the taper must be converted to degrees. A conversion table may be used. See **Figure 16-6**.

A careful study of the print will show whether the angle given is from center or is the included angle. **Figure 16-7** 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.

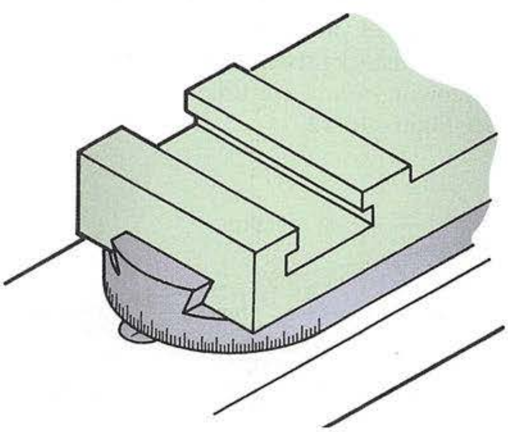


A

B

Goodheart-Willcox Publisher

Figure 16-4. Cutting tapers using the compound rest. A—External taper. The cut is being made from small diameter to large diameter. B—Internal taper being turned with the compound rest.



Goodheart-Willcox Publisher

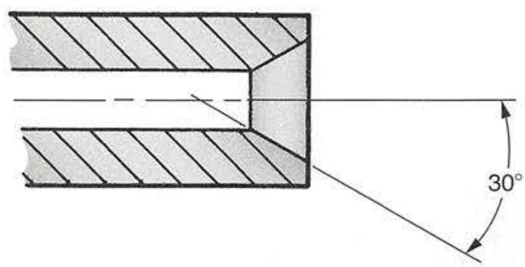
Figure 16-5. The base of the compound rest is marked in degrees to aid in precise positioning.

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″

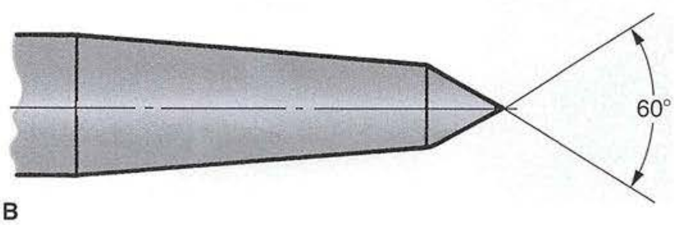
Goodheart-Willcox Publisher

Figure 16-6. You can use this table to convert taper per foot into corresponding angles to adjust the compound rest.

A



B



Goodheart-Willcox Publisher

Figure 16-7. The two methods used to measure angles. A—Angle measured from the centerline of the workpiece. B—Measurement of the included angle.

With the lathe's centerline representing 0°, pivot the compound rest to the desired angle and lock it in position. The usual practice is to turn a taper from the smaller diameter to the larger diameter. Refer to **Figure 16-4A**. The cutting tool must be set on exact center to cut a taper. Select a toolholder that provides ample clearance.

To machine the taper, bring the cutting tool into position with the work and lock the carriage to prevent it from shifting during the turning operation. See **Figure 16-8**. Since there is no power feed for the compound rest, the cutting tool must be fed evenly with both hands to achieve a smooth finish. Make the entire cut without stopping the cutting tool. Then move the compound rest back to the starting point and position it with the cross-slide for the next cut.

16.1.2 Turning Tapers by the Offset Tailstock Method

The ***offset tailstock method,*** also known as the *tailstock setover method,* is also used for turning tapers, **Figure 16-9**. Jobs that can be turned between centers may be taper-turned by this technique. Only external tapers can be machined in this way, however.

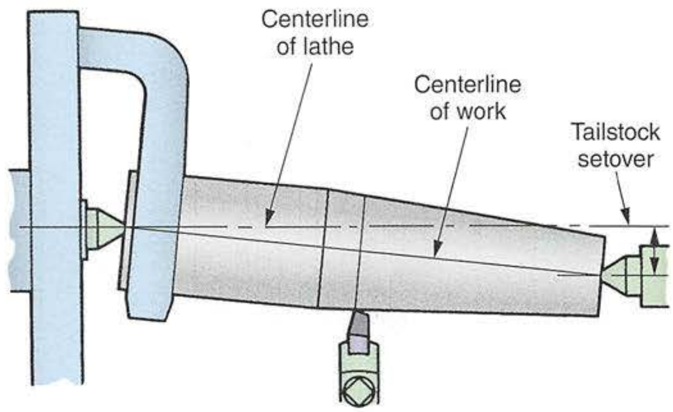
Most lathe tailstocks consist of two parts, allowing the upper portion to be shifted off-center. This movement, referred to as ***setover,*** is accomplished by loosening the anchor bolt that locks the tailstock to the ways, then making the proper adjustments with screws on the tailstock, **Figure 16-10**. After the setover has been made, the screws are drawn up snug, but not tight.

Calculating Tailstock Setover

Turning tapers 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.

Goodheart-Willcox Publisher

Figure 16-8. Check to be sure there is plenty of clearance between the chuck and the compound rest, then lock the carriage.



Goodheart-Willcox Publisher

Figure 16-9. Machining a taper using the offset tailstock method.

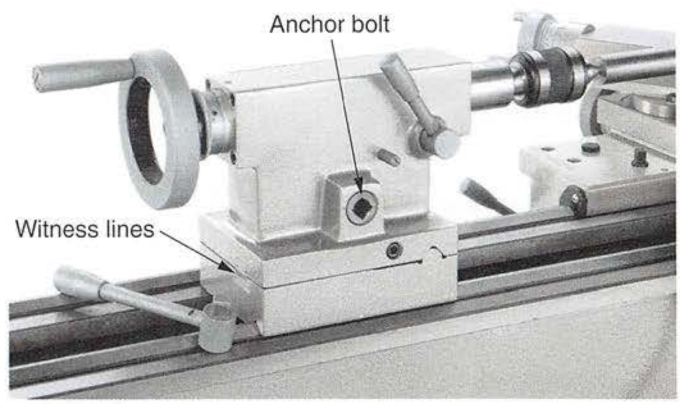
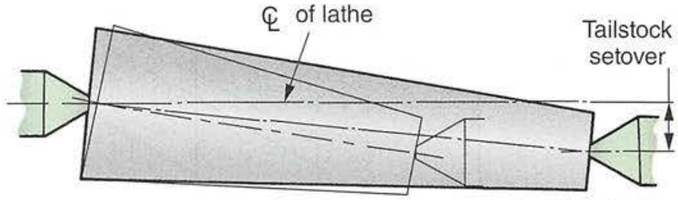


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 16-10. 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.

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 16-11**.

The following terms are used in calculating tailstock setover. See **Figure 16-12**.



Goodheart-Willcox Publisher

Figure 16-11. The length of the work causes the taper to vary even though the tailstock offset remains the same.

D = Diameter at large end of taper

d = Diameter at small end of taper

l = Length of taper

L = Total length of piece

TPI = Taper per inch

TPF = Taper per foot

When taper per inch (TPI) and total length of the piece (L) are known, setover can be calculated using the following formula:

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

Example: What will be the tailstock setover for the following job?

Taper per inch = 0.125

Total length of piece = 8.000

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

$$= \frac{8.000 \times 0.125}{2}$$

$$= 0.500''$$

Note: The same procedure is followed when using metric units. However, all dimensions are in millimeters.

When taper per foot (TPF) is known, it must be converted to taper per inch (TPI). The following formula takes this into account:

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

Plans often do not specify TPI or TPF, but they do give other pertinent information. Setover can be calculated using the dimensions of the tapered section, even if TPI or TPF is not stated. Calculations will be easier if all fractions are converted to decimals. All dimensions must either be in inches or in millimeters (not a mixture of both).

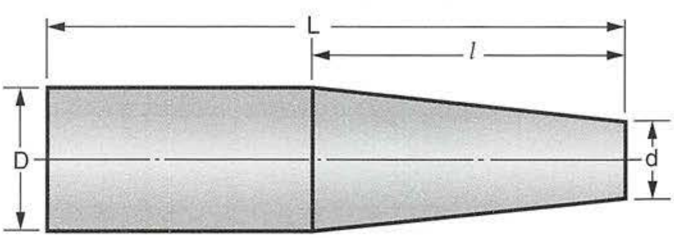


Figure 16-12. Basic taper information. D = diameter at large end of taper; d = diameter at small end of taper; l = length of taper; L = total length of piece.

$$\text{Offset} = \frac{L \times (D - d)}{2 \times l}$$

Where:

D = Diameter at large end of taper

d = Diameter at small end of taper

l = Length of taper

L = Total length of piece

Example: Calculate the tailstock setover for the following job.

D = 1.250″

d = 0.875″

l = 3.000″

L = 9.000″

$$\text{Offset} = \frac{L \times (D - d)}{2 \times l}$$

$$= \frac{9.000 \times (1.250 - 0.875)}{2 \times 3.000}$$

$$= \frac{9.000 \times 0.375}{6}$$

$$= 0.562''$$

Basic trigonometry, which is beyond the scope of this textbook, is necessary to calculate setover when a taper is given in degrees. However, any good machinist's handbook will provide this information. At least one such book should be part of every machinist's toolbox.

Measuring Tailstock Setover

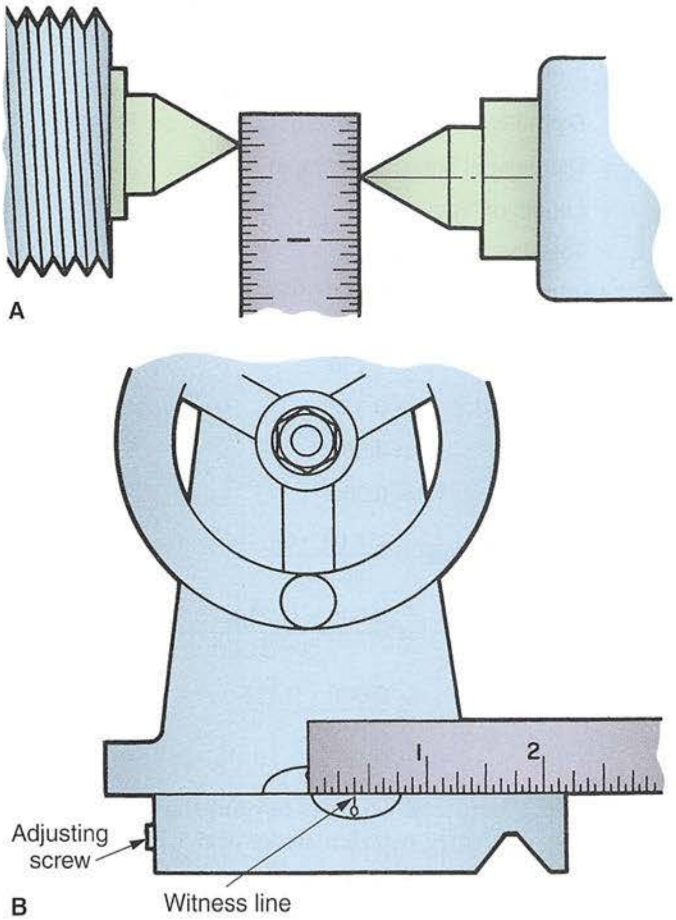
When an ample tolerance (±0.015″ or 0.05 mm) is allowed, the setover can be measured with a steel rule. There are two ways to measure, as follows:

- Place a rule that has graduations on both edges between the center points, **Figure 16-13A**. Measure the distance between the center points.
- Measure the distance between the two witness lines on the tailstock base, **Figure 16-13B**.

Accurate work requires care in making the tailstock setover. An additional factor enters into the calculations—the distance at which the center point enters the piece. Typically, 1/4″ (6.5 mm) is an ample allowance. This distance must be subtracted from the total length of the piece.

Use the appropriate method to calculate the offset. To make a precise setover, use the micrometer collar on the lathe cross-slide. See **Figure 16-14**.

1. Clamp the toolholder in a reverse position in the tool post.
2. Turn the cross-slide screw back to remove all play.



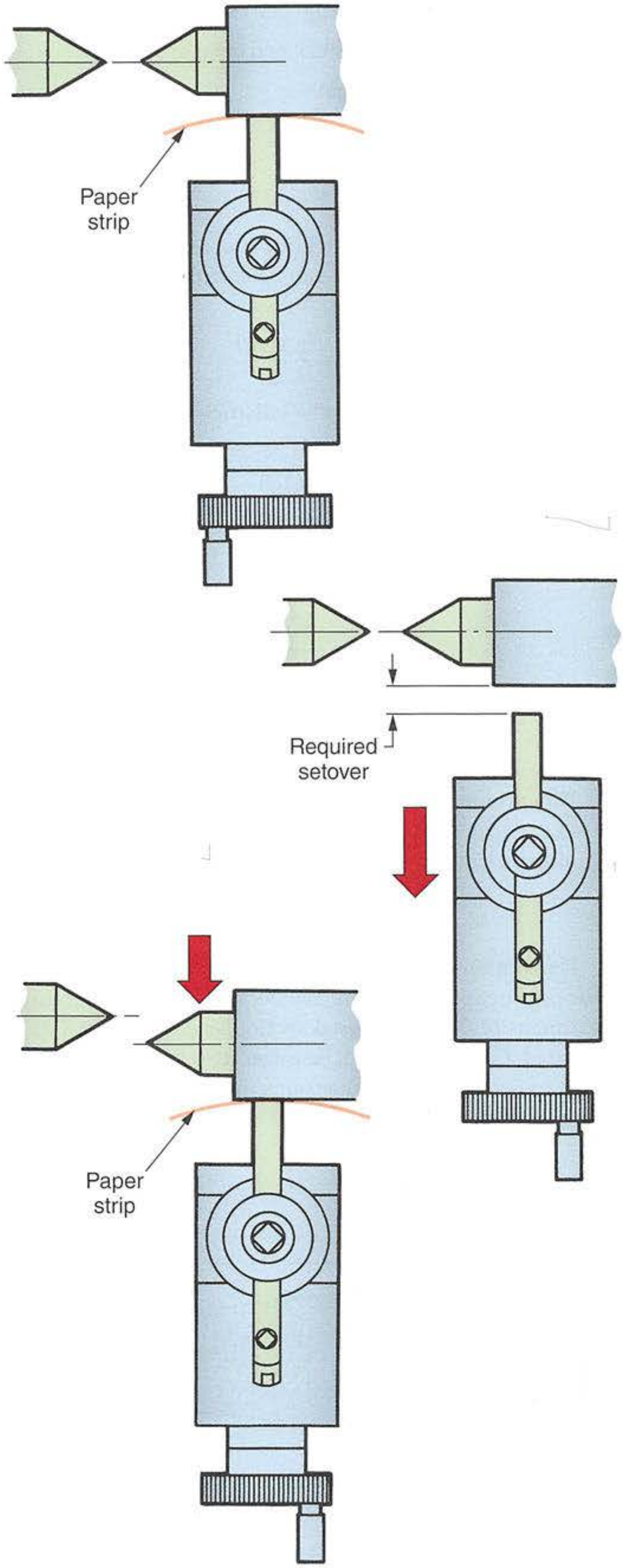
Goodheart-Willcox Publisher

Figure 16-13. 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.

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.

Instead of the toolholder and paper strip, a dial indicator can be used to establish the offset. See **Figure 16-15**.

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.



Goodheart-Willcox Publisher

Figure 16-14. Using the micrometer collar of the compound rest to make the setover measurement.

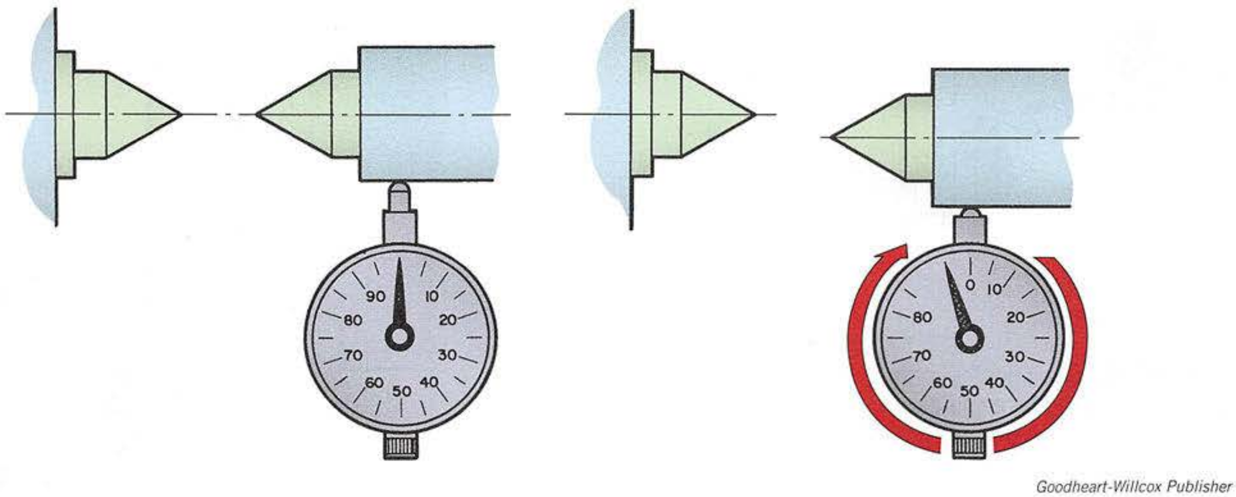


Figure 16-15. A dial indicator can also be used to measure amount of setover.

Cutting a Taper

When you cut a taper, additional strain is placed 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 during conventional turning between centers. It must be checked frequently for binding. A bell-type center drill reduces the strain somewhat. Some machinists prefer a center with a ball tip to produce an improved bearing surface. See **Figure 16-16**.

Make the cuts as in conventional turning. However, start the cut at the small end of the taper.

16.1.3 Turning a Taper with a Taper Attachment

A ***taper attachment*** is a guide that can be attached to most lathes. It provides 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 ensure 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 with the included angle of the taper in degrees.

The lathe does not have to be altered to work with and without the taper attachment. The machine can be used for straight turning by locking out the taper attachment. No realignment of the lathe is necessary.

Types of Taper Attachments

There are two types of taper attachments: plain and telescopic. See **Figure 16-17**. The plain taper attachment requires disengaging the cross-slide screw from the cross-slide feed nut. The cutting tool is advanced using the compound rest feed screw.

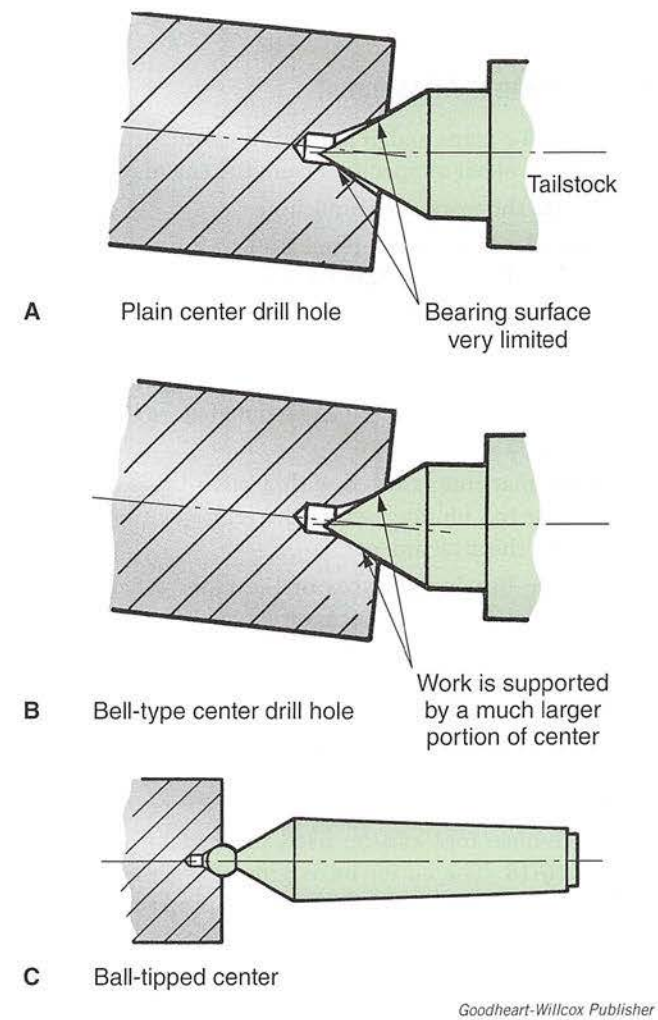


Figure 16-16. Taper turning done by the offset tailstock method is hard on the tailstock center. A—The center point does not bear evenly in a conventional center hole. B—A bell-type center drill reduces the problem by providing more bearing surface. C—A ball-tipped center lessens pressure on the tail center when turning tapers.

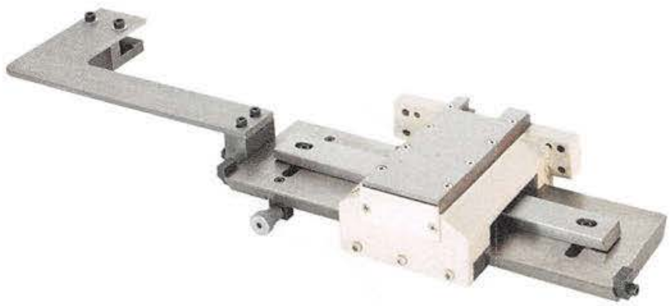


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 16-17. This telescopic taper attachment is fitted to the rear of the carriage and permits taper turning without disengaging the cross-slide feed nut.

The telescopic taper attachment does not require the cross-slide feed nut to be disconnected. The tool can be advanced into the work with the cross-slide screw in its usual place.

Setting a Taper Attachment

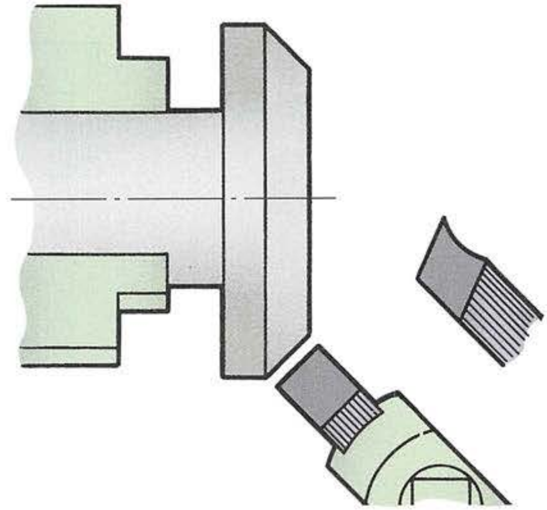
1. Study the plans and, if necessary, calculate the taper. Set the swivel bar as specified from the calculations.
2. Mount the work in the machine.
3. Slide the taper attachment unit to a position that will permit the cutting tool to travel the full length of the taper. Lock it to the ways.
4. Move the 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.

16.1.4 Turning a Taper with a Square-Nose Tool

A square-nose tool can be used to produce short tapers, **Figure 16-18**. 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.

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 the centers be "zeroed in" (put in perfect alignment). Then the necessary adjustments (tailstock setover, taper attachment adjustment) can be made.



Goodheart-Willcox Publisher

Figure 16-18. A short taper can be turned with a square-nose tool.

16.2 Measuring Tapers

There are two basic methods of testing the accuracy of machined tapers. They can be measured by comparison, or they can be measured directly.

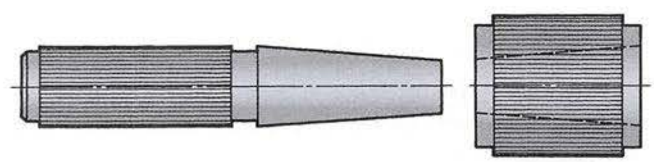
16.2.1 Measuring Tapers by Comparison

Taper plug gages and taper ring gages, **Figure 16-19**, serve two purposes. They measure the basic diameter of the taper as well as the angle of slope. To check the angle, apply layout dye to the machined surface or plug gage. Then insert the blued section into the mating part and slowly rotate it. If the layout dye rubs off evenly, the taper is correct. If the layout dye rubs off unevenly, **Figure 16-20**, the remaining material shows where the taper is incorrect and indicates the necessary machine adjustments.

Some gages also have notches to indicate the specified tolerance in taper diameter. The indentations show the go and no-go limits, **Figure 16-21**.

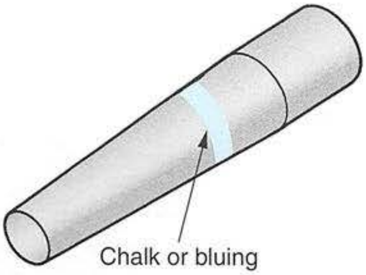
16.2.2 Measuring Tapers Directly

A taper test gage is used to check taper accuracy directly, **Figure 16-22**. 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



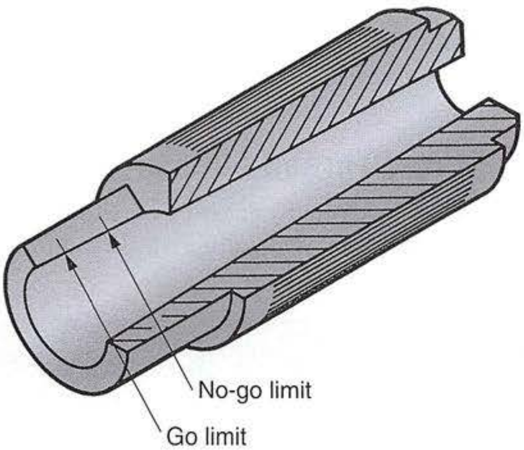
Goodheart-Willcox Publisher

Figure 16-19. Left—Plug gage. Right—Ring gage.



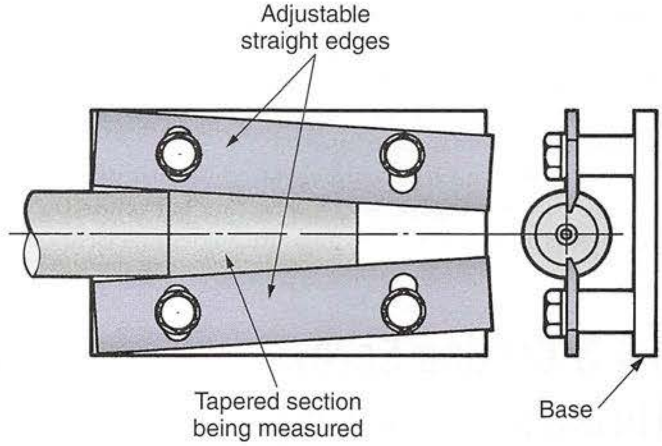
Goodheart-Willcox Publisher

Figure 16-20. When chalk or bluing does not rub off evenly, it indicates that the taper does not fit properly and additional machine adjustments need to be made.



Goodheart-Willcox Publisher

Figure 16-21. Typical go and no-go ring gage for measuring tapers.



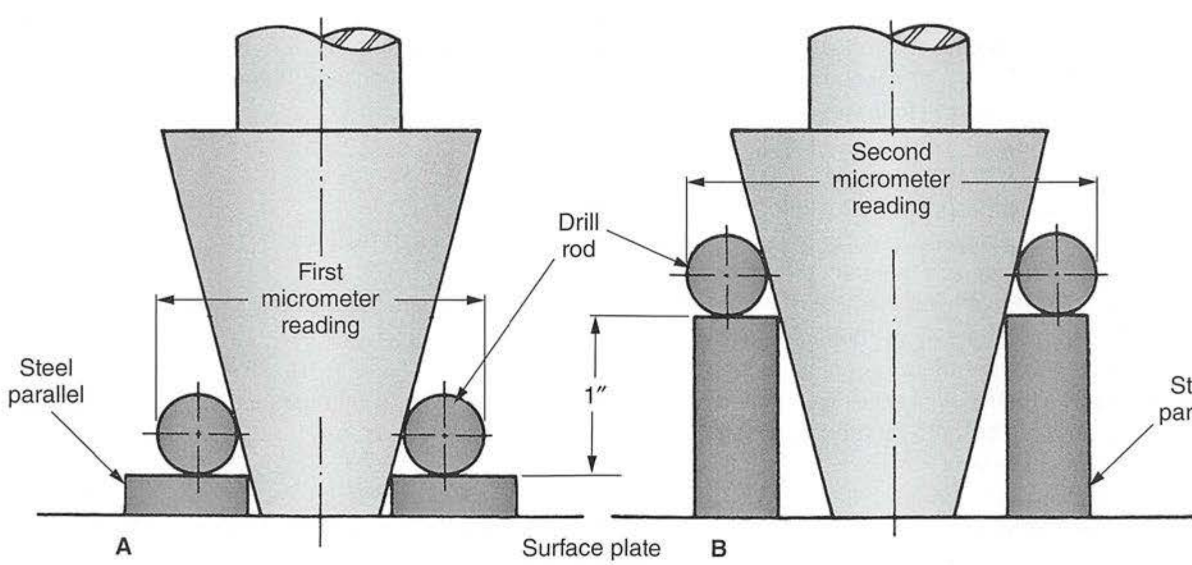
Goodheart-Willcox Publisher

Figure 16-22. A taper test gage can be set for different tapers.

by using two discs of known size that are located the correct distance apart.

Another technique for measuring tapers is to place 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 16-23**. 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 for the second reading. The rods are the same diameter as those used to make the first reading. After the second reading is made,

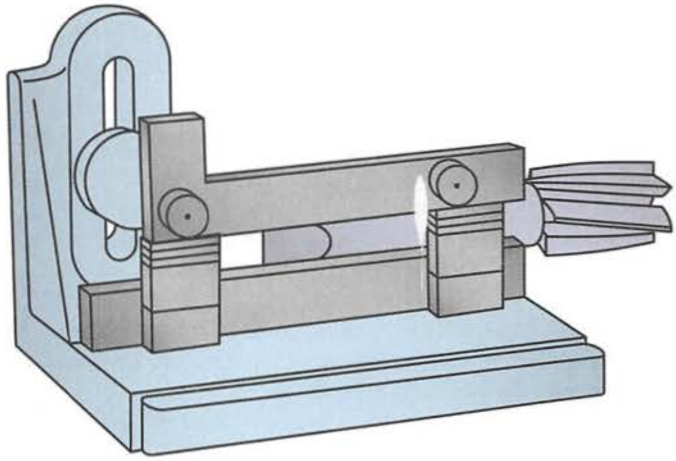


Goodheart-Willcox Publisher

Figure 16-23. Measuring a taper using parallels, drill rod, micrometer, and surface plate. A—Setup for first measurement. B—Setup for second measurement.

Figure 16-23B, the taper per foot can be determined. First, subtract to find the difference between the two measurements. Then multiply the difference 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 16-24**. The sine bar is used with gage blocks and sine tables to measure angles precisely.



Goodheart-Willcox Publisher

Figure 16-24. A sine bar and precision gage blocks can also be used to measure a taper.

16.3 Cutting Screw Threads on the Lathe

Screw threads are used for many applications. Important applications include:

- 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).

16.3.1 Screw Thread Forms

The first screw threads cut by machine were square in cross-section. Since that time, many different thread forms have

WORKPLACE SKILLS

Teamwork

Employers seek employees who can effectively serve as good team members. Due to the nature of most work today, teamwork is necessary. A team is a group of people working together for a common purpose. Often, cooperation requires flexibility and willingness to try new ways to get things done. If someone is uncooperative, it takes longer to accomplish the tasks. When people do not get along, strained relationships may occur, which also gets in the way of finishing the tasks.

You will be more desirable as an employee if you know how to be a team player. A big advantage of a team is its ability to develop plans and complete work faster than individuals working alone. In contrast, a team usually takes longer to reach a decision than an individual worker does. Team members need some time before they become comfortable with one another and function as a unit.

In teams, there is a protocol, or set of unspoken rules, that dictates behavior. Team protocol suggests that teams implement the following behaviors:

- State a clear unity of purpose.
- Have a clear set of performance goals.
- Create an atmosphere that is informal, comfortable, and relaxed.
- Encourage everyone to participate and feel free to express ideas and feelings.
- Lead members to a general consensus through discussion.

To be an effective team member, team protocol suggests that members practice the following behaviors:

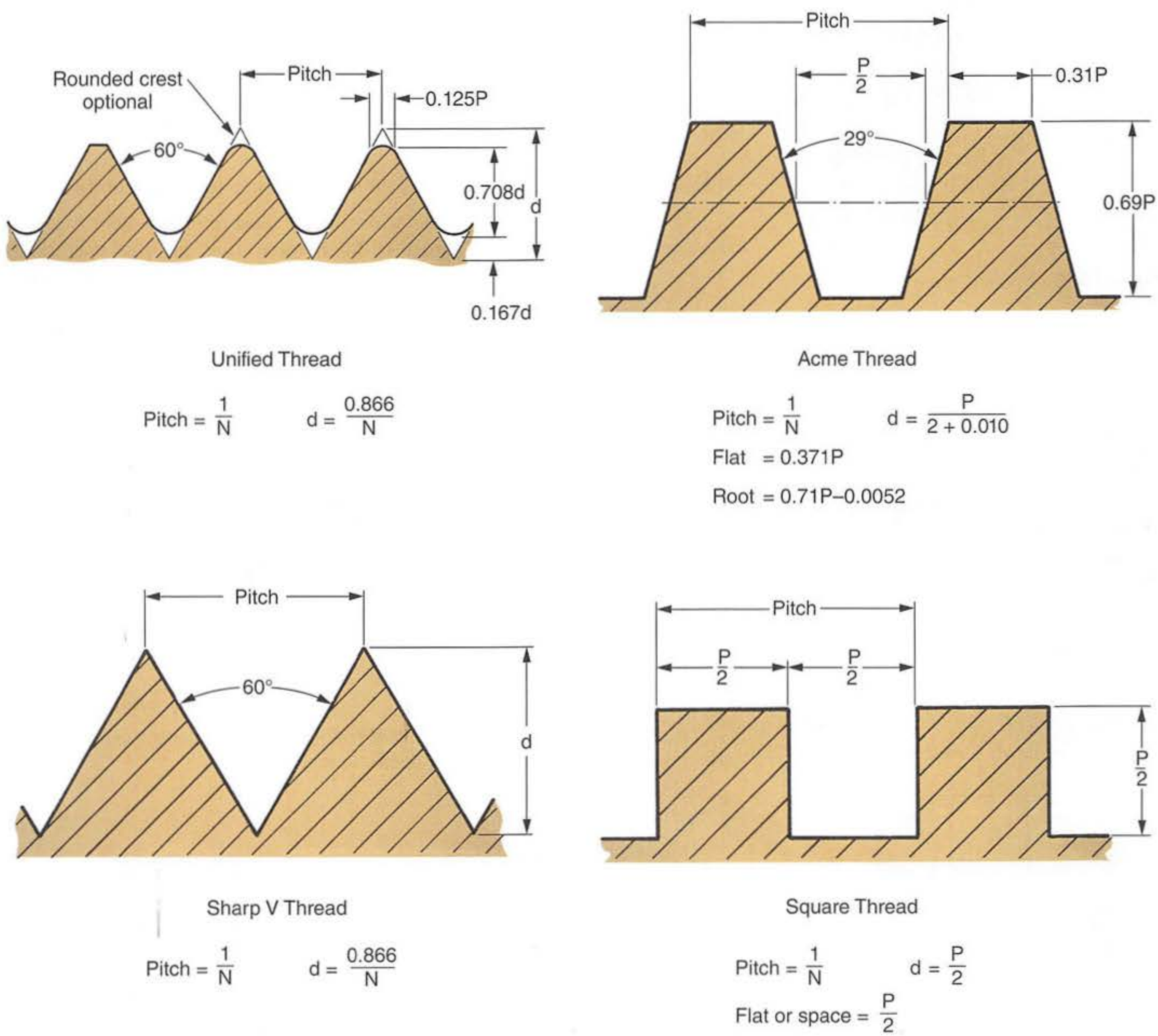
- Communicate freely with other team members.
- Avoid blaming others.
- Support the ideas of other group members; consider all ideas without immediate dismissal.
- Do not brag or try to be the "superstar," but act more as a team player.
- Listen actively.
- Get involved.

Creative ideas often develop from building on another person's idea. Honesty and openness are essential. Also, trying to understand the ideas of others before trying to get others to understand your ideas is an effective skill to develop.

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 16-25**. More than 75% of all threads cut in the United States are of the Unified (UN) 60° type.

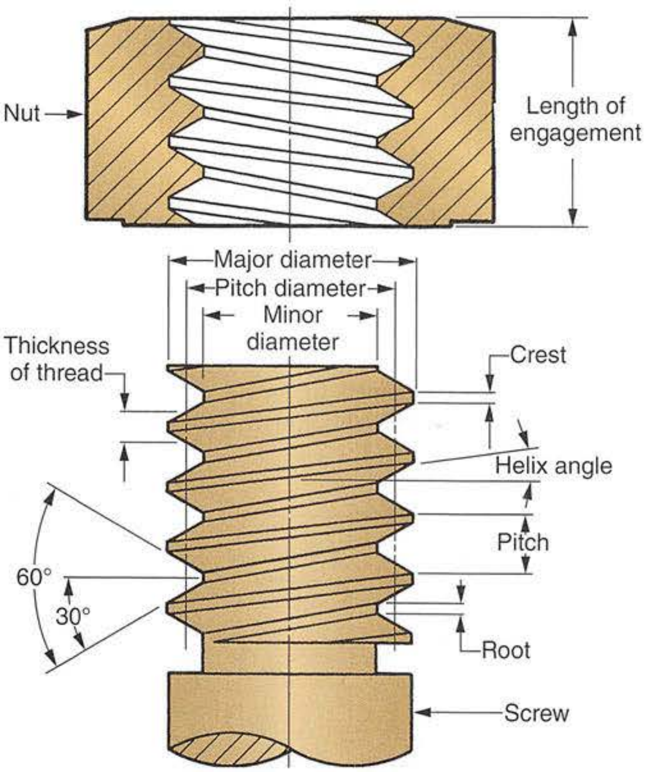
The following terms relate to screw threads, as shown in **Figure 16-26**:

- ***External threads*** are cut on the outside surface of the piece.
- ***Internal threads*** are cut on the inside surface of the 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 the points at which the width of the thread and width of the thread groove are equal.
- ***Pitch*** is the distance from one thread point to the same point on the next thread, measured parallel to the thread axis. The 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 create an increase in lead without weakening the thread. See **Figure 16-27**.



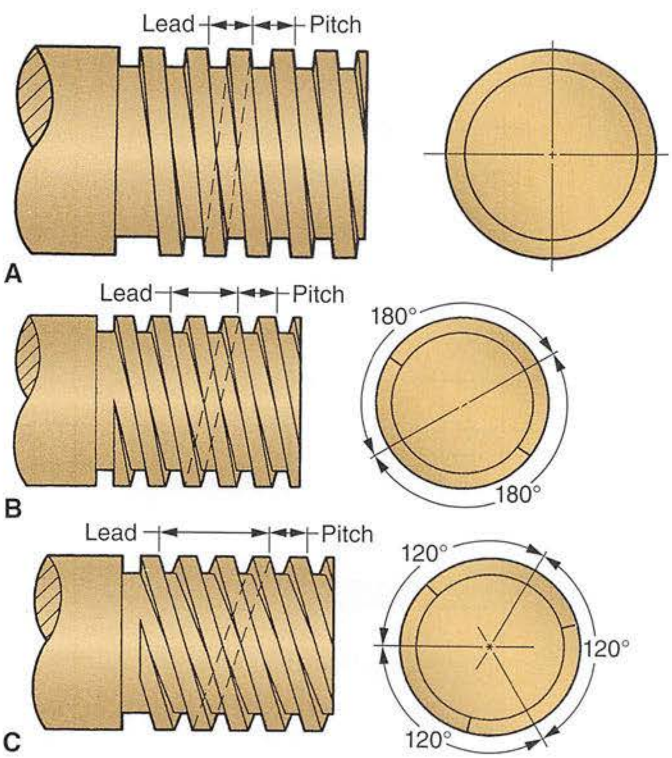
Goodheart-Willcox Publisher

Figure 16-25. Common thread forms. *Note:* In these formulas, N = Number of threads per inch; P = Pitch; d = depth of thread.



Goodheart-Willcox Publisher

Figure 16-26. Nomenclature of a thread.



Goodheart-Willcox Publisher

Figure 16-27. The difference between lead and pitch. A—On a single-thread screw, the pitch and lead are equal. B—On a double-thread screw, the lead is twice the pitch. C—On a triple-thread screw, the lead is three times the pitch.

16.3.2 Preparing to Cut 60° Threads on a Lathe

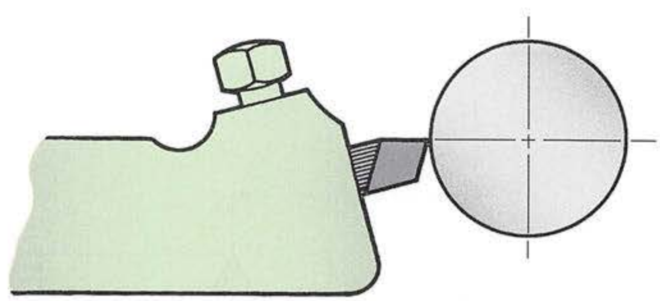
Sharpen the cutting tool to the correct shape, including the proper clearance. Grind the top flat with no side or back rake, **Figure 16-28**. Use an oilstone to touch up the cutting edges and form the radius on the tip. Use a center gage to grind and set the tool bit in position, **Figure 16-29**. The gage is often referred to as a *fishtail.*

Set up the work in the same manner as for straight turning. If mounted between centers, align the centers precisely to avoid producing a tapered thread. 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 faceplate slot.

A groove is frequently cut at the point where the thread is to terminate. The thread end groove is cut equal to the minor diameter of the thread and serves the following two purposes:

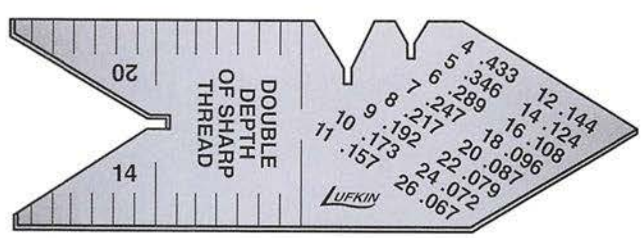
- It provides a place to stop the threading tool at the end of its cut.
- It permits a nut to be run up to the end of the thread.

Several methods may be used to terminate a thread, as shown in **Figure 16-30**. In most cases, beginners should use a groove until they gain sufficient experience to attempt the other methods. However, the design of some parts does not permit a groove to be used. In such cases, the threads must be terminated by another method. They require perfect



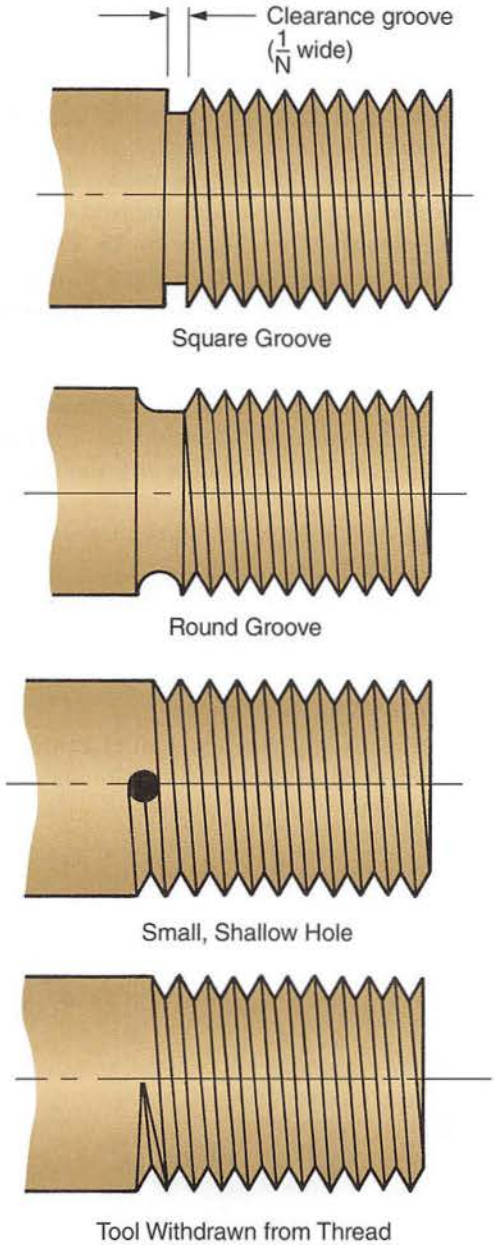
Goodheart-Willcox Publisher

Figure 16-28. Cutting tool positioned for cutting 60° threads. Set the tool on center as shown.



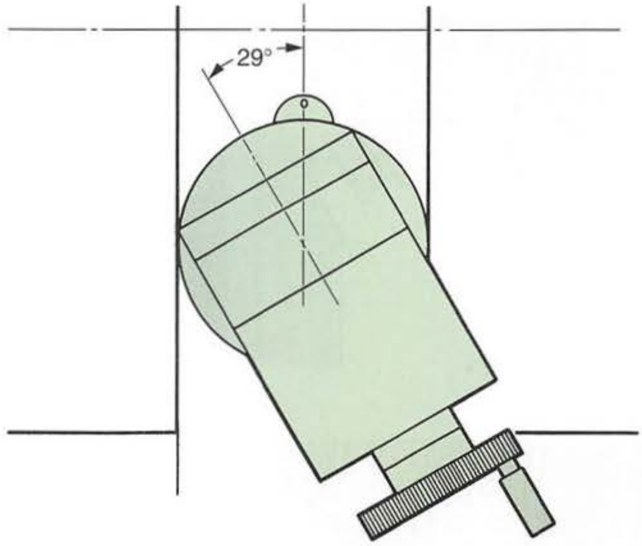
Lufkin Rule Co.

Figure 16-29. A center gage or fishtail.



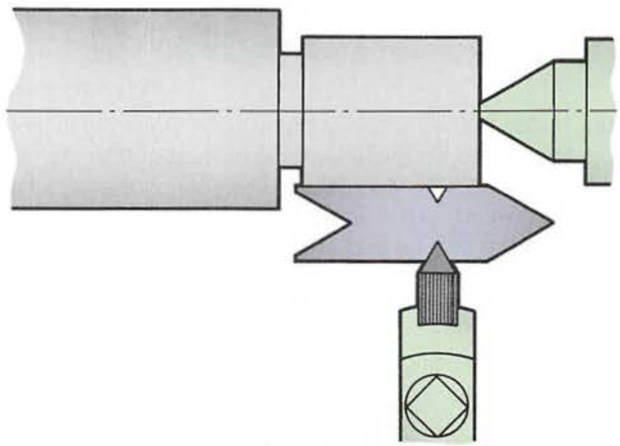
Goodheart-Willcox Publisher

Figure 16-30. Techniques for terminating a screw thread.



Goodheart-Willcox Publisher

Figure 16-31. The compound rest is rotated to 29° for machining right-hand external threads.



Goodheart-Willcox Publisher

Figure 16-32. Positioning a cutting tool for machining threads using a center gage.

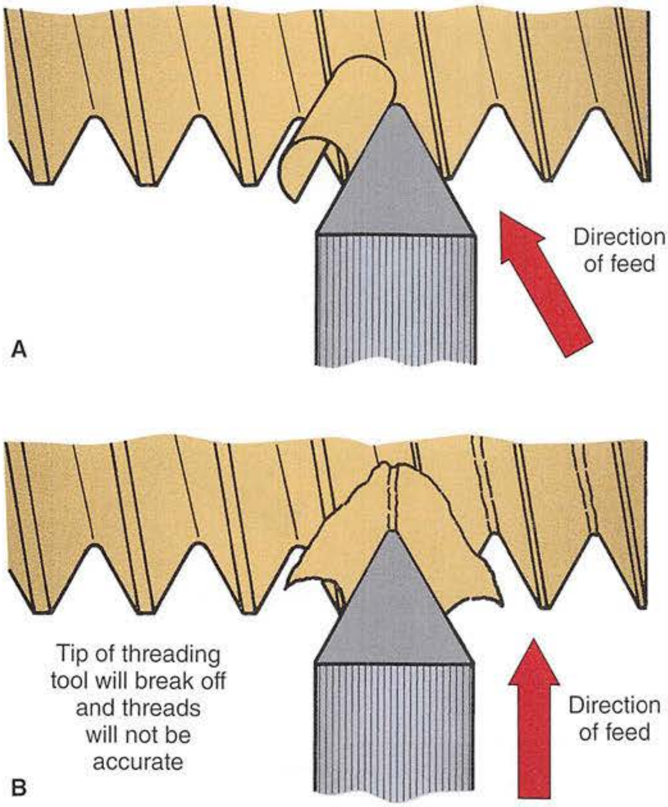
coordination and rapid operation of the cross-slide to get the tool out of position at the end of the cut.

Adjust the gearbox to cut the correct number of threads. Make apron adjustments to permit the half-nuts to be engaged. After making the proper apron and gear adjustments, pivot the compound rest to 29° to the right, **Figure 16-31**. 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. 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 16-32**. Tool height can be set using the centerline scribed on the tailstock spindle or using the center point.

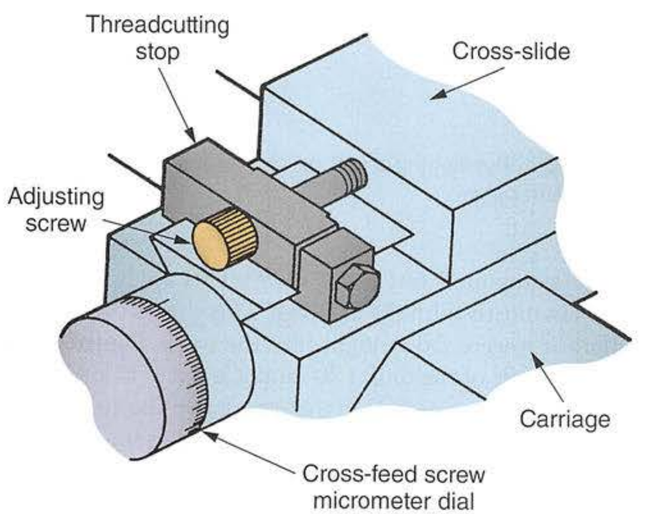
The compound rest is set at 29°. This angle causes the tool to cut mostly with the left edge. This shears the chip better than if it were fed straight into the work, **Figure 16-33**. Since the angle of the tool is 30° and it is fed in at an angle of 29°, the slight shaving action that results on the right side of the thread produces a smooth finish. 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 ***threadcutting stop*** may be used. Set the point of the tool to just touch the work. Then lock the stop to the saddle dovetail with the adjusting screw just bearing on the stop, **Figure 16-34**.



Goodheart-Willcox Publisher

Figure 16-33. Cutting action of the tool. A—When the tool is fed in at a 29° angle, the left edge does most of the cutting, and the cutting load is distributed evenly across the left edge. B—When the cutting tool is fed straight in, both edges are cutting and the weakest part of the tool, the point, is doing the hardest work.



Goodheart-Willcox Publisher

Figure 16-34. After being properly adjusted, the threadcutting stop lets you start the next cut in same position.

After making a cutting pass, 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 threadcutting stop. Then turn the compound rest in by a distance of 0.002″ to 0.005″ (0.05 mm to 0.12 mm) to position the tool for the next cut.

A ***thread dial*** that meshes with the lead screw is fitted to the carriage of most lathes, **Figure 16-35**. The thread dial indicates 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.

The face of the thread dial, **Figure 16-36**, 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.

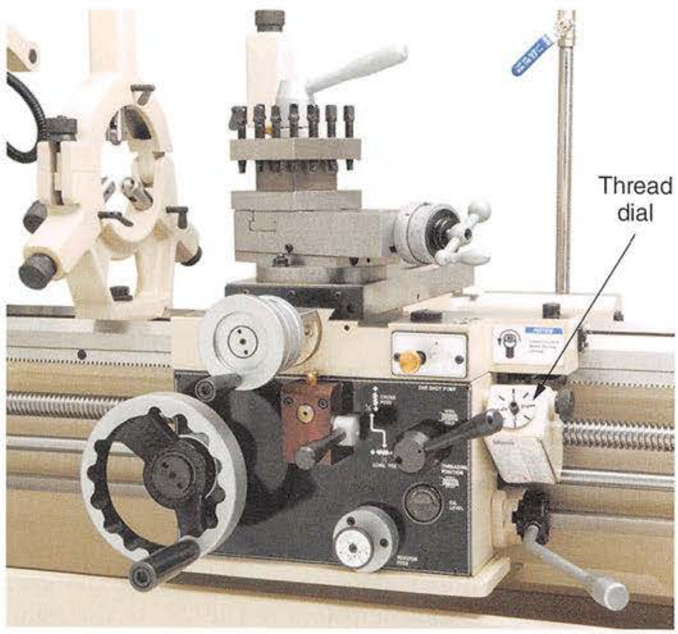
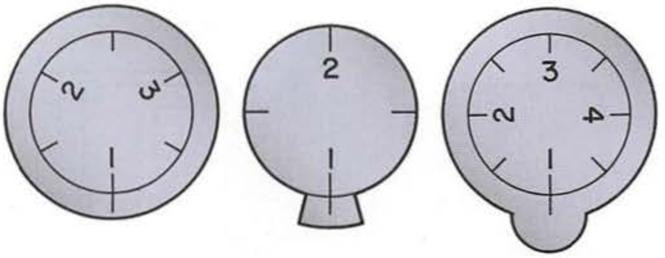


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 16-35. A thread dial used to cut either inch-based or metric-based threads. The housing contains a series of gears, with gear selection depending on the threads being cut.



Goodheart-Willcox Publisher

Figure 16-36. 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 cutting threads 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.

However, the thread dial can be used on lathes with full metric capabilities. The thread dial varies 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.

16.3.3 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. Make a first cut 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 16-37**. When 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. 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.

16.3.4 Resetting a Tool in the Thread

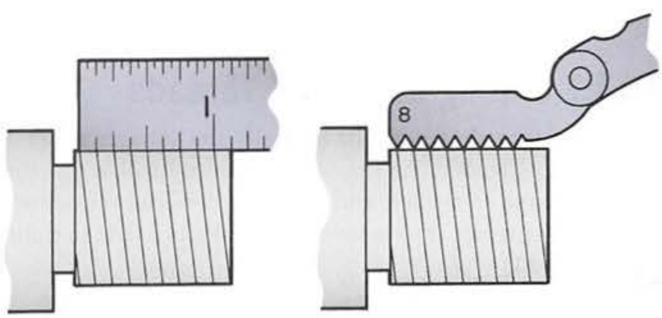
It is sometimes necessary to replace a broken cutting tool, or to resharpen the tool for the finish cuts. After replacing the tool, you must realign it with the portion of the thread already cut. This can be done as follows:

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. Use the compound rest screw and the cross-slide screw to align the tool in the existing thread.
5. Reset the threadcutting stop after the tool has been aligned.

16.3.5 Cutting Threads with Threading Inserts

There are two basic types of 60° threading inserts: the partial-profile insert and the full-profile insert. Partial-profile inserts, **Figure 16-38**, are the 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. Deburring may be required when cutting threads on most metals.

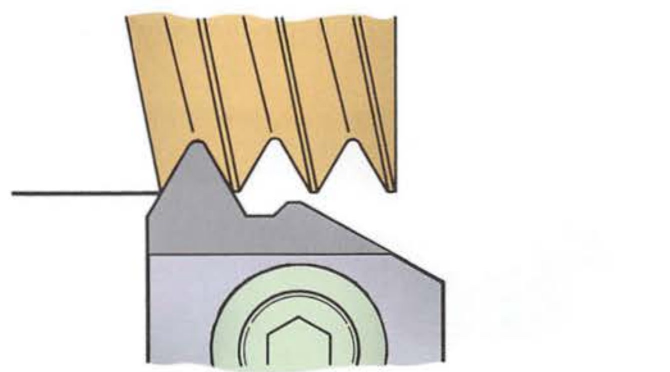
Full-profile inserts, **Figure 16-39**, produce the best thread form and finish. The tool cuts the leading flank, the root, and the trailing flank simultaneously. The machinist needs only to check the pitch diameter to determine if the major and minor diameters of the thread are to size. No deburring is necessary because the insert trims the thread



A B

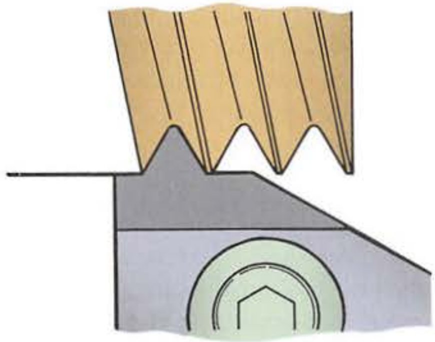
Goodheart-Willcox Publisher

Figure 16-37. Always check thread pitch after making a first, light cut. A—Checking with a rule. B—Checking with a screw pitch gage.



Goodheart-Willcox Publisher

Figure 16-38. Cutting threads with a partial-profile insert. The major (outside) diameter of the thread must be cut to size before using this type of insert.



Goodheart-Willcox Publisher

Figure 16-39. Using a full-profile insert to cut a thread. A separate insert is required for each thread pitch.

crest. The disadvantage of the full-profile insert is that a separate insert is required for each thread pitch.

16.3.6 Measuring Threads

Measure threads at frequent intervals during the machining operation to ensure 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.

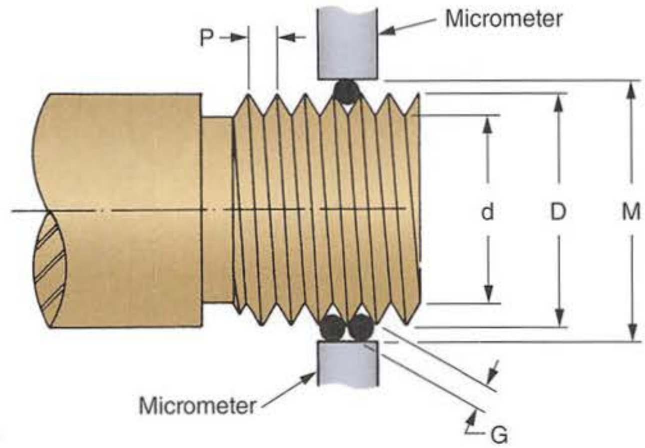
A thread micrometer can be used to make quick, accurate thread measurements. It has a pointed spindle and a double-V anvil to engage the thread. See **Figure 16-40**. The micrometer reading given is the true pitch diameter. It equals the outside diameter of the screw minus the depth of one thread. Each micrometer is designed to read a limited number of thread pitches. Micrometers are available in both inch and millimeter graduations.

In the ***three-wire method*** of measuring threads, three wires of a specific diameter are fitted into the threads and a micrometer measurement is made over the wires, **Figure 16-41A**. The formula in **Figure 16-41B** is used to calculate the correct measurement over the wires.

A three-wire thread measuring system has been developed to simplify and speed up the three-wire measuring process. It consists of a digital micrometer mounted in a fixture that holds the threaded workpiece and the three wires. See **Figure 16-42**.

Mahr Federal Inc.

Figure 16-40. A thread micrometer can be used to check cut threads precisely.



B

$$M = D + 3G - \frac{1.5155}{N}$$

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.

Goodheart-Willcox Publisher

Figure 16-41. Three-wire method of measuring 60° screw threads. A—Arrangement of the workpiece, wires, and micrometer. B—The three-wire thread measuring formula.

16.3.7 Cutting Left-Hand Threads

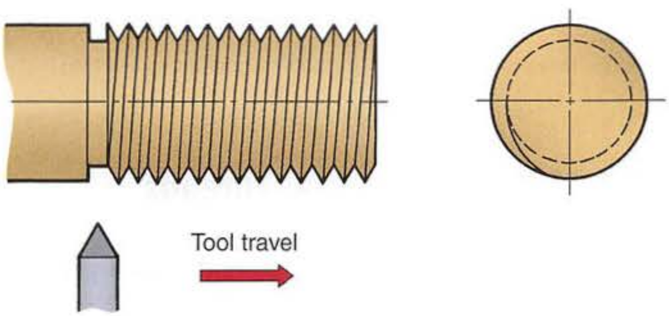
Left-hand threads are cut in basically the same manner as right-hand threads. The major differences involve pivoting the compound to the left and changing the lead screw rotation so the carriage travels toward the tailstock (left to right), **Figure 16-43**.

16.3.8 Cutting Square Threads

Square threads are used to transmit motion. They are more difficult to cut than 60° threads.

Mitutoyo/MTI Corp.

Figure 16-42. Thread measurements can be made in a fraction of the time normally needed with this three-wire measuring system. Wires are mounted in individual holders that fit into the clamping fixture.



Goodheart-Willcox Publisher

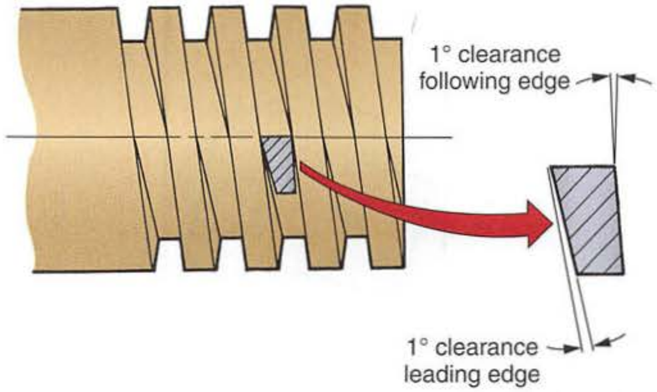
Figure 16-43. Direction of tool travel for cutting left-hand threads.

To cut a square thread, first calculate the width of the required tool bit (0.5 × thread pitch). If the square thread is fairly coarse, grind a roughing tool 0.010″ to 0.015″ (0.2 mm to 0.4 mm) smaller than the thread groove width. Grind the cutting point of the finishing tool 0.002″ to 0.003″ (0.05 mm to 0.08 mm) wider than the calculated groove width. Be sure to grind adequate clearance on the cutting tool, **Figure 16-44.**

16.3.9 Cutting Acme Threads

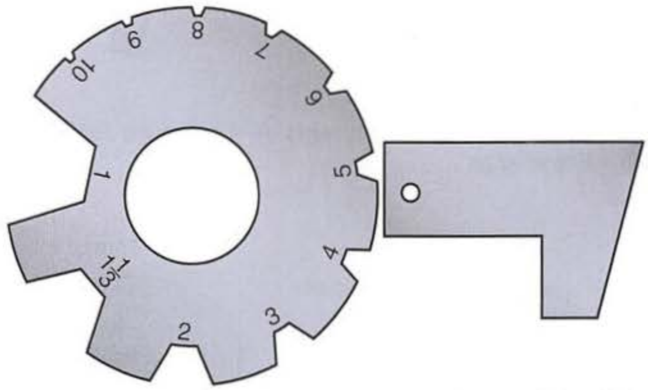
On the Acme thread, the top and bottom are flat, but the sides have a 29° included angle. This thread type was originally developed to replace the square thread. Its advantages are the strength and ease with which it can be cut, compared to the square thread. The thread form is used in machine tools for precise control of component movement.

The Acme screw thread gage is the standard for grinding and setting Acme threadcutting 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 16-45.**



Goodheart-Willcox Publisher

Figure 16-44. Allow adequate side clearance when sharpening a tool to cut square threads.



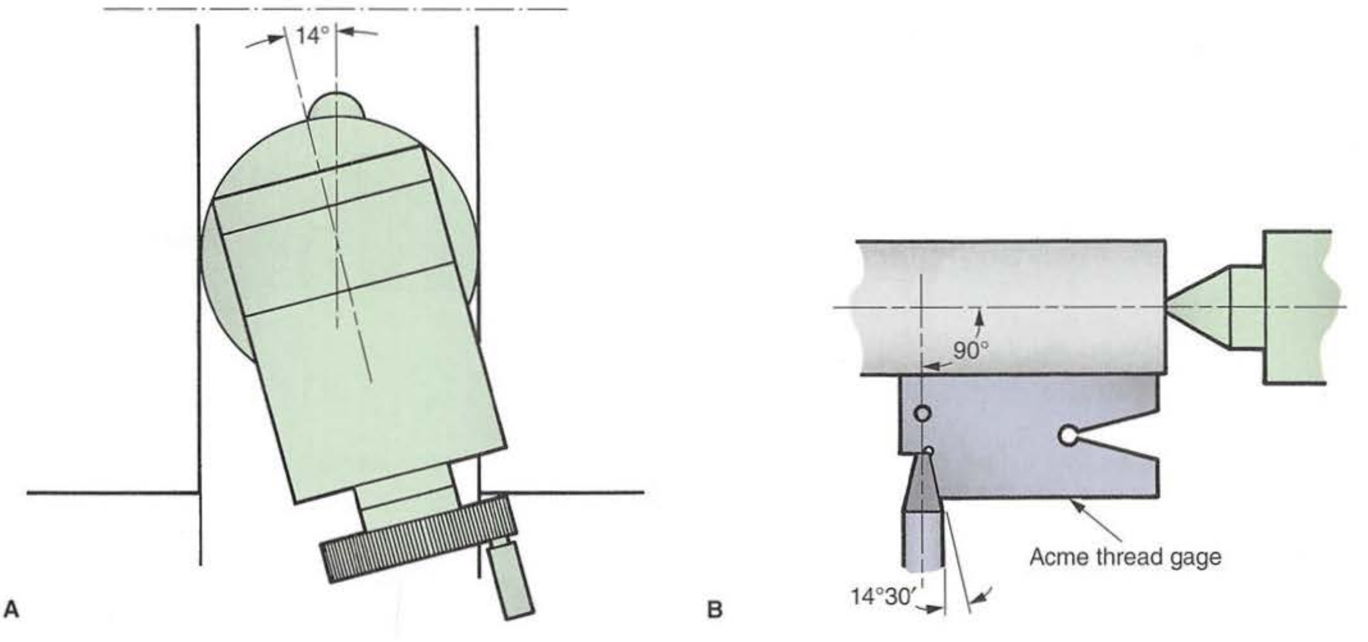
Goodheart-Willcox Publisher

Figure 16-45. The Acme screw thread gage and tool setup gage allow you to check lathe settings.

To cut the threads, the groove is usually roughed out with a square-nosed tool to approximate depth and is then finished with an Acme-shaped tool. The compound rest is set to 14°, and the tool is positioned using the thread gage, **Figure 16-46.** Other than this, Acme threads are cut in the same manner as the Sharp V thread.

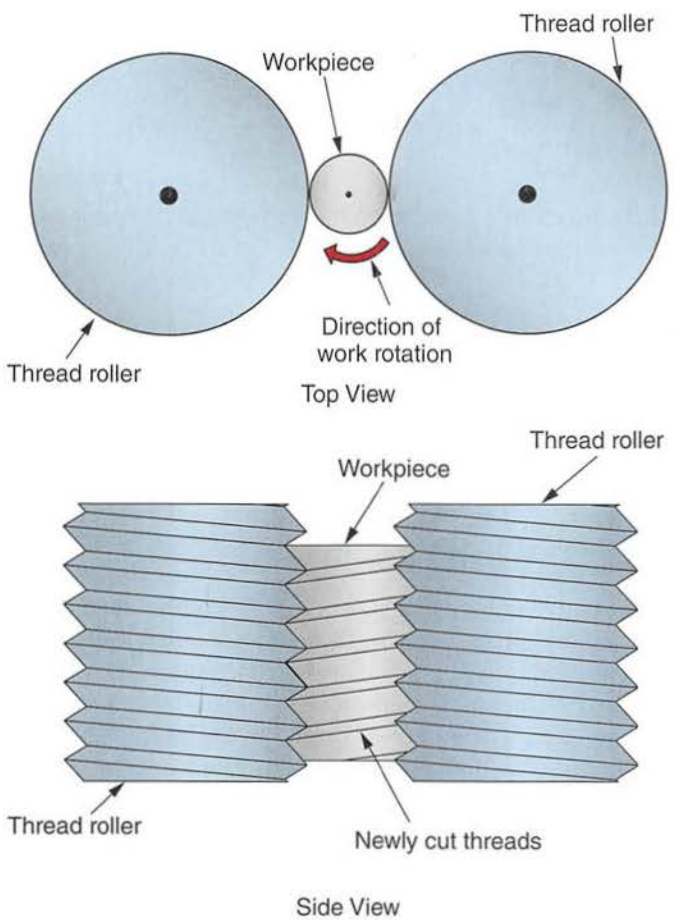
16.3.10 Thread Rolling

For high-volume production, thread rolling is a faster method for cutting external threads on an outside diameter. ***Thread rolling*** is the process of creating screw threads on external diameters by deforming the material to create the screw thread. The process is similar to the knurling process, in which dies with the proper thread shape and pitch are fed into the workpiece. Some thread rolling tools are fed in a direction perpendicular to the center axis of the workpiece. Others are threaded along the axis. Not only is thread rolling faster than cutting threads manually, but the process also makes the material denser at the weaker parts of the thread, making the threads stronger and extending their life. See **Figure 16-47.**



Goodheart-Willcox Publisher

Figure 16-46. Cutting Acme threads. A—Compound setting for cutting Acme threads. B—The cutting tool is positioned with an Acme thread gage.



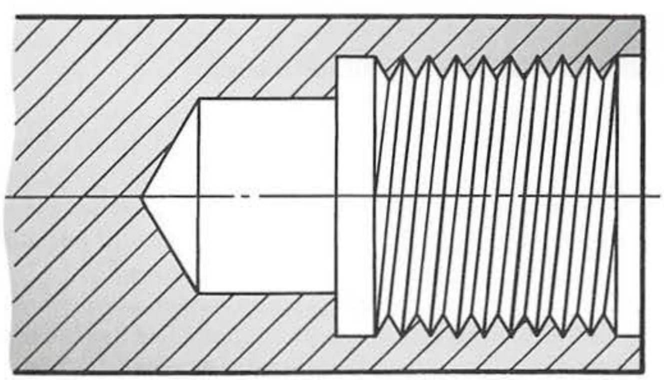
Goodheart-Willcox Publisher

Figure 16-47. Thread rolling.

16.3.11 Cutting Internal Threads

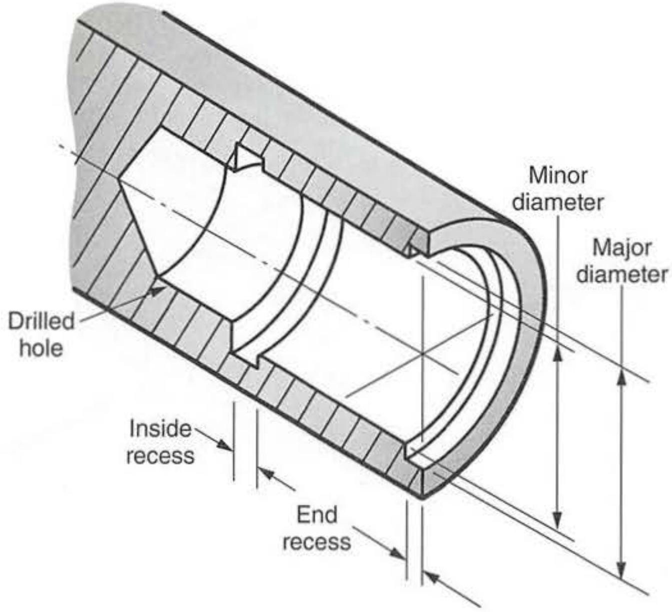
Internal threads, **Figure 16-48**, are made on the lathe with a conventional boring bar and a cutting tool sharpened to the proper shape. When cutting internal threads, tool infeed and removal from the cut are the reverse of those used when cutting external threads.

Before internal threads can be machined, the work must be prepared. Drill a hole and bore it to the correct size for the thread's minor diameter. Then machine a recess with a square-nosed tool at the point where the thread terminates, **Figure 16-49**. The diameter of the recess is equal to the major diameter of the thread.



Goodheart-Willcox Publisher

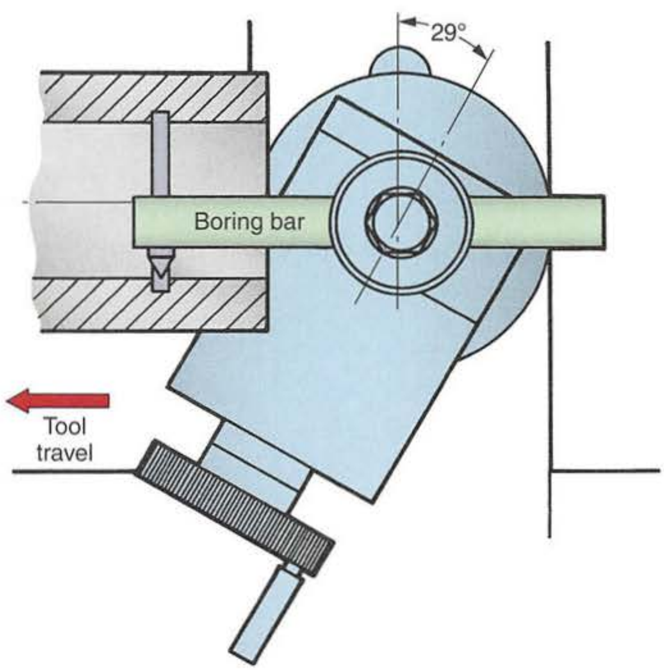
Figure 16-48. Internal threads.



Goodheart-Willcox Publisher

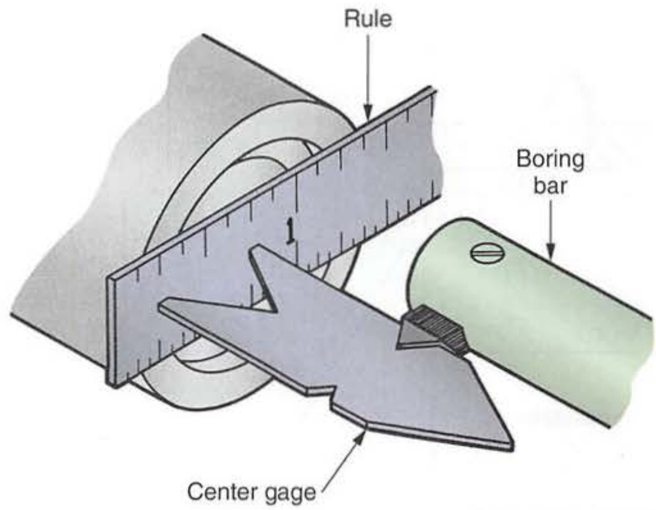
Figure 16-49. The opening for internal screw threads has been drilled and grooves have been machined to the major diameter.

To cut right-hand internal threads, pivot the compound rest 29° to the left. See **Figure 16-50**. Mount the tool on center and align it using a center gage, **Figure 16-51**. 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).



Goodheart-Willcox Publisher

Figure 16-50. Compound setting for cutting internal right-hand screw threads.



Goodheart-Willcox Publisher

Figure 16-51. Positioning the cutting tool for machining internal screw threads.

GREEN MACHINING

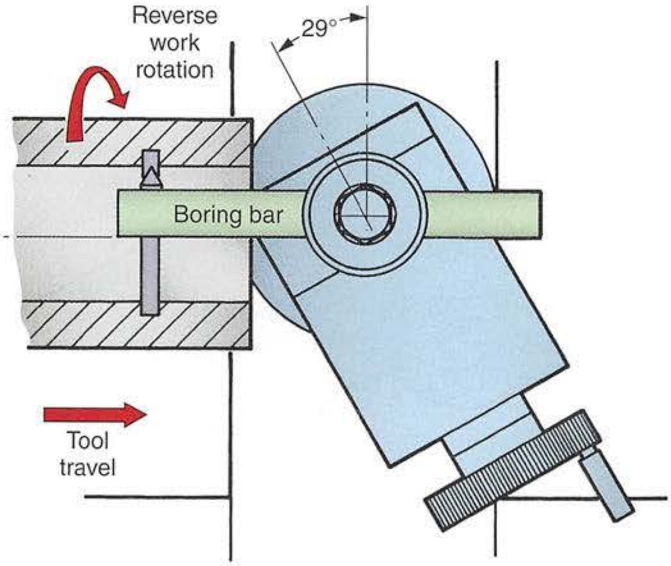
Recycling Cutting Tools

Cutting tools can be recycled in several ways. They may be redesigned or reshaped for additional use after they are initially worn out. Reusing tools is both environmentally and economically sensible, since reuse saves the materials used in the cutting tools themselves and the cost to purchase the tools. If the tool cannot be reshaped, sometimes the tool holding can be redesigned to fit a partially worn tool. Tools that are too worn to reuse may be recycled into new tools. Properly recycled materials require substantially less energy to craft new tools when compared to crafting new tools from raw materials.

You may have trouble determining when the tool has traveled far enough into the hole that the half-nuts can be disengaged. One method is to use 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.

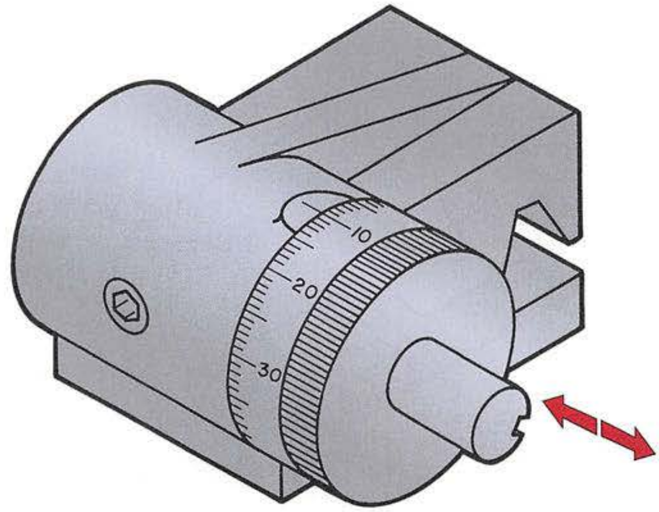
Another technique allows you to start at the back of the hole when cutting internal threads. Pivot the compound rest 29° to the right. Place the threading tool to the rear of the boring bar with the cutting edge up. See **Figure 16-52**. Run the lathe spindle in reverse. To prevent the tool from being placed too far into the hole to start the cut, mount a micrometer carriage stop on the ways. See **Figure 16-53**. Return the carriage until it touches the stop. For cutting the threads, follow the same general procedure previously described.

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.



Goodheart-Willcox Publisher

Figure 16-52. An alternative setup for cutting internal right-hand threads. The work rotates in a direction opposite that of normal turning operations.

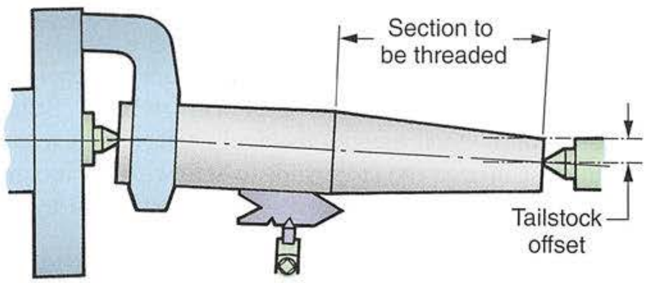


Goodheart-Willcox Publisher

Figure 16-53. When using the 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 the hole when starting each cut.

16.3.12 Cutting Threads on a Tapered Surface

Sometimes tapered threads must be cut 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 16-54.**



Goodheart-Willcox Publisher

Figure 16-54. Tool setup for machining screw threads on a taper. The tool is not positioned on the taper, but rather to its centerline.

Chapter Review

Summary

- There are several methods for turning a taper on a lathe. Each has its own advantages and disadvantages.
- The tailstock setover can be calculated using a variety of measurements: taper per inch (TPI), taper per foot (TPF), diameter of the long end (D), diameter of the small end (d), length of the taper (l), and total length of the piece (L).
- When cutting a taper, additional strain is placed on the centers. The workpiece must be checked frequently for binding.
- Cutting a taper with a taper attachment is more accurate than other methods.
- Tapers can be measured by comparison, or they can be measured directly.
- The Unified 60° thread form is the most common type of thread. Other thread forms include the left-hand, square, and Acme threads.
- A variety of methods and tools exist for cutting threads.
- Three methods of measuring threads are using a threaded hole or nut, a thread micrometer, or the three-wire method.

Review Questions

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

1. What is a taper?
2. There are five ways of machining tapers on a lathe. List them.

Machine adjustments must be calculated for each tapering job. The information given below will enable you to calculate the necessary tailstock setover for questions 3–5.

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 the tapered section are known, but TPI or TPF is not given,

$$\text{Offset} = \frac{L \times (D - d)}{2 \times l}$$

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 also be used when dimensions are in mm.

3. What will the tailstock setover be for the following job?
   Taper per inch = 0.125″
   Total length of piece = 4.000″
4. What will the tailstock setover be for the following job?
   D = 2.50″
   d = 1.75″
   l = 6.00″
   L = 9.00″
5. 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
6. What two functions do taper plug gages and taper ring gages serve?
7. List five important uses of screw threads.
8. A groove is cut to the depth of the thread at the point where the thread is to terminate. What two purposes does this groove serve?
9. The tip of a cutting tool to cut a Sharp V thread is sharpened using a(n) _____ to check that it is the correct shape.
10. 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.
11. The _____ that is fitted to many lathe carriages meshes with the lead screw and is used to indicate when to engage the half-nuts to permit the threadcutting tool to follow exactly in the original cut.

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

For questions 12–15, calculate the correct measurement over the wires for the following threads. Use the wire size given in the problem. Round your calculations to the nearest thousandth.

12. 1/2-20 UNF
(wire size 0.032″)
13. 1/4-20 UNC
(wire size 0.032″)
14. 3/8-16 UNC
(wire size 0.045″)
15. 7/16-14 UNC
(wire size 0.060″)

For questions 16–22, match the following terms and identifying phrases.

16. Cut on outside surface of piece.
17. Cut on inside surface of piece.
18. Largest diameter of thread.
19. Smallest diameter of thread.
20. 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.
21. Distance from one point on a thread to a corresponding point on next thread.
22. Distance a nut will travel in one complete revolution of screw.

A. Minor diameter
B. Major diameter
C. Pitch
D. External thread
E. Pitch diameter
F. Internal thread
G. Lead

CHAPTER **17**

The Milling Machine

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Identify the various types of milling machines.
- Describe milling operations and methods.
- Select the proper cutter for the job to be done.
- Explain how milling cutters are held and driven on milling machines.
- Calculate cutting speeds and feeds.
- List the purposes of cutting fluids.
- Describe milling work-holding attachments.
- Understand and follow milling safety practices.

Technical Terms

arbor
climb milling
column-and-knee milling machine
conventional milling
cutting speed
face milling
feed
fixed-bed milling machine
horizontal milling machine
peripheral milling
side milling cutter
vertical milling machine

A milling machine rotates a multitoothed cutter into the workpiece to remove material, **Figure 17-1**. Each tooth of the cutter removes a small, individual chip of material. A variety of cutting operations can be performed on a milling machine. The milling machine is capable of machining flat or contoured surfaces, slots, grooves, recesses, threads, gears, spirals, and other configurations.

Milling machines are available in more variations than any other family of machine tools, **Figure 17-2**. These machines are well suited to computer-controlled operation. Work may be clamped directly to the machine table, held in a fixture, or mounted in or on one of the numerous work-holding devices available for milling machines.

17.1 Types of Milling Machines

It is difficult to classify the various categories of milling machines because their designs tend to merge with one another. For practical purposes, however, milling machines may be grouped into two large families:

- Fixed-bed milling machines.
- Column-and-knee milling machines.

BIG Kaiser Precision Tooling Inc.

Figure 17-1. A milling machine rotates a multitoothed cutter into the workpiece.

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

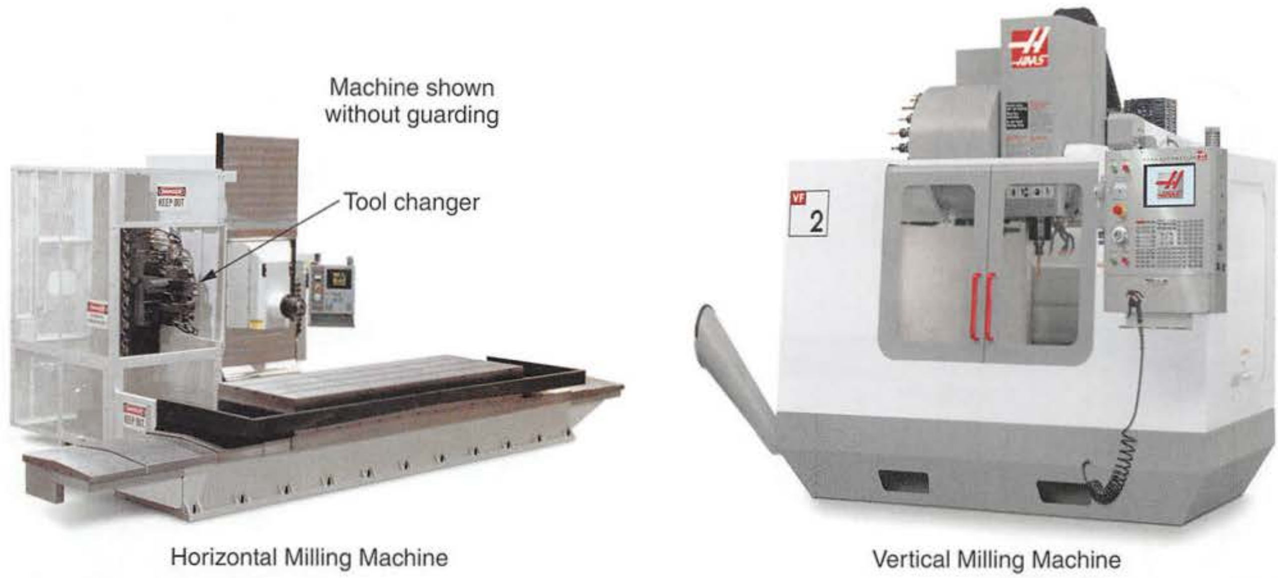
Figure 17-2. A knee-type vertical milling machine that incorporates longitudinal and traverse power feeds, digital readout, and a built-in coolant system.

Both groups are made with horizontal or vertical spindles, **Figure 17-3**. On a ***horizontal milling machine***, the cutter is fitted onto an arbor mounted in the machine on an axis parallel with the worktable. Multiple cutters may be mounted on the spindle for some operations.

The cutter on a ***vertical milling machine*** is normally perpendicular (at a right angle) to the worktable. However, on many vertical milling machines, the spindle can be tilted to perform angular cutting operations.

17.1.1 Fixed-Bed Milling Machines

Fixed-bed milling machines have a very rigid worktable construction and support, **Figure 17-4**. The worktable moves only in a longitudinal (back and forth/X-axis) direction, and can vary in length from 3′ to 30′ (0.9 to 9.0 m). Vertical (up and down/Z-axis) and cross (in and out/Y-axis) movements are obtained by moving the cutter head.

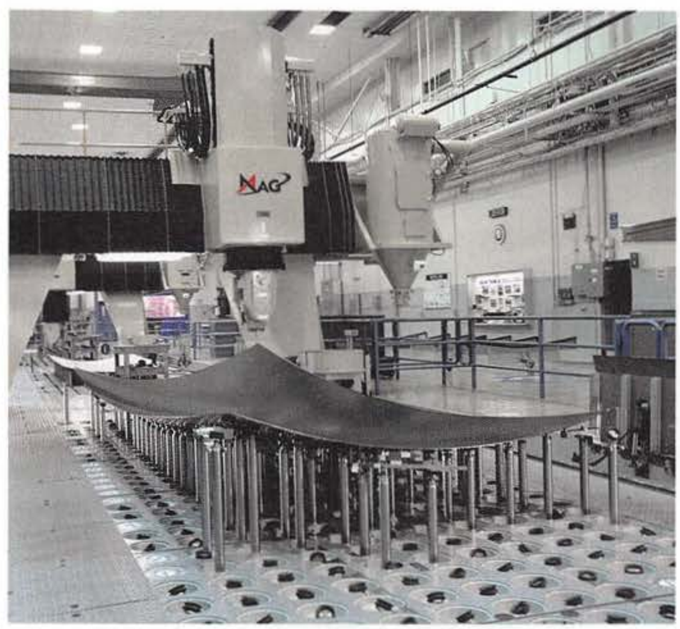


Haas Automation, Inc.

Figure 17-3. Horizontal and vertical CNC milling machines.

Autocon Technologies, Inc.

Figure 17-4. Fixed-bed or bed mills have a very rigid worktable that moves only in a longitudinal direction. The machine shown can be operated manually or by CNC.



MAG IAS, LLC

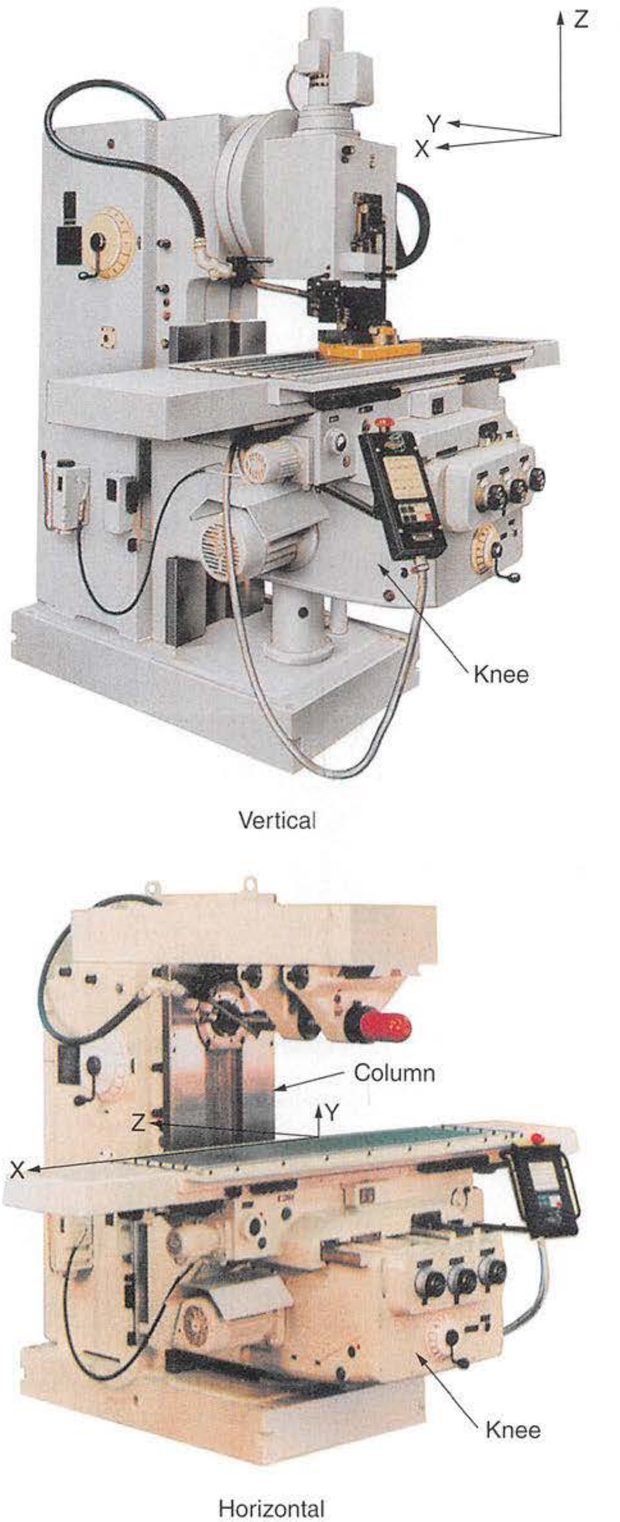
Figure 17-5. This large, 5-axis, fixed-bed, vertical profiler milling machine is cutting a contoured section of an aircraft wing skin.

Fixed-bed milling machines can be further classified as horizontal, vertical, or planer machines. The bed permits heavy cutting on large workpieces, **Figure 17-5**.

17.1.2 Column-and-Knee Milling Machines

The ***column-and-knee milling machine*** is so named because of the parts that provide movement to the workpiece. They consist of a column that supports and guides the knee in vertical (up and down/Z-axis) movement and a knee that supports the mechanism for obtaining table movements. These movements are traverse (in and out/Y-axis) and longitudinal (back and forth/X-axis). See **Figure 17-6**.

These machines are commonly referred to as *knee-type milling machines*. The three basic categories of knee-type milling machines are plain (horizontal) milling machines, universal milling machines, and vertical milling machines.

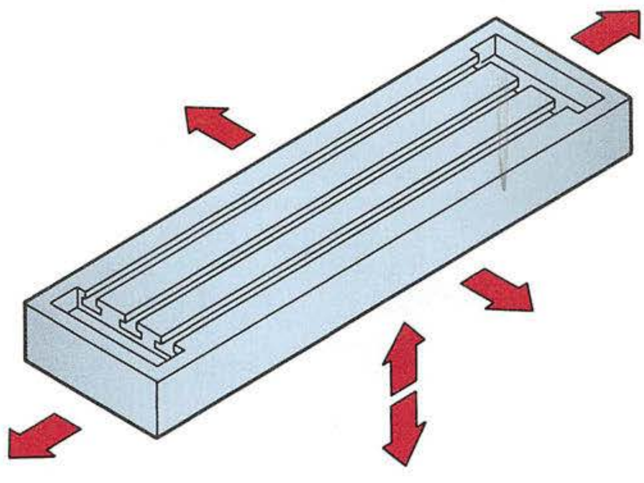


WMW Machinery Company, Inc.

Figure 17-6. Column-and-knee milling machines.

Sharp Industries, Inc.

Figure 17-7. Plain milling horizontal machine.



Goodheart-Willcox Publisher

Figure 17-8. Table movements of a plain horizontal milling machine.

Plain Milling Machine

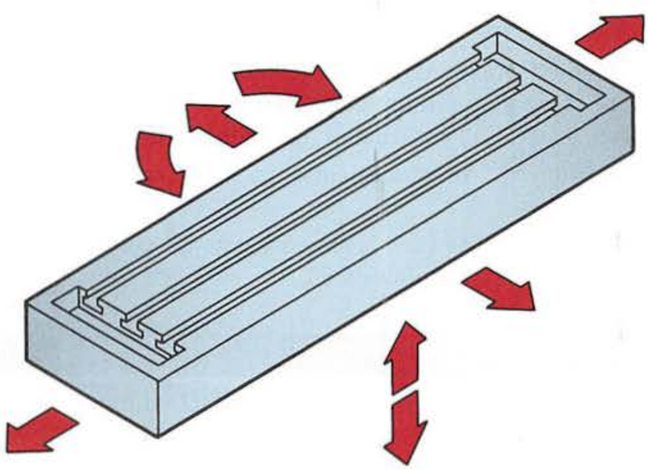
On the plain milling machine, the cutter spindle projects horizontally from the column, **Figure 17-7**. The worktable has three movements: vertical, cross, and longitudinal (X, Y, and Z axes), **Figure 17-8**.

Universal Milling Machine

A universal milling machine, **Figure 17-9**, is similar to the plain milling machine, but the table has a fourth axis of movement. On this type of machine, the table can be swiveled on the saddle through an angle of 45° or more, **Figure 17-10**. This makes it possible to produce spiral gears, spiral splines, and similar workpieces.

WMW Machinery Company, Inc.

Figure 17-9. On a universal horizontal milling machine, the table can be swiveled 45° or more.



Goodheart-Willcox Publisher

Figure 17-10. Table movements possible on a universal milling machine.

Vertical Milling Machine

A vertical milling machine differs from the plain and universal machines in that its cutter spindle is vertical, at a right angle to the top of the worktable. See **Figure 17-11**. The cutter head can be raised and lowered by hand or by power feed. This type of milling machine is best suited for use with an end mill or face mill cutter.

Types of vertical mills include swivel-head, sliding-head, and rotary-head mills. A swivel-head milling machine, **Figure 17-12**, is the type often found in training programs. The spindle can be swiveled for angular cuts.

Rem Sales, Inc.

Figure 17-11. The vertical milling machine.

GREEN MACHINING

Cleaning Machined Parts

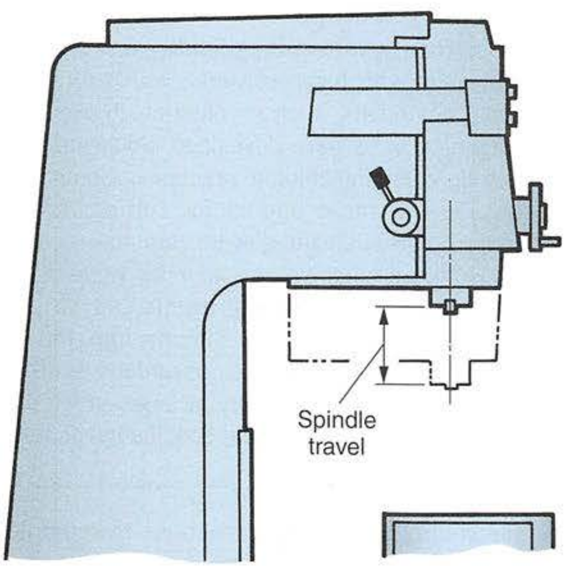
After machining with cutting fluids, parts are traditionally cleaned with harsh solvents, which often contain harmful chemicals, such as chlorine. Newer methods for cleaning parts have developed alongside the push for other green machining practices. Green cleaning options mirror those offered for cutting fluids—eco-friendly, nontoxic cleaning agents are now readily available. Nontoxic cleaners are safer for workers and the environment. These cleaning agents can be disposed of more easily (sometimes directly into the sewage system) because they are less hazardous. With continued support from the industry at large, green cleaning options are expanding to meet specific needs in various machining fields.

On the sliding-head milling machine, the spindle head is fixed in a vertical position. The head can be moved up and down (vertically) by hand or under power, **Figure 17-13**.

The spindle on the rotary-head milling machine can be moved vertically and in circular arcs of adjustable radii about a vertical centerline, **Figure 17-14**. It can be adjusted manually or under power feed.

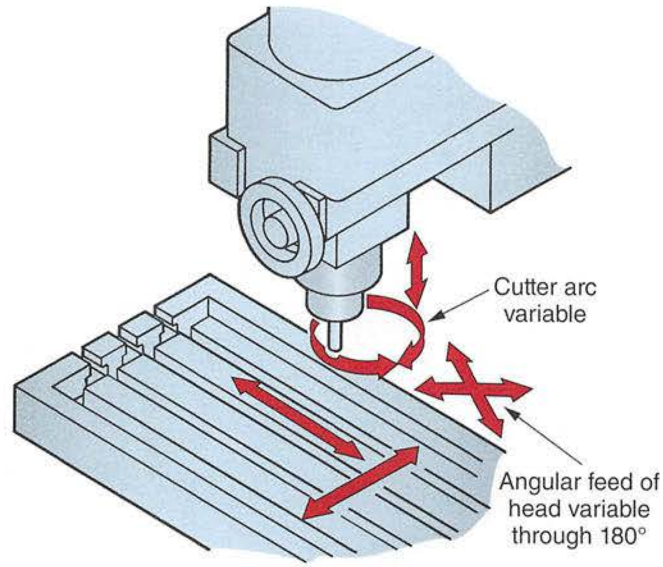
Republic-Lagun Machine Tool Co.

Figure 17-12. A typical swivel-head milling machine.



Goodheart-Willcox Publisher

Figure 17-13. The spindle head is fixed in a vertical position on a sliding-head milling machine. The entire head is moved to make cutting adjustments.



Goodheart-Willcox Publisher

Figure 17-14. Spindle movements possible on a rotary-head vertical milling machine. Milling machines with CNC capabilities are replacing this type of machine.

17.1.3 Methods of Milling Machine Control

The method employed to control table movement is another way of classifying milling machines, as well as all machine tools in general. There are four methods of control:

- **Manual.** All movements are made by hand lever control.
- **Semiautomatic.** Movements can be controlled by either hand or power feed.

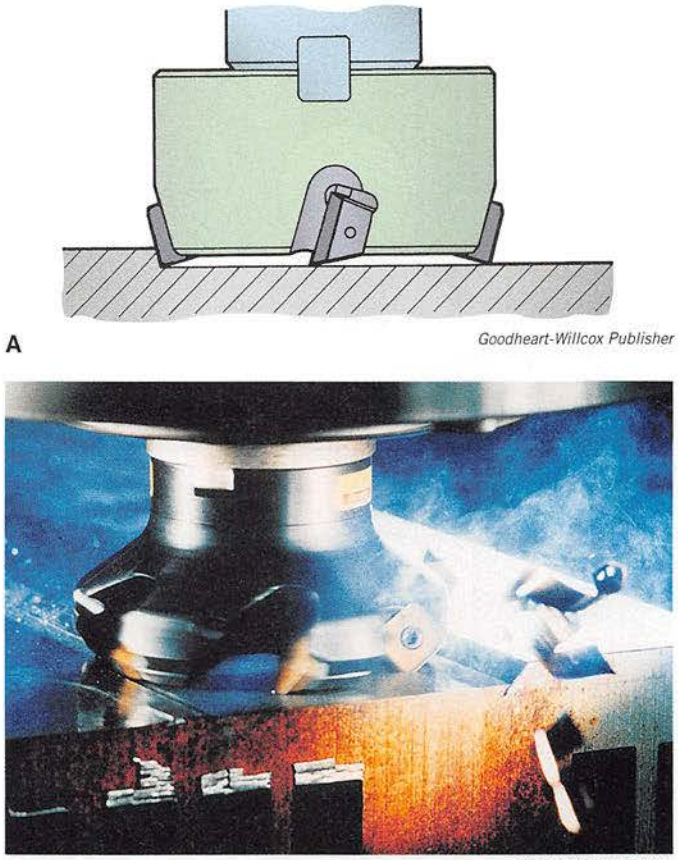
SAFETY NOTE

The operator should always verify that all hand cranks and handwheels are disengaged before activating power feed features.

- **Fully automatic.** A complex hydraulic feed arrangement that follows two- or three-dimensional templates to automatically guide one or more cutters. Specifications can also be programmed to guide the cutters and table through the required machining operations.
- **Computerized (CNC).** Machining coordinates are entered into a computer using a programming language. Instructions from the computer operate actuators (electric, hydraulic, or pneumatic devices) that move the table and cutter or cutters through the required machining sequence. Manually operated milling machines can be retrofitted with computerized control systems, **Figure 17-15**.

Autocon Technologies, Inc.

Figure 17-15. This manual vertical milling machine was retrofitted with a CNC control unit. The retrofit provides two-axis (X and Y) machine control capabilities and displays depth (Z axis) information on the monitor.



A Goodheart-Willcox Publisher

B Sandvik Coromant Co.

Figure 17-16. Face milling. A—The surface being machined is parallel with the cutter face. B—An example of face milling.

Small milling machines may have power feed available only for longitudinal table movement. On larger machines, automatic feed or power feed is used for all table movements.

Table movement (feed) can be engaged at cutting speed. However, there is a rapid traverse feed that allows fast power movement in any direction of feed engagement. This permits work to be positioned at several times the fastest rate indicated on the feed chart. The operator positions the automatic power feed control lever to give the desired directional movement and activates the rapid traverse lever.

SAFETY NOTE

Never activate rapid traverse while the cutter is positioned in a cut.

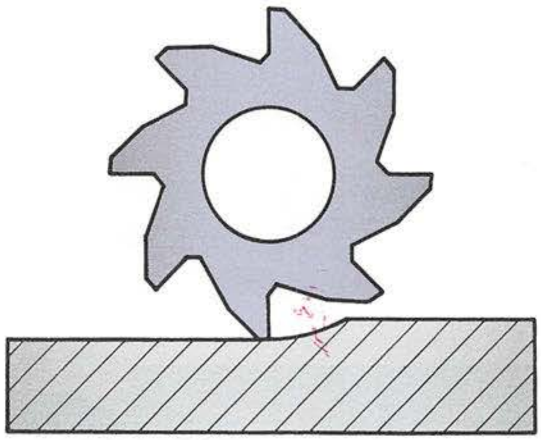
17.2 Milling Operations

There are two main categories of milling operations. ***Face milling*** is machining performed on a surface that is parallel to the cutter face, **Figure 17-16**. Large, flat surfaces are machined using this technique. ***Peripheral milling***, also known as *edge milling*, is machining performed on a surface that is parallel with the periphery of the cutter, **Figure 17-17**.

There are also two distinct methods of milling: conventional milling and climb milling. In ***conventional milling***, also known as *up-milling*, the work is fed into the rotation of the cutter, **Figure 17-18A**. The chip is at minimum thickness at the start of the cut. The cut is so light that the cutter has a tendency to slide over the work until sufficient pressure builds up to cause the teeth to bite into the material. This initial sliding motion, followed by the sudden breakthrough as the tooth completes the cut, leaves the "milling marks" so familiar on many milled surfaces. The marks and ridges can be kept to a minimum by keeping the table gibs properly adjusted.

In ***climb milling*** or *down-milling*, the work moves in the same direction as cutter rotation, **Figure 17-18B**. Full engagement of the cutter tooth is instantaneous. The sliding action of conventional milling is eliminated, resulting in a better finish and longer tool life. The main advantage of climb milling is the tendency of the cutter to press the work down on the worktable or holding device.

Climb milling is not recommended on light machines, nor on large older machines that are not in top condition or



A

Goodheart-Willcox Publisher

B

BIG Kaiser Precision Tooling Inc.

Figure 17-17. Peripheral milling. A—In this milling method, the surface being machined is parallel with the periphery of the cutter. B—An example of peripheral milling.

are not fitted with an antibacklash device to take up play. There is danger of a serious accident if there is play in the table, or if the work or work-holding device is not mounted securely.

17.3 Milling Cutter Basics

The typical milling cutter is circular in shape with a number of cutting edges (teeth) located around its circumference. Milling cutters are manufactured in a large number of stock shapes, sizes, and kinds. See **Figure 17-19**.

17.3.1 Milling Cutter Classification

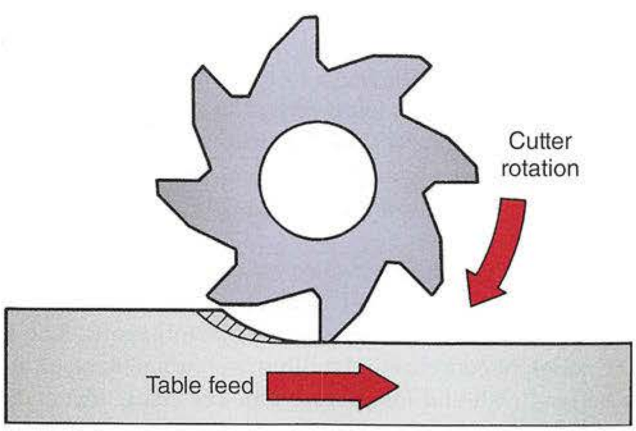
Milling cutters are often classified by the method used to mount them on the machine. Milling cutters can be mounted directly by their shanks, directly to the spindle nose, or with an arbor. Shank cutters are fitted with either a straight or taper shank that is an integral part of the cutter. They are held in the machine by collets or sleeves. Arbor cutters have a suitable hole for mounting to an arbor.

There are two general types of milling cutters. Solid cutters are made with the shank, body, and cutting edges all in one piece, **Figure 17-20**. Indexable insert cutters have teeth made of a cutting material. The teeth are brazed or clamped in place, **Figure 17-21**. Worn and broken teeth can be replaced easily instead of discarding the entire cutter.

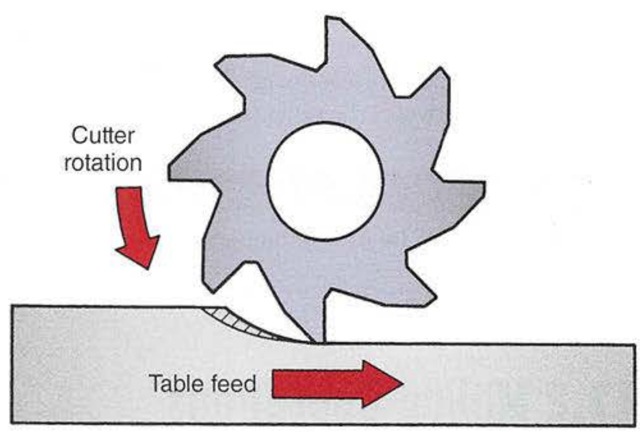
17.3.2 Milling Cutter Materials

Considering the wide range of materials that must be machined, the ideal milling cutter should have the following attributes:

- **High abrasion resistance.** The cutting edges should not wear away rapidly due to the abrasive nature of some materials.



A



B

Goodheart-Willcox Publisher

Figure 17-18. Milling methods. A—Cutter and work movement in conventional or up-milling. B—Cutter and work movement in climb or down-milling.

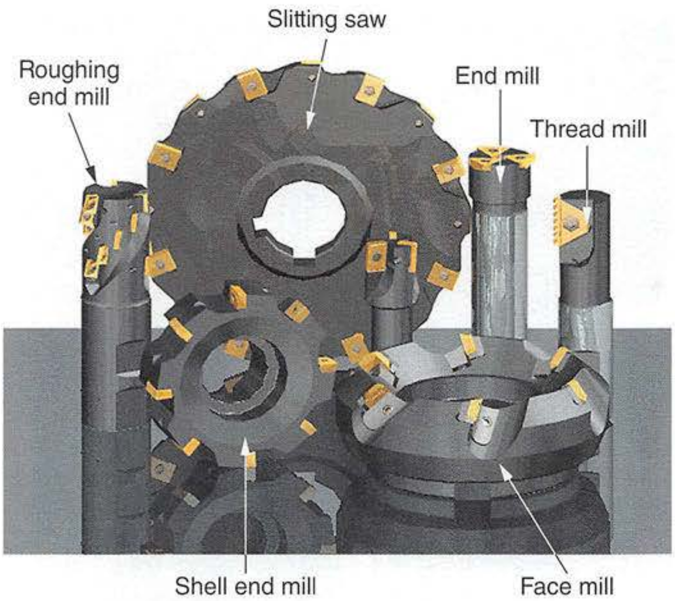


Figure 17-19. A selection of indexable insert milling cutters.

Figure 17-20. A selection of two- and four-flute HSS solid milling cutters. The gold on the cutters is a titanium nitride (TiN) coating that increases wear resistance and the lifetime of the cutter significantly.

- **Red hardness.** The cutting edges should not be affected by the terrific heat generated by many machining operations.
- **Edge toughness.** The cutting edges should not readily break down due to the loads imposed on them by the cutting operation.

Since no single material can meet these requirements in all situations, cutters are made from materials that are, by necessity, a compromise.

High-speed steels (HSS) are the most versatile cutter materials. Cutters made from HSS are excellent for general-purpose work and where vibration and chatter are problems. They are preferred for use on low-power machines.

HSS milling cutters can be improved by the application of surface lubricating treatments, surface hardening treatments, or coatings (such as chromium, tungsten, or tungsten carbide) to the cutting surfaces. The treated tools cost two to six times as much as conventional HSS tools, but they may last 5% to 10% longer or provide 50% to 100% higher metal removal rates with the same tool life.

Figure 17-21. A variety of cutters with indexable carbide inserts. When the cutting edges dull, new cutting edges are rotated into position. The inserts are available in different grades. The grade chosen for a job depends on the material being machined.

Cemented tungsten carbides include a broad family of hard metals. They are produced by powder metallurgy techniques and have qualities that make them suitable for metal cutting tools. Cemented carbides can, in general, be operated at speeds 3 to 10 times faster than conventional HSS cutting tools. Cemented carbide cutters are excellent for long production runs and for milling materials with a scale-like surface, such as cast iron, cast steel, or bronze.

In most cases, only the cutting tips (not the entire cutter) are made of cemented carbides. They are brazed or clamped to the cutter body. See **Figure 17-22**. Most inserted-tooth cutters use indexable inserts. Each insert has several cutting edges. When an edge becomes dull, the insert is indexed (turned) so that a new cutting edge contacts the metal.

17.4 Types and Uses of Milling Cutters

Milling cutters are commonly grouped into categories based on their shape and function. Selection of the proper cutting tool is very important to efficient milling operations. The following are the most commonly used milling cutters, with a summary of the work for which they are best suited.

Mitsubishi Materials USA Corporation

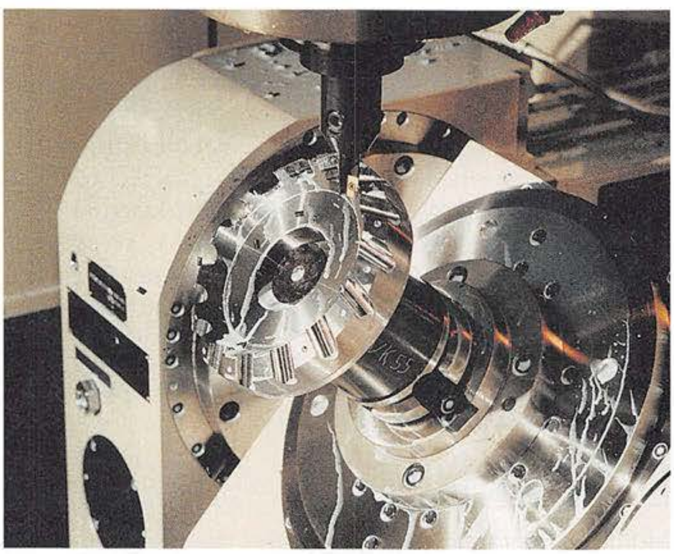
Figure 17-22. These inserted-tooth cutters have teeth that are clamped to the cutter body. The cutter teeth (gold in this photo) can be indexed four times to present a fresh cutting edge as they wear.

17.4.1 End Mills

End milling cutters are designed for machining slots, keyways, pockets, and similar work, **Figure 17-23**. The cutting edges are on the circumference and end. End mills may have straight or helical flutes, **Figure 17-24**, and have straight or taper shanks, **Figure 17-25**. Straight shank end mills are available in single- and double-end styles, **Figure 17-26**.

The terms *right-hand* and *left-hand* are used to describe the direction of cutter rotation and the helix of the flutes, **Figure 17-27**. When viewed from the cutting end, a right-hand cutter rotates counterclockwise, and a left-hand cutter rotates clockwise.

Ball-nose end mills, **Figure 17-28**, are used for tracer milling, computer-controlled contour milling, die-sinking, fillet milling, and other radius work. A cut with a depth equal to one-half the end mill diameter can generally be taken in solid stock. See **Figure 17-29**.

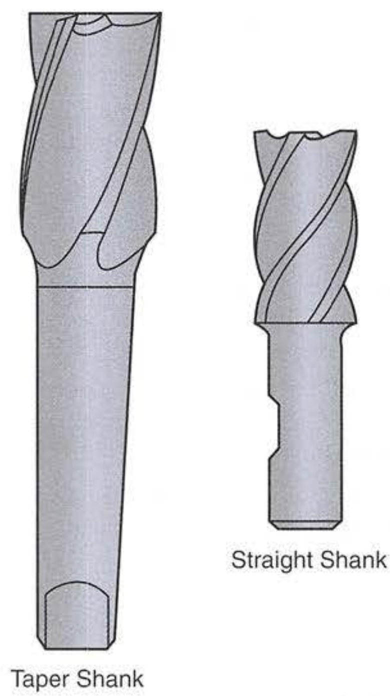


Kennametal

Figure 17-23. An indexable-insert end mill cutting clearance slots on a face milling cutter.

Mitsubishi Materials USA Corporation

Figure 17-24. End mills with multiple indexable inserts are available with either helical or straight flutes. Two face cutters are also shown.

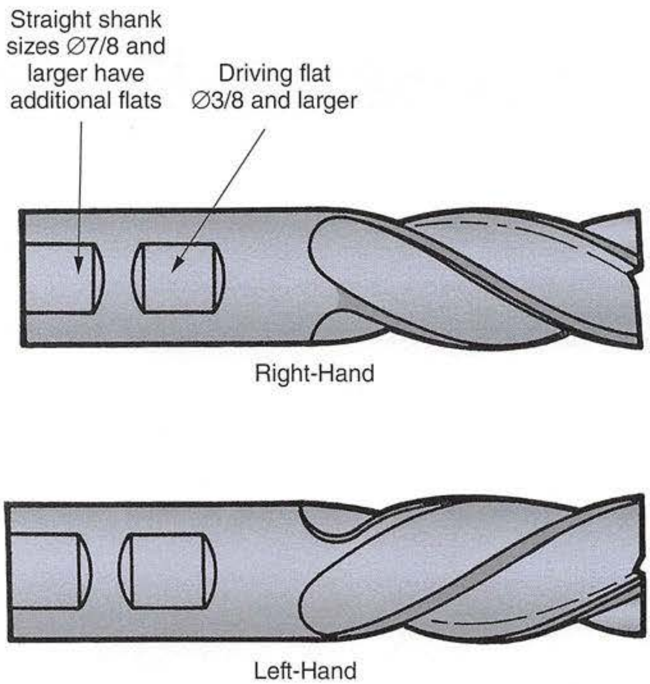


Goodheart-Willcox Publisher

Figure 17-25. Straight shank and taper shank end mills.

Goodheart-Willcox Publisher

Figure 17-26. Three large single-end end mills and a smaller double-end end mill.



Goodheart-Willcox Publisher

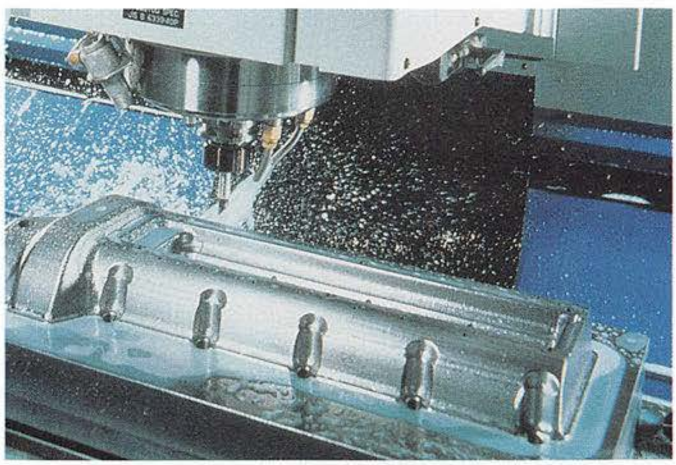
Figure 17-27. A right-hand cutter rotates counterclockwise when viewed from the cutting end. A left-hand cutter rotates clockwise.

Several end mill styles are available, including:

- **Two-flute end mill.** Can be fed into the work like a drill. It has two cutting edges on the circumference, with the end teeth cut to the center, **Figure 17-30**.
- **Multiflute end mill.** Can be run at the same speed and feed as a comparable two-flute end mill, but has a longer cutting life and produces a better finish. It is recommended for conventional milling when plunge cutting (feeding into the work like a twist drill) is not necessary. See **Figure 17-31**.
- **Shell end mill.** Has teeth similar to those on a multiflute end mill but is mounted on a stub arbor, **Figure 17-32**. The cutter is designed for both face and end milling. Shell end mills are made with right-hand cut/right-hand helix or with left-hand cut/left-hand helix.

Mitsubishi Materials USA Corporation

Figure 17-28. A ball-nose end mill has a rounded tip. These mills have two replaceable cutting inserts.



Delcam International

Figure 17-29. A ball-nose end mill being used for three-dimensional milling.

Goodheart-Willcox Publisher

Figure 17-30. The two-flute end mill can be fed into work like a drill.

Goodheart-Willcox Publisher

Figure 17-31. Multiflute end mills. The peripheral grooves in the cutter at right reduce chip size, lowering cutting forces. Most modern cutters have a nitride coating that improves resistance to abrasive wear and corrosion.

Goodheart-Willcox Publisher

Figure 17-32. Three indexable insert shell end mills.

CAREER CONNECTION

Industrial Machinery Mechanic

What does an industrial machinery mechanic do?

The machinery in any manufacturing area requires routine and expert service from professionals trained in the maintenance and repair of these machines. These professionals may be called *industrial machinery mechanics*, *maintenance workers*, or *millwrights*. Manufacturing machinery layouts change regularly to meet the needs of production.

What education and skills are needed to be an industrial machinery mechanic?

Candidates for industrial mechanic positions need at least a high school diploma and some additional technical training. Relevant courses include computer programming, electronics, mechanical drawing, mathematics, physics, welding, and print reading. Some community colleges also offer associate's degree programs in industrial maintenance.

What is it like to be an industrial machinery mechanic?

Industrial machinery mechanics often work in manufacturing environments that depend heavily on machinery, such as factories, power plants, and refineries. Each facility will have its own schedule of shifts, so work hours may vary. Due to exposure to manufacturing and machining hazards, this work can be dangerous. Follow all safety procedures and wear recommended personal protective equipment to avoid injury or illness.

According to the *Occupational Outlook Handbook*, over half of the industrial mechanics in the United States are employed by manufacturing industries. Their median annual wage is $49,600.

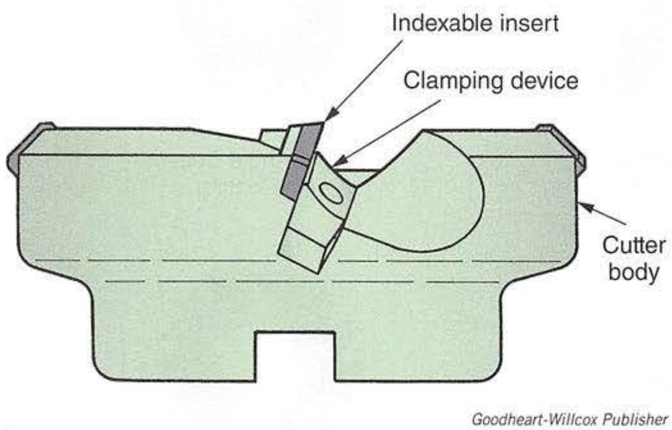
17.4.2 Face Milling Cutters

Face milling cutters are used to machine large, flat surfaces parallel to the face of the cutter, **Figure 17-33**. The teeth are designed to make roughing and finishing cuts in one operation. Because of their size and cost, most face milling cutters have inserted cutting edges. See **Figure 17-34**. Facing cutters can be mounted directly to a machine's spindle nose or on a stub arbor.

A fly cutter is a single-point cutting tool used as a face mill. An example is shown in **Figure 17-35**. Fly cutters are capable of machining very fine surface finishes. Fly cutters should not be used to make heavy roughing cuts.

17.4.3 Arbor Milling Cutters

Common arbor milling cutters include plain milling cutters, side milling cutters, angle cutters, metal-slitting saws, and formed milling cutters. Each of these types is described in the following sections. Refer to **Figure 17-36**.



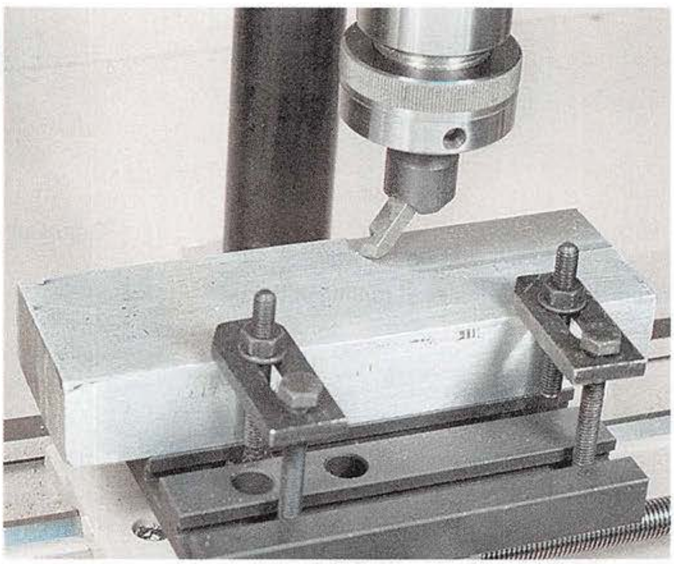
A

B

Figure 17-33. Face mill with indexable inserts. A—The replaceable inserts are mechanically clamped in place on the cutter body. As cutting edges wear, the inserts can be turned (indexed) to present a fresh cutting edge. B—An example of a face mill with indexable inserts. Insert selection is based on the material to be machined.

Valenite, Inc.

Figure 17-34. This face milling cutter has an unusual design, making use of bearing-mounted inserts that rotate as they cut. The manufacturer claims that the better heat dissipation provided by the rotating inserts increases cutter life and permits higher cutting speeds.



Goodheart-Willcox Publisher

Figure 17-35. The fly cutter is a single-point cutting tool used for face milling.

Figure 17-36. Examples of milling cutters.

Plain Milling Cutter

Plain milling cutters are cylindrical, with teeth located around their circumference. Plain milling cutters less than 3/4″ (20 mm) are made with straight teeth. Wider plain cutters, called *slab cutters,* are made with helical teeth designed to cut with a shearing action. This reduces the tendency for the cutter to chatter.

The different designs of plain milling cutters serve different purposes, as follows:

- **Light-duty plain milling cutter.** Used chiefly for light slabbing cuts and shallow slots.
- **Heavy-duty plain milling cutter.** Recommended for heavy cuts when considerable material must be removed.

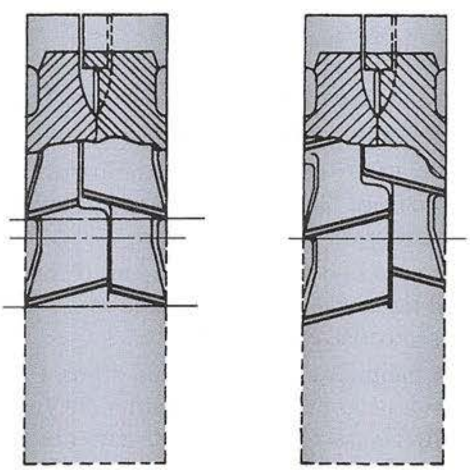
It has fewer teeth than a comparable light-duty cutter. The cutting edges are better supported, and chip spaces are ample to handle the larger volume of chips.

- **Helical plain milling cutter.** Has fewer teeth than either of the two previously mentioned cutters. The helical cutter can be run at high speeds and produces exceptionally smooth finishes.

Side Milling Cutter

Cutting edges are located on the circumference and on one or both sides of ***side milling cutters***. They are made in solid form or with inserted teeth. Types of side milling cutters include the following:

- **Plain side milling cutter.** The teeth are on the circumference and on both sides of the cutter. It is recommended for side cutting, straddle milling, and slotting. Plain side milling cutters are available in diameters ranging from 2″ (50 mm) to 8″ (200 mm) and in widths from 3/16″ (5 mm) to 1″ (25 mm).
- **Staggered-tooth side milling cutter.** Has alternating right-hand and left-hand helical teeth that help reduce chatter. They also provide adequate chip clearance for higher operating speeds and feeds than are possible with the plain side milling cutter. This type of cutter is especially good for machining deep slots.
- **Half side milling cutter.** Has helical teeth on the circumference but side teeth on one side only. It is made as a right-hand or left-hand cutter, and it is recommended for heavy straddle milling and milling to a shoulder.
- **Interlocking side milling cutter.** This cutter is ideally suited for milling slots or bosses and for making other types of cuts that must be held to extremely close tolerances. The unit is made as two cutters with interlocking teeth that can be adjusted to the required width using spacers or collars, **Figure 17-37**. The alternating right and left shearing action eliminates side pressures, producing a good surface finish.



Morse Tool Co.

Figure 17-37. Tooth pattern on interlocking side milling cutters.

Angle Cutters

Angle cutters differ from other cutters in that the cutting edges are neither parallel nor at right angles to the cutter axis.

- **Single-angle milling cutter.** Teeth are on the angular face and on the side adjacent to the large diameter. Single-angle cutters are made in both right-hand and left-hand cut, with included angles of 45° and 60°.
- **Double-angle milling cutter.** Used to mill threads, notches, serrations, and similar work. Double-angle cutters are manufactured with included angles of 45°, 60°, and 90°. Other angles can be special-ordered.

Metal-Slitting Saws

Metal-slitting saws are thin milling cutters that resemble circular saw blades. They are employed for narrow slotting and cutoff operations. Slitting saws are available in diameters as small as 2 1/2″ (60 mm) and as large as 8″ (200 mm).

- **Plain metal-slitting saw.** Used for ordinary slotting and cutoff operations. It is essentially a thin plain milling cutter. Both sides are ground concave for clearance. The hub is the same thickness as the cutting edge. It is stocked in thicknesses ranging from 1/32″ (0.8 mm) to 3/16″ (5 mm).
- **Side chip clearance slitting saw.** Similar to the plain side milling cutter, this saw is especially suitable for deep slotting and sawing applications because of its ample chip clearance.

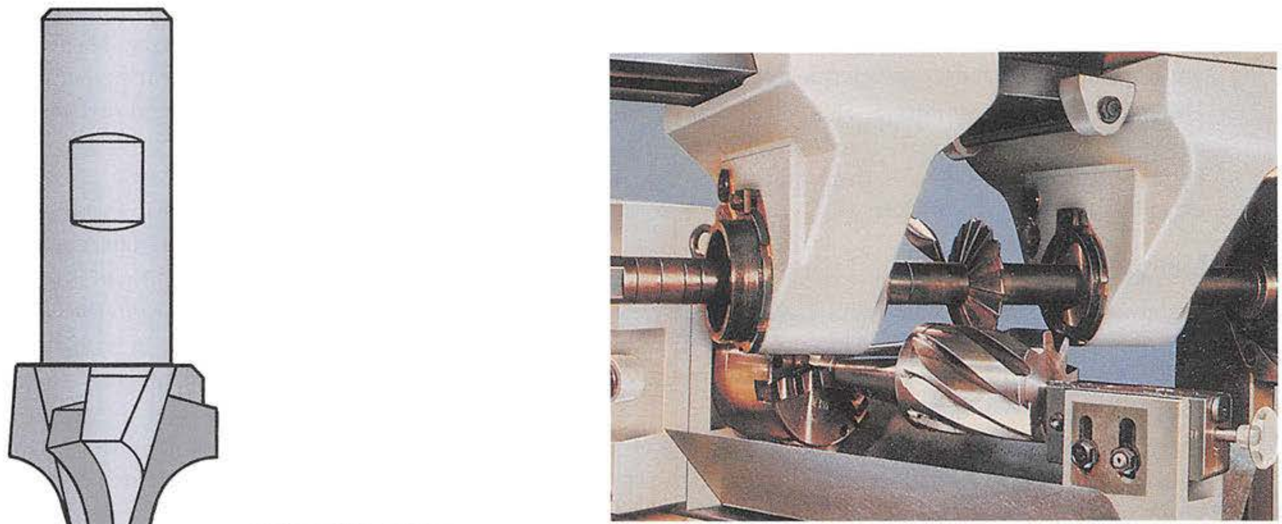
Formed Milling Cutters

Formed milling cutters are used to accurately duplicate a required contour. A wide range of shapes can be machined with the standard cutters available. See **Figure 17-38**. Included in this cutter classification are the concave cutter, convex cutter, corner-rounding cutter, and gear cutter.

17.4.4 Miscellaneous Milling Cutters

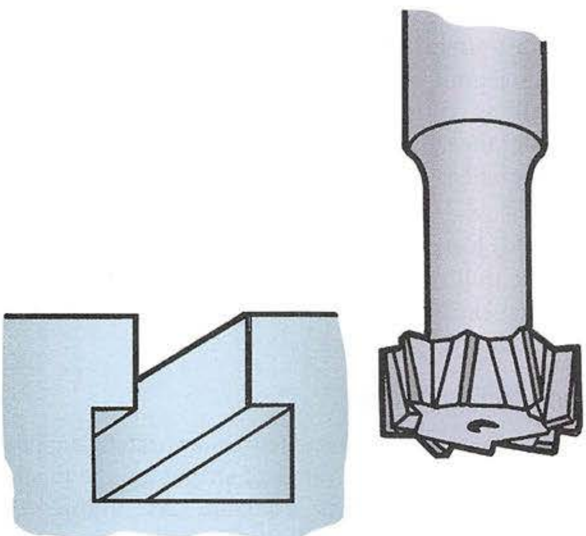
Some milling cutters commonly used in industry do not fit into any of the previously mentioned groups. These cutters include:

- **T-slot milling cutter.** Has cutting edges for milling the bottoms of T-slots after they have been cut with an end mill or side cutter, **Figure 17-39**.
- **Woodruff keyseat cutter.** Used to mill the semicircular keyseat for a Woodruff key.
- **Dovetail cutter.** Used to mill dovetail ways and is used in much the same manner as the T-slot cutter. See **Figure 17-40**.



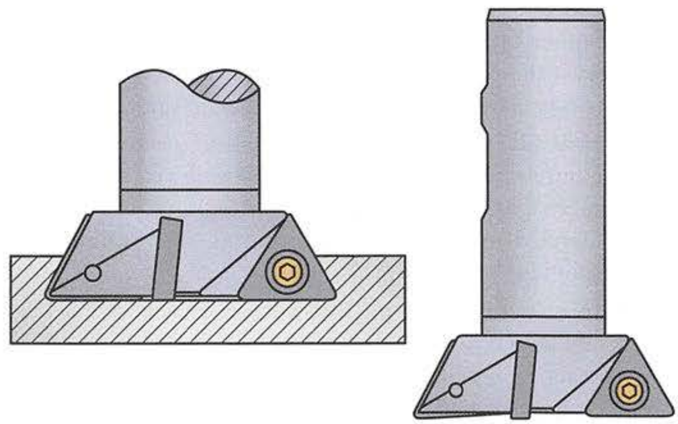
A Goodheart-Willcox Publisher B WMW Machinery Company, Inc.

Figure 17-38. Formed milling cutters. A—Corner-rounding end mill with replaceable tungsten carbide cutting edges. B—Gear cutter. Note that the table is swiveled at an angle to the cutting tool. This feature makes it possible to machine helical-flute workpieces like the one being cut.



Goodheart-Willcox Publisher

Figure 17-39. A T-slot milling cutter and the cut it produces.



Goodheart-Willcox Publisher

Figure 17-40. A dovetail cutter with indexable insert cutting edges.

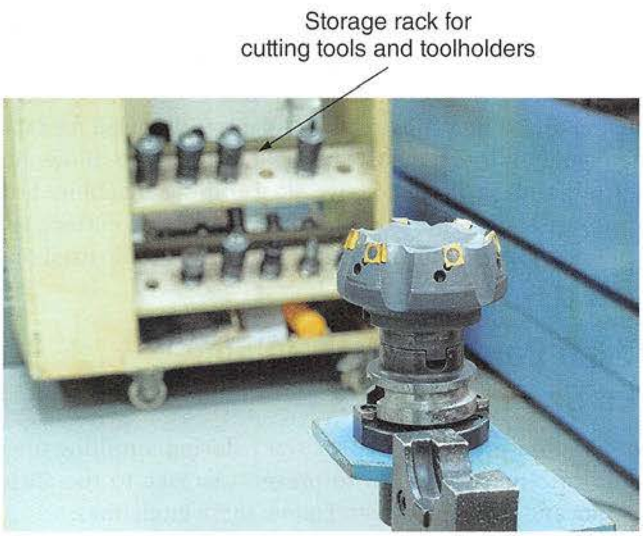
17.4.5 Care of Milling Cutters

Milling cutters are expensive and can be easily damaged if care is not taken in their use and storage. The following recommendations will help extend cutter life:

- Use sharp cutting tools. Machining with dull tools results in low-quality work and eventually damages the cutting edges to such an extent that they cannot be salvaged by grinding.
- Properly support tools and make sure the work is held rigidly.
- Use the correct cutting speed and feed for the material being machined.
- Ensure an ample supply of cutting fluid, **Figure 17-41**.
- Use the correct cutter for the job.
- Store cutters in individual compartments or on wooden pegs. They should never come in contact with other cutters or tools. See **Figure 17-42.**
- Clean cutters before storing them. If they are to be stored for any length of time, it is best to give them a light protective coating of oil.
- Never hammer a cutter onto an arbor. Examine the arbor for nicks or burrs if the cutter does not slip onto it easily. Do not forget to key the cutter to the arbor.
- Place a wooden board under an end mill when removing it from a vertical milling machine. This will prevent cutter damage if it is dropped accidentally. Protect your hand with a heavy cloth or gloves.

Sharnoa Corp.

Figure 17-41. Rigidly supported cutters permit heavier cuts and prolong cutter life. An ample supply of cutting fluid is essential.



PathomP/Shutterstock.com

Figure 17-42. Store cutters in a rack or in individual compartments so that they are not in contact with other tools.

17.5 Holding and Driving Cutters

An ***arbor*** is a device used to hold and drive cutters on metalworking machines. Arbors are made in a number of sizes and styles.

17.5.1 Arbor Styles

Arbors with self-holding tapers were used on some small hand milling machines and on older models of larger millers, but these are seldom used now. There are three basic arbor styles in general use today, **Figure 17-43**:

- Style A is fitted with a small pilot end that runs in a bronze bearing in the arbor support. This style is best used when maximum arbor support clearance is required.
- Style B has a large bearing collar that can be positioned on any part of the arbor. This feature permits heavy cuts by allowing the bearing support to be mounted as close to the cutter as possible for maximum cutter support.
- Style C is used to hold smaller sizes of shell end and face milling cutters that cannot be mounted directly to the spindle nose.

In general, use the shortest arbor possible that will permit adequate clearance between the arbor support and the work.

Both style A and style B arbors have a keyway milled their entire length. This allows a key to be used to prevent the cutter from revolving on the arbor. See **Figure 17-44**.

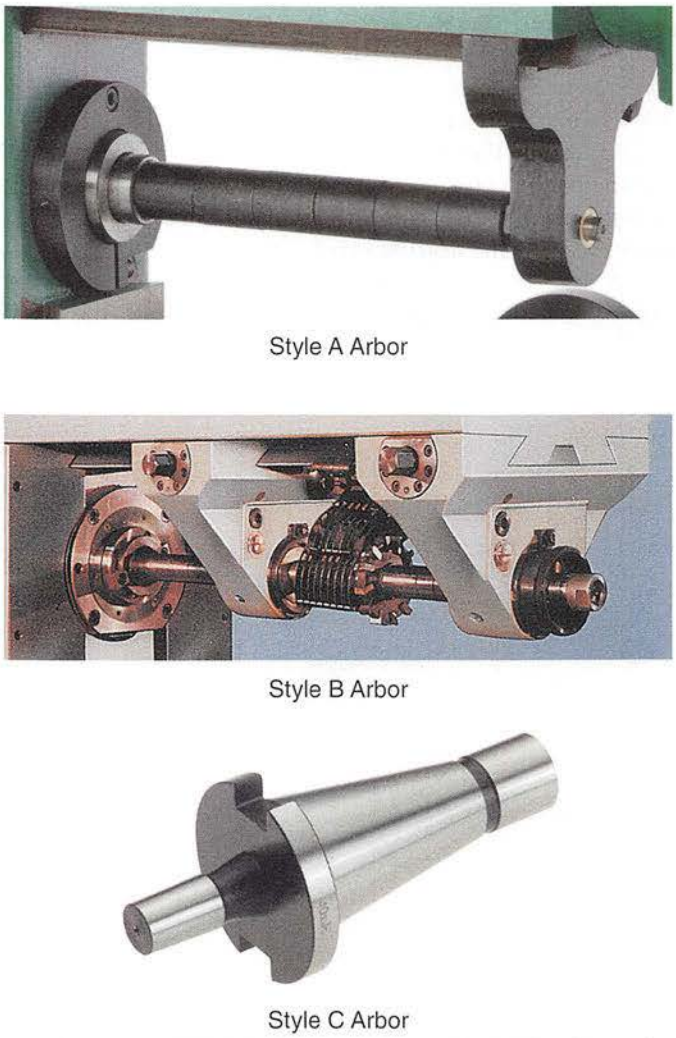
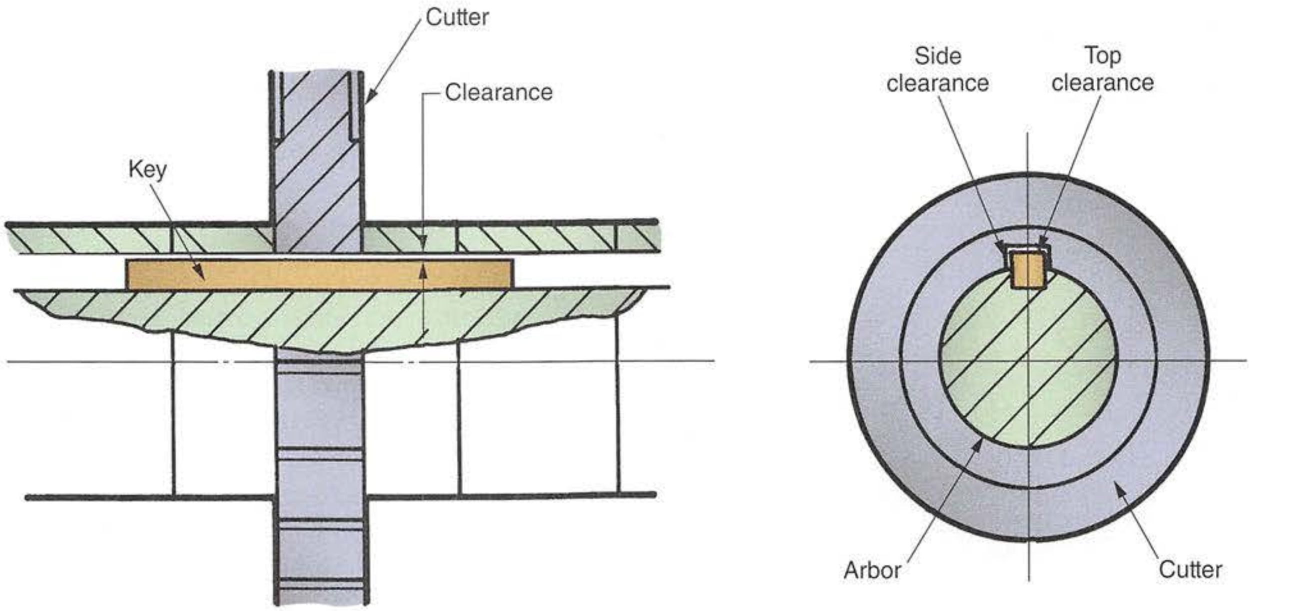


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com; WMW Machinery Company, Inc.

Figure 17-43. Basic arbor styles.



Goodheart-Willcox Publisher

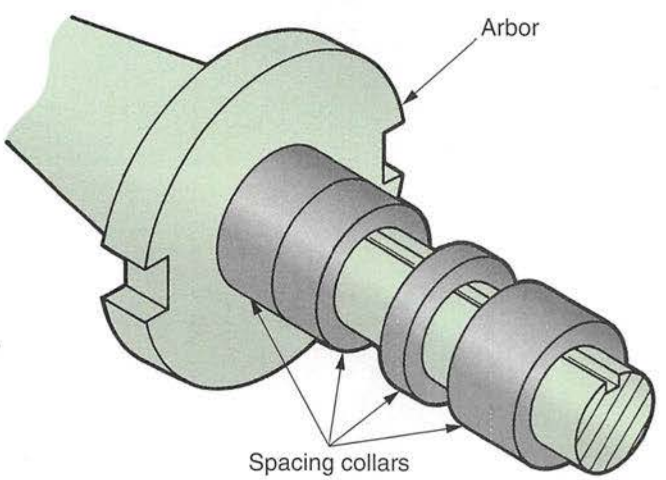
Figure 17-44. Keying the cutter to the arbor prevents it from slipping during the cutting operation.

17.5.2 Other Holding Devices

Accurately made in a number of widths, spacing collars allow two or more cutters to be precisely spaced for gang and saddle milling, **Figure 17-45**. A left-hand threaded nut tightens the cutter and collars on the arbor. The nut should not be tightened directly against the bearing collar on the style B arbor, because the bearing may be damaged.

A draw-in bar is used on most vertical and horizontal milling machines, **Figure 17-46**. It fits through the spindle and screws into the arbor or collet to hold it firmly on the spindle. Drive keys on the nose of the spindle fit into corresponding slots on the arbor, collet, or collet holder to provide positive (nonslip) drive.

End mills may be mounted in spring collets, adapters, shell end mill holders, or stub arbors, depending on the type of work to be done. See **Figure 17-47**. Spring collets accommodate straight shank end mills and drills. Some collets must be fitted in a collet chuck. Adapters are used for taper shank end mills and drills. Shell end mill holders allow shell end mills to be fitted to a vertical milling machine. Stub arbors are short arbors that permit various side cutters, slitting saws, formed cutters, and angle cutters to be used on a vertical mill.



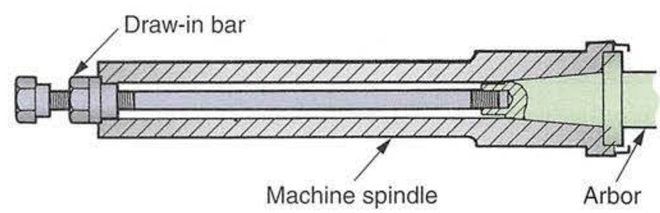
Goodheart-Willcox Publisher

Figure 17-45. Spacing collars are manufactured in many different widths. They are used to position one or more cutters on the arbor.

17.5.3 Care of Cutter Holding and Driving Devices

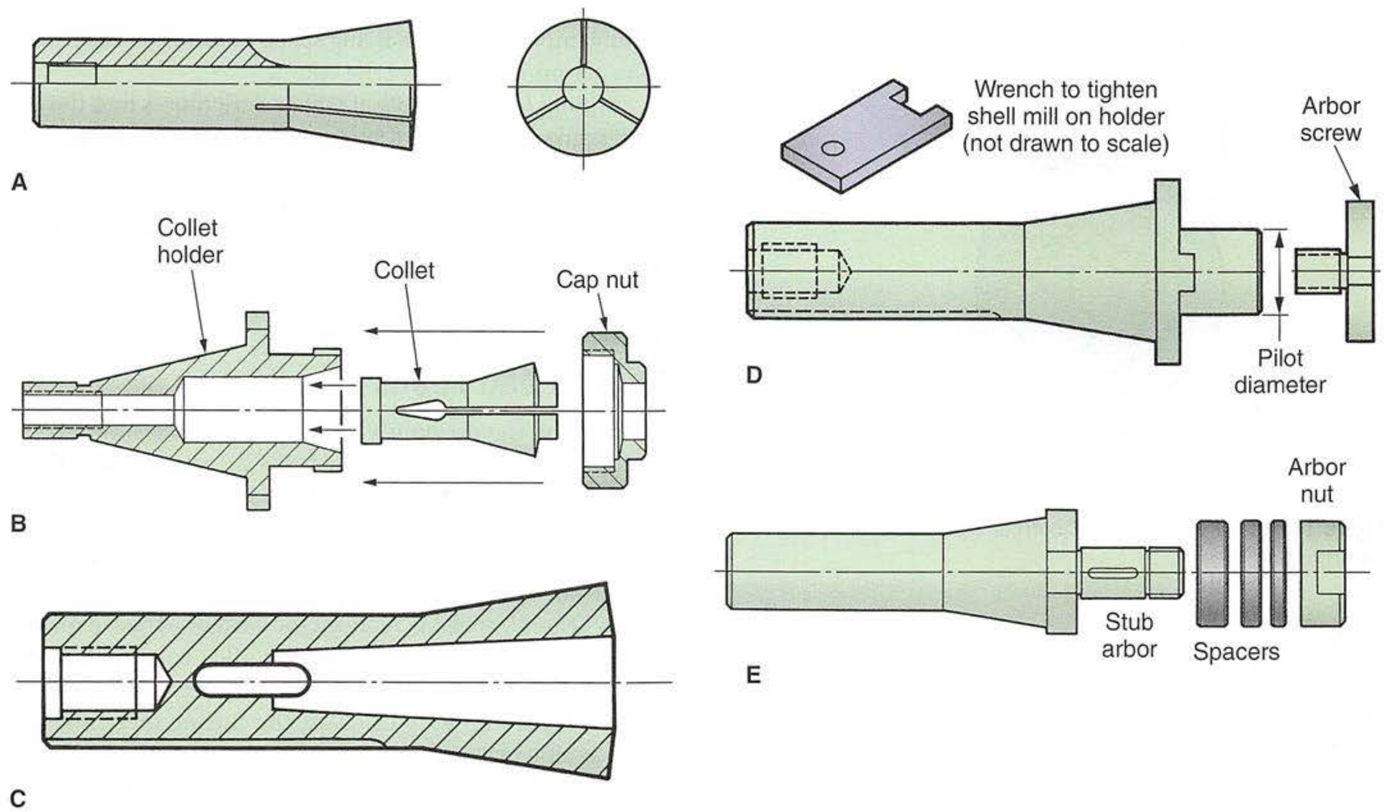
To maintain precision and accuracy during a milling operation, care must be taken to prevent damage to the cutter holding and driving devices. Follow these guidelines:

- Keep the taper end of the arbor and the spindle taper clean and free of nicks.



Goodheart-Willcox Publisher

Figure 17-46. The draw-in bar holds the arbor on the spindle. Avoid operating a milling machine if the arbor is not held in place with a draw-in bar.



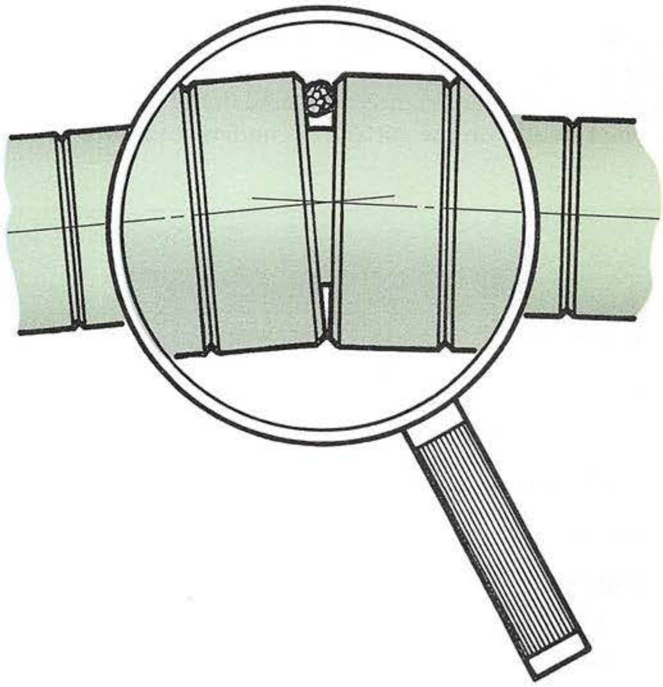
Goodheart-Willcox Publisher

Figure 17-47. Mounting devices. A—Spring collet (R-8 taper). B—Collet chuck and collet. C—Adapter used with taper shank cutting tools. D—Shell end mill holder (R-8 taper). E—Stub arbor (R-8 taper).

- Clean and lubricate the bearing sleeve before placing the arbor support on it, and make sure the bearing sleeve fits snugly.
- Clean the spacing collars before slipping them onto an arbor, **Figure 17-48**. Otherwise, cutter runout will occur, making it difficult to make an accurate cut.
- Store arbors separately and in a vertical position.
- Never loosen or tighten an arbor nut unless the arbor support is locked in place, because this could spring the arbor so that it will not run true.
- Use a wrench of the correct size on the arbor nut, **Figure 17-49**. Make sure at least four threads are engaged before tightening the arbor nut.
- Avoid tightening an arbor nut by striking the wrench with a hammer or mallet. This can crack the nut and distort the threads.
- Do not force a cutter onto an arbor. Check to see what is making it difficult to slide on. Correct any problem.
- Key all cutters to the arbor.

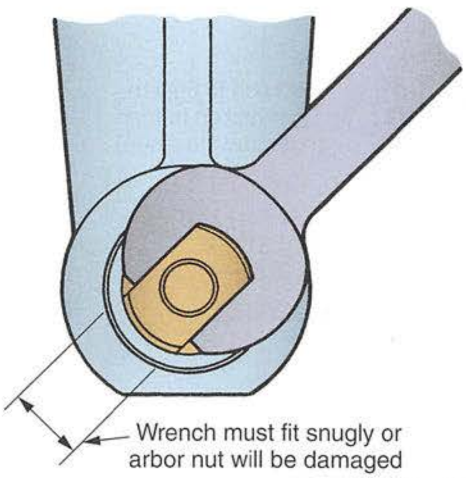
To remove an arbor or adapter from the machine:

1. Loosen the draw-in bar nut a few turns. Do not remove it from the arbor completely.
2. Tap the draw-in bar with a lead hammer to loosen the arbor in the spindle.



Goodheart-Willcox Publisher

Figure 17-48. A chip between spacing collars can cause an arbor to be sprung out of true. The end in the arbor is greatly exaggerated in this drawing.



Goodheart-Willcox Publisher

Figure 17-49. Use a wrench of correct type and size to loosen an arbor nut.

3. Hold the loosened arbor with one hand and unscrew the draw-in bar with the other.
4. Remove the arbor from the spindle. Clean and store it properly.

17.6 Milling Cutting Speeds and Feeds

The time required to complete a milling operation and the quality of the finish is almost completely governed by the cutting speed and feed rate of the cutter. Milling ***cutting speed*** refers to the distance, measured in feet or meters, that a point (tooth) on the cutter's circumference moves in one minute. It is expressed in feet per minute (fpm) or meters per minute (mpm). Milling cutting speed depends on the revolutions per minute (rpm) of the cutter.

Milling ***feed*** is the rate at which work moves into the cutter. It is given in feed per tooth per revolution (ftr). Proper feed rate is probably the most difficult setting for a machinist to determine. In view of the many variables (width of cut, depth of cut, machine condition, cutter sharpness), feed should be as coarse as possible, consistent with the desired finish.

17.6.1 Calculating Cutting Speeds and Feeds

Considering the previously mentioned variables, the speeds listed in **Figure 17-50**, and the feeds listed in **Figure 17-51**, are suggested. The usual procedure is to start with the mid-range figure and increase or decrease speeds until the most satisfactory combination is obtained, consistent with cutter life and surface quality.

In general, speed is reduced for hard or abrasive materials, deep cuts, and metals with high alloy content. Speed is increased for soft materials, better finishes, and light cuts. Refer to **Figure 17-52** to calculate the cutting speed and feed for a specific material.

Example Problem: Determine the approximate cutting speed and feed for a 6″ (152 mm) diameter side cutter (HSS) with 16 teeth, when milling free cutting steel.

Information Available:

Recommended cutting speed for free cutting steel (midpoint in range) = 200 fpm

Recommended feed per tooth (midpoint in range) = 0.008″

Cutter diameter = 6″

Number of teeth on cutter = 16

Material	High-Speed Steel Cutter		Carbide Cutter	
	Feet per Minute	Meters per Minute*	Feet per Minute	Meters per Minute*
Aluminum	550–100	170–300	2200–4000	670–1200
Brass	250–650	75–200	1000–2600	300–800
Low-Carbon Steel	100–325	30–100	400–1300	120–400
Free-Cutting Steel	150–250	45–75	600–1000	180–300
Alloy Steel	70–175	20–50	280–700	85–210
Cast Iron	45–60	15–20	180–240	55–75

Reduce speeds for hard materials, abrasive materials, deep cuts, and high alloy materials. Increase speeds for soft materials, better finishes, light cuts, frail work, and setups. Start at midpoint on the range and increase or decrease speed until best results are obtained.

*Figures rounded off.

Goodheart-Willcox Publisher

Figure 17-50. Recommended cutting speeds for milling. Speed is given in surface feet per minute (fpm) and in surface meters per minute (mpm).

Type of Cutter	Material									
	Aluminum		Brass		Cast Iron		Free-Cutting Steel		Alloy Steel	
End Mill	0.009 0.022	(0.22) (0.55)	0.007 0.015	(0.18) (0.38)	0.004 0.009	(0.10) (0.22)	0.005 0.010	(0.13) (0.25)	0.003 0.007	(0.08) (0.18)
Face Mill	0.016 0.040	(0.40) (1.02)	0.012 0.030	(0.30) (0.75)	0.007 0.018	(0.18) (0.45)	0.008 0.020	(0.20) (0.50)	0.005 0.012	(0.13) (0.30)
Shell End Mill	0.012 0.030	(0.30) (0.75)	0.010 0.022	(0.25) (0.55)	0.005 0.013	(0.13) (0.33)	0.007 0.015	(0.18) (0.38)	0.004 0.009	(0.10) (0.22)
Slab Mill	0.008 0.017	(0.20) (0.43)	0.006 0.012	(0.15) (0.30)	0.003 0.007	(0.08) (0.18)	0.004 0.008	(0.10) (0.20)	0.001 0.004	(0.03) (0.10)
Side Cutter	0.010 0.020	(0.25) (0.50)	0.008 0.016	(0.20) (0.40)	0.004 0.010	(0.10) (0.25)	0.005 0.011	(0.13) (0.28)	0.003 0.007	(0.08) (0.18)
Saw	0.006 0.010	(0.15) (0.25)	0.004 0.007	(0.10) (0.18)	0.001 0.003	(0.03) (0.08)	0.003 0.005	(0.08) (0.13)	0.001 0.003	(0.03) (0.08)

US Customary value expressed in inches per tooth. Metric value (shown in parentheses) expressed in millimeters per tooth.

Increase or decrease feed until the desired surface finish is obtained.

Feeds may be increased 100 percent or more depending on the rigidity of the machine and the power available, if carbide tipped cutters are used.

Goodheart-Willcox Publisher

Figure 17-51. Recommended feed rates in inches per tooth and millimeters per tooth for HSS milling cutters.

Determine the speed setting (cutter rpm) using the information in **Figure 17-52.**

Rule: Divide the feet per minute by the circumference of the cutter, expressed in feet.

$$\text{rpm} = \frac{\text{fpm} \times 12}{\pi D}$$

$$= \frac{200 \times 12}{3.14 \times 6}$$

$$= \frac{2400}{18.84}$$

$$= 127.39 \text{ rpm}$$

Determine feed setting (feed in inches per minute or F) using the information in **Figure 17-52.**

Rule: Multiply feed per tooth per revolution by the number of teeth on the cutter and by speed (rpm).

$$F = \text{ftr} \times T \times \text{rpm}$$

$$= 0.008 \times 16 \times 127$$

$$= 16.25$$

17.6.2 Adjusting Cutting Speed and Feed for Milling

Depending on the make of the milling machine, cutting speed (cutter rpm) and rate of feed (table movement speed) may be changed by any of the following:

- Shifting V-belts.
- Adjusting variable speed pulleys.
- Utilizing a quick-change gearbox and shifting or dialing to the required speed and feed, **Figure 17-53.**

On some machines, speed and feed changes are made hydraulically or electronically. Always make speed changes as specified in the instruction manual for the specific machine being used.

SAFETY NOTE

Make sure the machine has come to a complete stop before attempting to adjust V-belts.

The speed and feed guidelines are only approximate. Set the machine to the closest setting, either higher or lower. Increase or reduce speed until satisfactory cutting conditions are achieved.

17.7 Cutting Fluids

Cutting fluids serve several purposes:

- They carry away heat generated during machining.
- They act as a lubricant.
- They prevent the chips from sticking to or fusing with the cutter teeth.

Rules for Determining Speed and Feed			
To Find	**Having**	**Rule**	**Formula**
Speed of Cutter in Feet per Minute (fpm)	Diameter of cutter and revolutions per minute	Diameter of cutter (in inches) multiplied by 3.1416 (π) multiplied by revolutions per minute, divided by 12	$fpm = \frac{\pi D \times rpm}{12}$
Speed of Cutter in Meters per Minute	Diameter of cutter and revolutions per minute	Diameter of cutter multiplied by 3.1416 (π) multiplied by revolutions per minute, divided by 1000	$mpm = \frac{D(mm) \times \pi \times rpm}{1000}$
Revolutions per Minute (rpm)	Feet per minute and diameter of cutter	Feet per minute, multiplied by 12, divided by circumference of cutter (πD)	$rpm = \frac{fpm \times 12}{\pi D}$
Revolutions per Minute (rpm)	Meters per minute and diameter of cutter in millimeters (mm)	Meters per minute multiplied by 1000, divided by the circumference of cutter (πD)	$rpm = \frac{mpm \times 1000}{\pi D}$
Feed per Revolution (FR)	Feed per minute and revolutions per minute	Feed per minute, divided by revolutions per minute	$FR = \frac{F}{rpm}$
Feed per Tooth per Revolution (ftr)	Feed per minute and number of teeth in cutter	Feed per minute (in inches or millimeters) divided by number of teeth in cutter × revolutions per minute	$ftr = \frac{F}{T \times rpm}$
Feed per Minute (F)	Feed per tooth per revolution, number of teeth in cutter, and rpm	Feed per tooth per revolution multiplied by number of teeth in cutter, multiplied by revolutions per minute	$F = ftr \times T \times rpm$
Feed per Minute (F)	Feed per revolution and revolutions per minute	Feed per revolution multiplied by revolutions per minute	$F = FR \times rpm$
Number of Teeth per Minute (TM)	Number of teeth in cutter and revolutions per minute	Number of teeth in cutter multiplied by revolutions per minute	$TM = T \times rpm$

rpm = Revolutions per minute
T = Teeth in cutter
D = Diameter of cutter
π = 3.1416 (pi)
frm = Speed of cutter in feet per minute
TM = Teeth per minute
F = Feed per minute
FR = Feed per revolution
ftr = Feed per tooth per revolution
mpm = Speed of cutter in meters per minute

Goodheart-Willcox Publisher

Figure 17-52. Rules for determining cutting speed and feed.

- They flush away chips.
- They influence the finish quality of the machined surface.

For safety and for best results, always use the correct cutting fluid for the material you are machining.

17.8 Milling Work-Holding Attachments

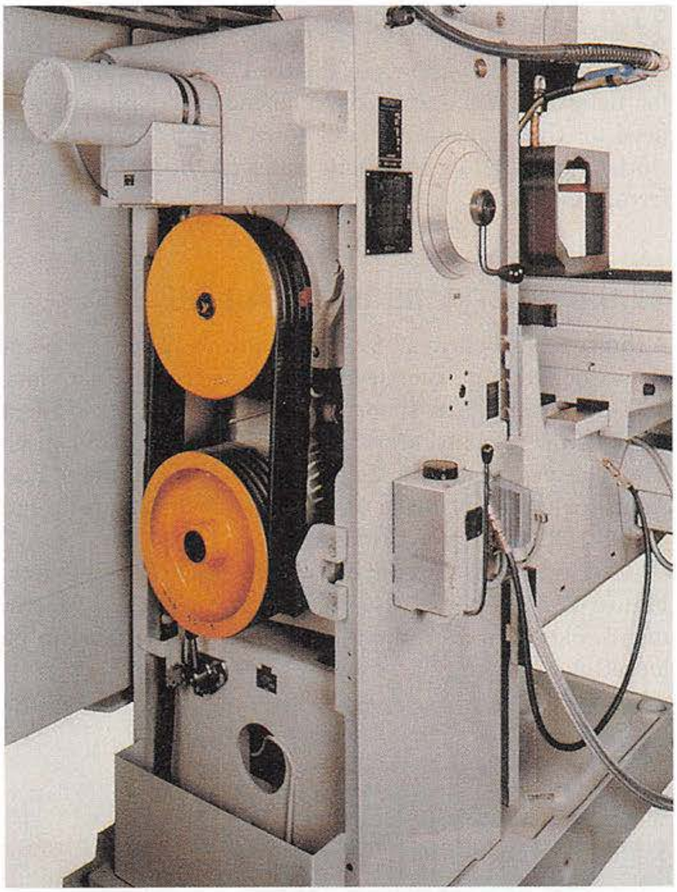
One of the more important features of the milling machine is that it can be used with a large number of work-holding attachments. Each of these attachments increases the usefulness of the milling machine.

17.8.1 Vise

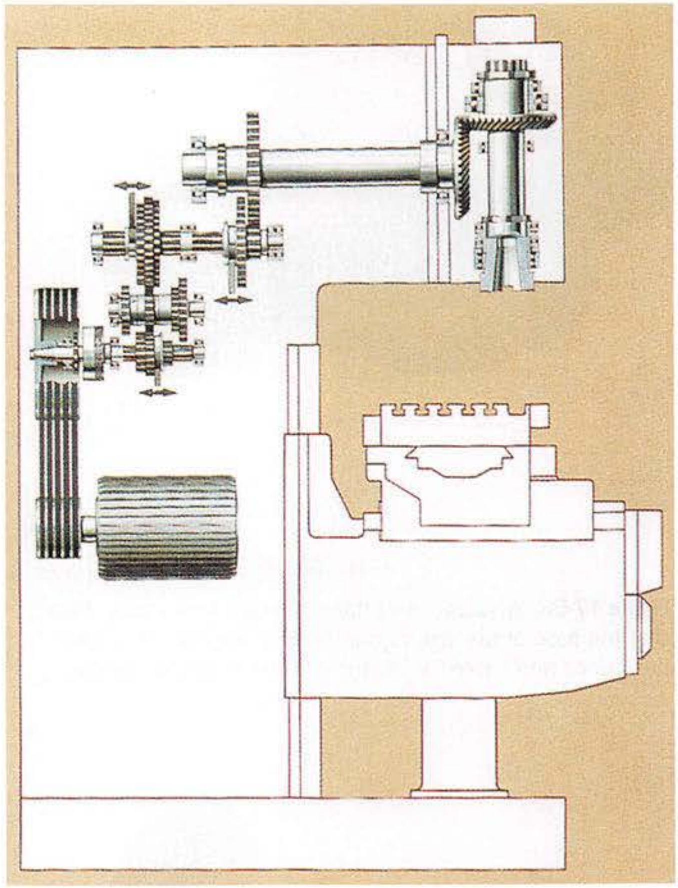
The vise is probably the most widely employed device for holding work for milling. The jaws are hardened to resist wear, and they are ground for accuracy. A milling vise, like other work-holding attachments, is keyed to the table slot with lugs, **Figure 17-54.**

A flanged vise has slotted flanges for fastening the vise to the table, **Figure 17-55**. The slots allow the vise to be mounted on a horizontal milling machine either parallel to the spindle or at right angles to it.

The body of a swivel vise is similar to that of a flange vise, but it is fitted with a circular base that is graduated in degrees. This allows it to be locked at any angle to the spindle. See **Figure 17-56**. The toolmaker's universal vise permits compound or double angles to be machined without complex or multiple setups. See **Figure 17-57**.



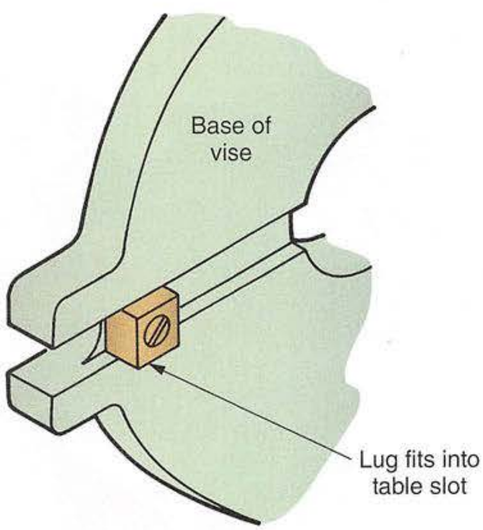
A



B

WMW Machinery Company, Inc.

Figure 17-53. Spindle speed control. A—Main belt drive and spindle speed control dial on a vertical milling machine. B—Cutaway showing the sliding gear mechanism that permits dialing any of 18 speed choices.



Goodheart-Willcox Publisher

Figure 17-54. Lugs on the base of a work-holding attachment position the device on the worktable.

© Copyright Jergens, Inc. 2013

Figure 17-55. A typical flanged vise.

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

Figure 17-56. A swivel vise rotates to align work easily. Note that the base of the vise is graduated in degrees. It can be positioned and locked at an angle to the machine spindle.

17.8.2 Magnetic Chuck

A magnetic chuck, shown in **Figure 17-58**, is ideally suited for many milling operations. The magnet eliminates the need for time-consuming hold-down clamps to mount the work to the table. The magnetic chuck can be used only with ferrous metals.

17.8.3 Rotary and Index Tables

A rotary table, **Figure 17-59**, can perform a variety of operations, including cutting segments of circles, circular slots, and cutting irregular-shaped slots. A dividing attachment can be fitted to many rotary tables in place of the handwheel. The table is graduated in degrees around its circumference. Adjustments can be made accurately with the handwheel to within 1/30 of a degree (2 minutes).

An index table permits the rapid positioning of work, **Figure 17-60**. Indexing is usually performed in 15° increments. However, a clamping device allows the table to be locked at any setting.

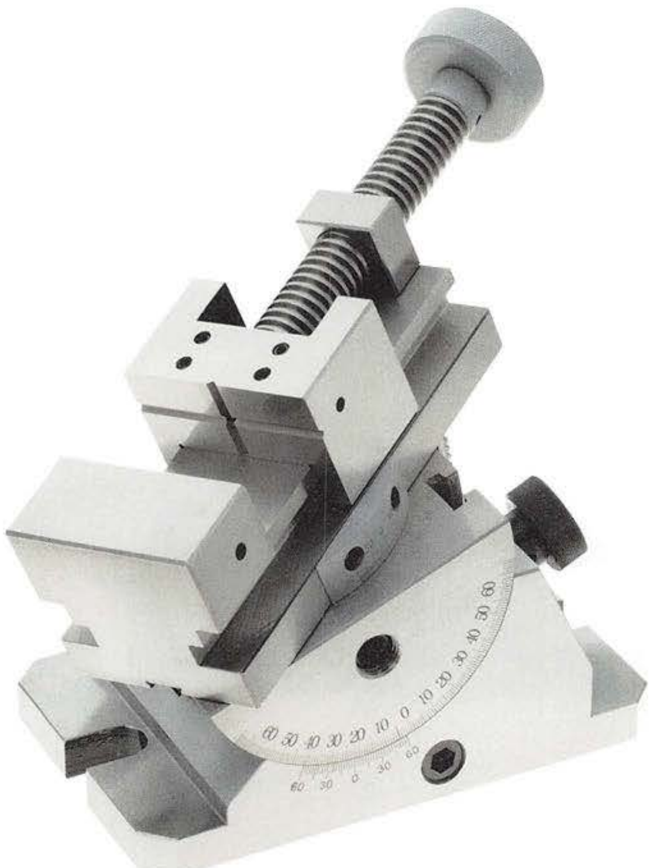
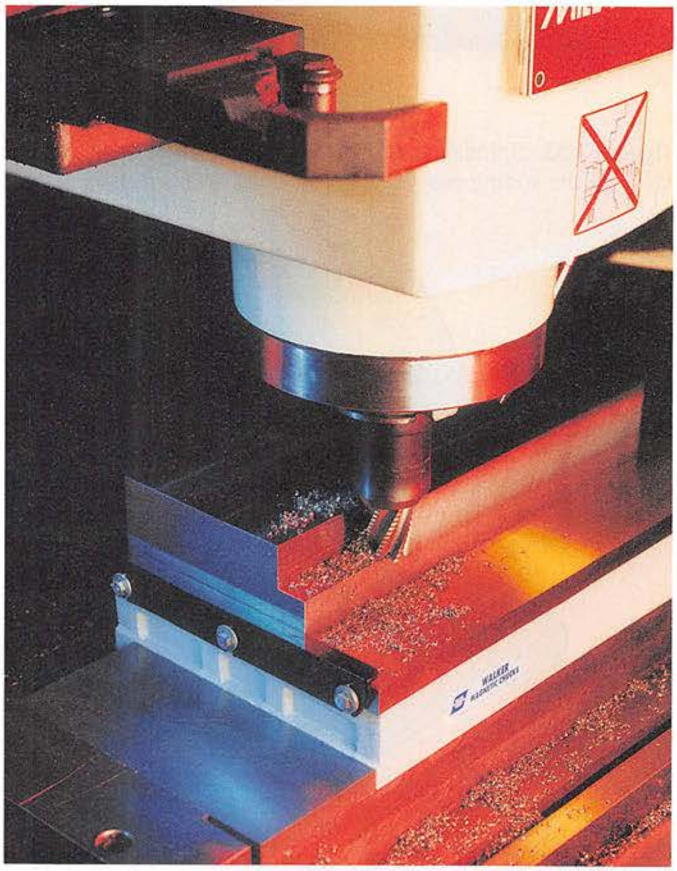


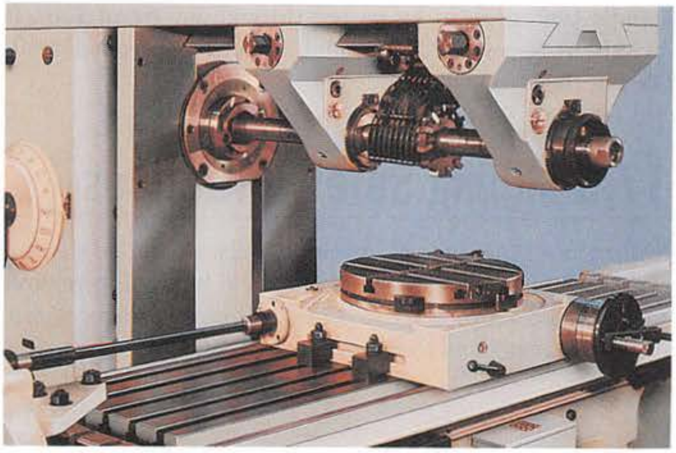
Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 17-57. The toolmaker's universal vise can be pivoted on several planes.



O.S. Walker Co.

Figure 17-58. Milling a workpiece held with a magnetic chuck.



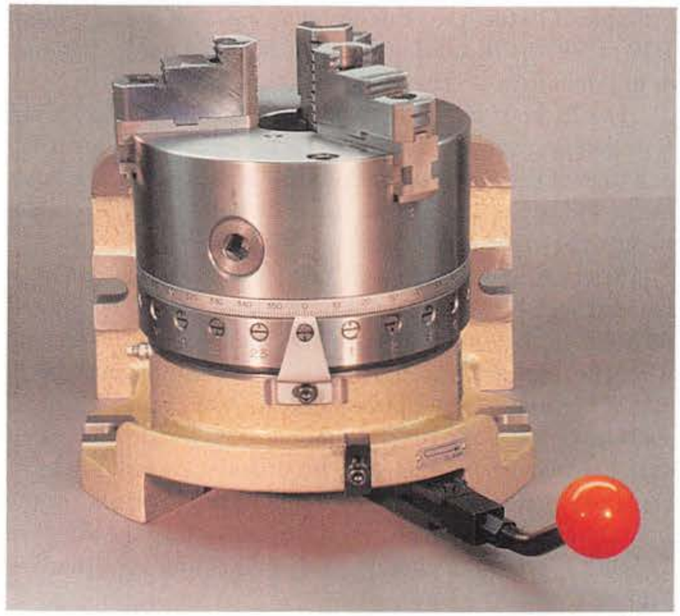
WMW Machinery Company, Inc.

Figure 17-59. A rotary table can be used to machine parts with round and curved shapes.

17.8.4 Dividing Head

A dividing head divides the circumference of circular work into equally spaced units. This feature makes a dividing head indispensable when milling gear teeth, cutting splines, and spacing holes on a circle. It also makes possible the milling of squares, hexagons, and various other regular shapes. A dividing head is a precision device that has an indexing accuracy of about one minute of arc. This is the equivalent of 1/21,600 part of a circle.

The dividing head consists of two parts: the dividing unit and the footstock. Work may be mounted between centers, in a chuck, or in a collet. See **Figure 17-61**. An index plate, identified by circles of holes on its face, and the index crank, which revolves on the index plate, are fundamental in the dividing operation.

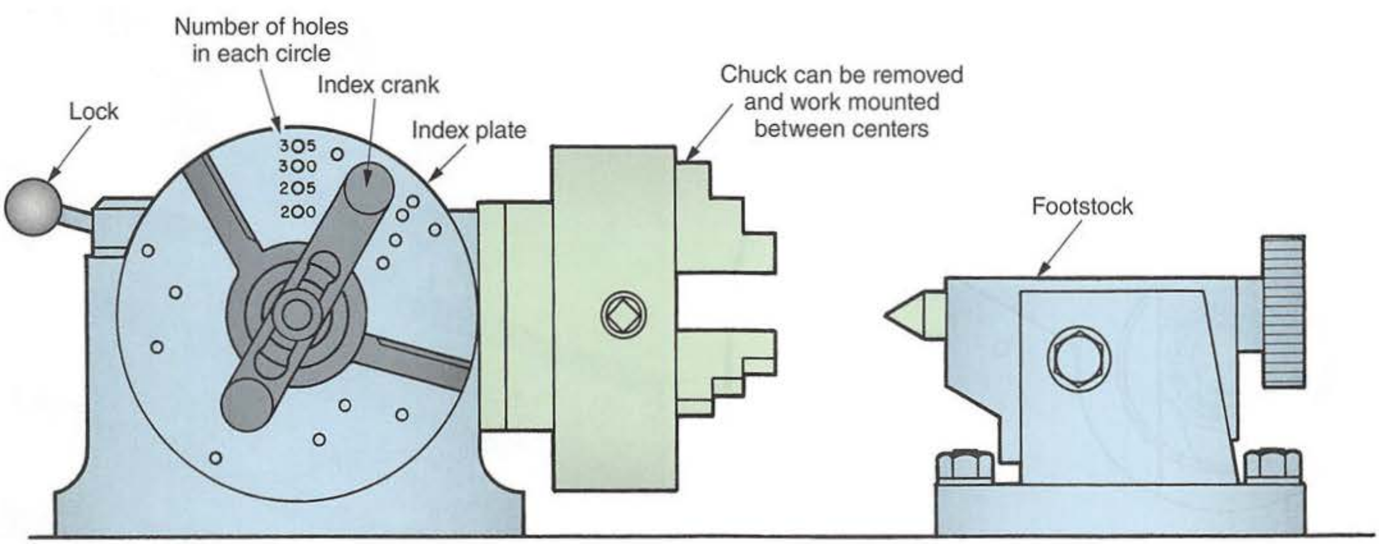


Yukiwa Seiko USA, Inc.

Figure 17-60. An index table permits work to be positioned rapidly. The 3-jaw chuck is used for round workpieces. Note that the unit can be mounted on the milling machine vertically or horizontally.

Rotating the index crank causes the dividing head spindle (to which the work is mounted) to rotate. The standard ratios for the dividing head are five turns of the index crank for one complete revolution of the spindle (5:1) or 40 turns of the index crank for one revolution of the spindle (40:1).

The ratio between index crank turns and spindle revolutions, plus the index plate with its series of equally spaced hole circles, makes it possible to divide the circumference of the work into the required number of equal spaces. For



Goodheart-Willcox Publisher

Figure 17-61. Dividing head and footstock. Work can be mounted between centers, in a collet, or in a chuck, as shown. The index plate and index head are fundamental to the dividing operation.

example, if 10 teeth were to be cut on a gear, it would require 1/10 of 5 turns (assuming the dividing head has a 5:1 ratio), or one-half turn of the index crank for each tooth.

For 25 teeth, the number of crank turns would be 1/25 of 5, or dividing 5 by 25. By further reduction, this becomes 1/5 of a turn of the crank for each tooth. This is where the holes in the index plate come into use. The holes allow fractional turns to be made accurately.

Select an index plate with a series of holes divisible by 5. On one such plate, the circles have 47, 49, 51, 53, 54, 57, and 60 holes. In this situation, 60 is divisible by 5. Thus, indexing would be through 12 holes on the 60-hole circle for each tooth.

When indexing, it is not necessary to count 12 holes each time the work is repositioned after a tooth has been cut. Two arms, called *sector arms* or *index fingers*, are loosened and positioned. One is located against the pin on the index crank. The other is moved clockwise until the arms are 12 holes apart, not including the hole that holds the pin. See **Figure 17-62**.

To index, first move the workpiece clear of the cutter. Disengage the crank by withdrawing the pin from the index plate and rotating it clockwise through the section marked by the sector arms. Drop the pin into the hole at the position of the second sector arm and lock the dividing head mechanism. Next, move the sector arms in the same direction as crank rotation to catch up with the pin in the index crank. For each cut, repeat this operation. The dividing head spindle typically can be moved through an arc of 100°. It swivels 5° below horizontal and 5° beyond perpendicular. See **Figure 17-63**.

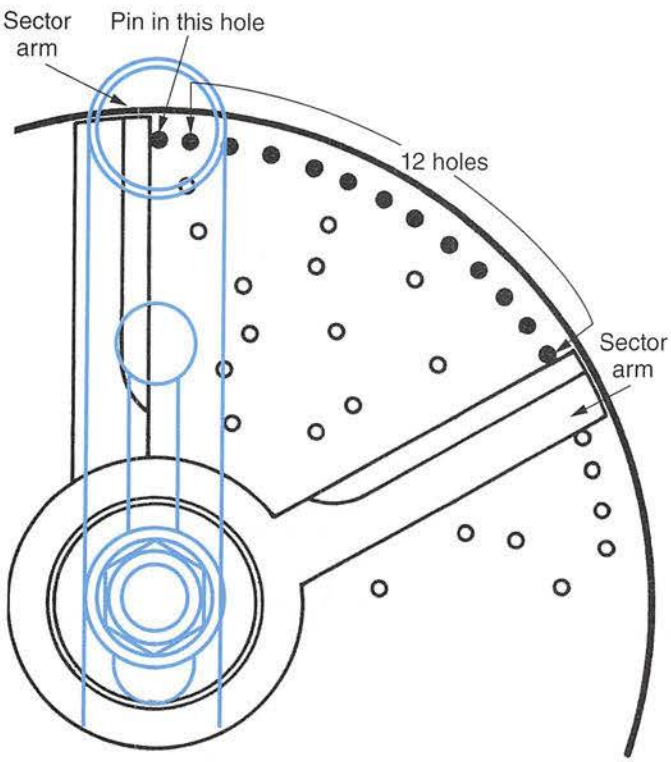


Figure 17-62. Positioning sector arms for proper movement of the spindle. When positioning sector arms, do not count the hole that holds the pin on the index crank.

17.9 Milling Safety Practices

Milling machines, like all machine tools, should be cleaned after each work session. A brush should be used to remove accumulated chips, **Figure 17-64**. Chips are razor-sharp. Never use your hand to remove them.

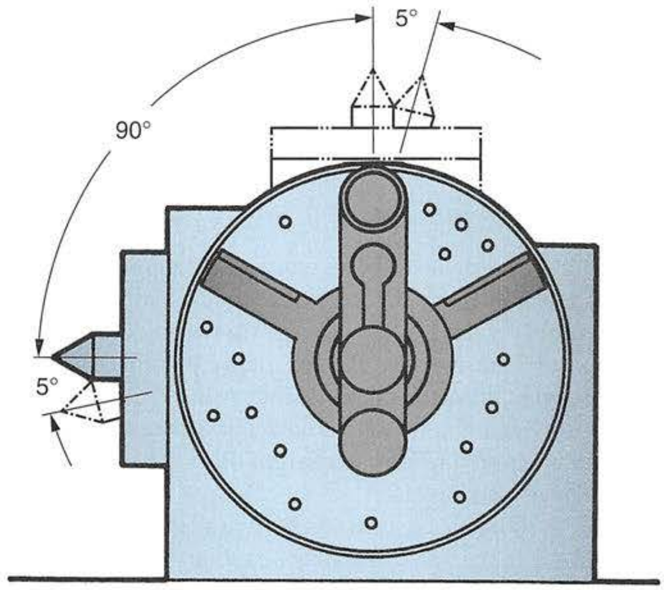


Figure 17-63. Most dividing heads can pivot the spindle through an arc of 100°.

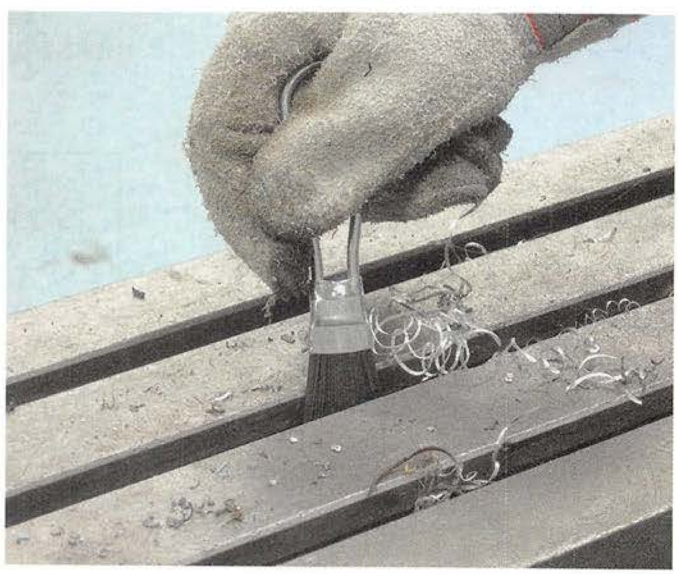


Photo courtesy of Weiler Corporation

Figure 17-64. Use a brush to remove metal chips. Never use your hand, because sharp chips can cause serious cuts.

Never remove chips with compressed air. The flying chips may injure you or a nearby person. Also, if cutting oil was used in the machining operation, the compressed air will create a highly flammable oily mist. If ignited by an open flame, the mist can explode. Finish by wiping down the machine with a soft cloth.

The following procedures are suggested for the safe operation of a milling machine.

- Become thoroughly familiar with the milling machine before attempting to operate it. When in doubt, obtain additional instructions.
- Never attempt to operate a milling machine while your senses are impaired by medication or other substances.
- Wear appropriate clothing and approved safety glasses.
- Stop the machine before making adjustments or measurements, or trying to remove accumulated chips.
- Be sure all power to the machine is turned off before opening or removing guards and covers.
- Use a piece of heavy cloth or gloves for protection when handling milling cutters. Avoid using your bare hands.
- Get help to move any heavy machine attachment, such as a vise, dividing head, rotary table, or large work.
- Be sure the work-holding device is mounted solidly to the table and the work is held firmly. Spring or vibration in the work can cause thin cutters to jam and shatter.
- Before engaging a rotating tool against the workpiece, take the time to verify that the spindle is rotating in the correct direction.
- Never reach over or near a rotating cutter.
- Avoid talking with anyone while operating a machine tool. Do not allow anyone to turn on the machine for you.
- Never "fool around" when operating a milling machine. Keep your mind on the job and be ready for any emergency.
- Keep the floor around the machine clear of chips, and wipe up spilled cutting fluid immediately. Place sawdust or oil-absorbing compound on slippery floors. Place all oily rags in an approved metal container that can be closed tightly.
- Be thoroughly familiar with the location of the machine's emergency stop switch, lever, or button.
- Treat any small cuts and skin punctures as potential infections. Clean them thoroughly. Apply antiseptic and cover the injury with a bandage. Report any injury, no matter how minor, to your instructor or supervisor.
- Launder work clothes frequently. Greasy clothing is a fire hazard.

Chapter Review

Summary

- Milling machines can be grouped into two general families: fixed-bed and column-and-knee.
- The two main categories of milling operations are face milling and peripheral milling.
- Milling cutters can be classified as either a solid cutter or an inserted-tooth cutter and can be made of a variety of materials to suit the cutting needs.
- Great care should be taken of milling tools to ensure precision and accuracy.
- Arbors are used to hold and drive milling cutters.
- Cutting speed and rate of feed can be changed in several different ways, depending on the type of milling machine.
- For safety and for best results, it is important to use the correct cutting fluid for the material you are machining.
- Work-holding attachments, such as vises, magnetic chucks, rotary tables, index tables, and dividing heads, provide the milling machine with much of its versatility.

Review Questions

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

1. The two broad classifications of milling machines are fixed-bed and _____ milling machines.
2. A(n) _____ milling machine has a horizontal spindle, and the worktable has three movements.
3. A(n) _____ milling machine is similar to the machine in question 2, but a fourth type of movement allows the worktable to cut spiral shapes.
4. The spindle of a(n) _____ milling machine is perpendicular or at right angles to the worktable.
5. List the four methods of machine control.
6. The surface being machined is parallel with the cutter face in _____ milling.
7. The surface being machined is parallel with the periphery of the cutter in _____ milling.
8. In conventional or up-milling, _____.
   A. the work is fed into the rotation of the cutter
   B. the work moves in the same direction as the rotation of the cutter
   C. Neither of the above.
9. In climb or down-milling, _____.
   A. the work is fed into the rotation of the cutter
   B. the work moves in the same direction as the rotation of the cutter
   C. Neither of the above.
10. What are the two general types of milling cutters?
11. What do the terms *left-hand* and *right-hand* describe, in reference to an end mill?
12. How should milling cutters be stored?
13. What is a draw-in bar, and how is it used?
14. The distance, measured in feet or meters, that a point (tooth) on the circumference of a cutter moves in one minute, is known as the _____.
15. The rate at which the work moves into the cutter is known as the _____.
16. Calculate machine speed (rpm) and feed (F) for a 1.5″ diameter tungsten carbide 5 tooth (T) end mill when machining cast iron. The recommended cutting speed is 190 fpm and feed per tooth (ftr) is 0.004″. Use the following formulas:

$$rpm = \frac{fpm \times 12}{\pi D}$$

and

$$F = ftr \times T \times rpm$$

17. Determine machine speed (rpm) and feed (F) for a 2.5″ diameter HSS shell end mill with 8 teeth (T) machining aluminum. Recommended cutting speed is 550 fpm. Feed per tooth (ftr) is 0.010″.
18. Calculate machine speed (rpm) for machining aluminum with a 6″ diameter HSS side milling cutter. Recommended cutting speed is 550 fpm.
19. Determine machine speed (rpm) and feed (F) for a 4″ diameter HSS side milling cutter with 16 teeth (T) milling free cutting steel. Recommended cutting speed is 200 fpm. Feed per tooth (ftr) is 0.005″.
20. Calculate machine speed (rpm) and feed (F) for a 2.5″ diameter HSS slab milling cutter with 8 teeth (T) machining brass. Recommended cutting speed is 250 fpm. Feed per tooth (ftr) is 0.006″.
21. Cutting fluids serve several purposes. List three of them.
22. Name two reasons why chips should never be removed using compressed air.

23. Before removing a machine's guards or covers, be sure that all power to the machine is _____.
24. Milling cutters are sharp. Protect your hands with a cloth or _____ when handling them.
25. Treat all small cuts and skin punctures as potential sources of infection. Which of the following should be done?
    A. Clean them thoroughly.
    B. Apply antiseptic and cover with a bandage.
    C. Promptly report the injury to your instructor.
    D. All of the above.
    E. None of the above.

For questions 26–35, match each term with the correct sentence below.

26. Can be fed into work like a drill.
27. Recommended for conventional milling when plunge cutting (going into work like a drill) is *not* required.
28. Mounts on a stub arbor.
29. Intended for machining large, flat surfaces parallel to the cutter face.
30. A facing mill with a single-point cutting tool.
31. Cylindrical cutter with teeth located around the circumference.
32. Cutter with helical teeth designed to cut with a shearing action.
33. Has cutting teeth on the circumference and on one or both sides.
34. Has alternate right-hand and left-hand helical teeth.
35. Thin milling cutter designed for machining narrow slots and for cutoff operations.

A. Side milling cutter
B. Slab cutter
C. Fly cutter
D. Two-flute end mill
E. Plain milling cutter
F. Multiflute end mill
G. Face milling cutter
H. Metal slitting saw
I. Shell end mill
J. Staggered-tooth side cutter

For questions 36–43, match each term with the correct sentence below.

36. Can only be mounted parallel to or at right angles on worktable.
37. Has a circular base graduated in degrees.
38. Permits compound angles (angles on two planes) to be machined without complex or multiple setups.
39. Can only be used with ferrous metals.
40. Needed when cutting segments of circles, circular slots, and irregular-shaped slots.
41. Permits rapid positioning of circular work in 15° increments and can be locked at any angular setting.
42. Used to divide circumference of round work into equally spaced divisions.
43. Arms that are loosened and repositioned to divide a circumference equally.

A. Index fingers
B. Flanged vise
C. Rotary table
D. Magnetic chuck
E. Swivel vise
F. Index table
G. Dividing head
H. Universal vise

CHAPTER **18**

Milling Machine Operations

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Describe how milling machines operate.
- Set up and safely operate vertical milling machines.
- Set up and safely operate horizontal milling machines.
- Explain the basic care of a milling machine.
- Make the needed calculations and cut spur gears.
- Make the needed calculations and cut a bevel gear.
- Describe current thread milling procedures.
- Point out safety precautions that must be observed when operating a milling machine.
- List industrial applications for milling operations.
- Describe high-velocity machining.

Technical Terms

addendum
bevel gear
circular pitch
dedendum
diametral pitch
gang milling
machine backlash
pitch circle
slitting
slotting
spur gear
straddle milling

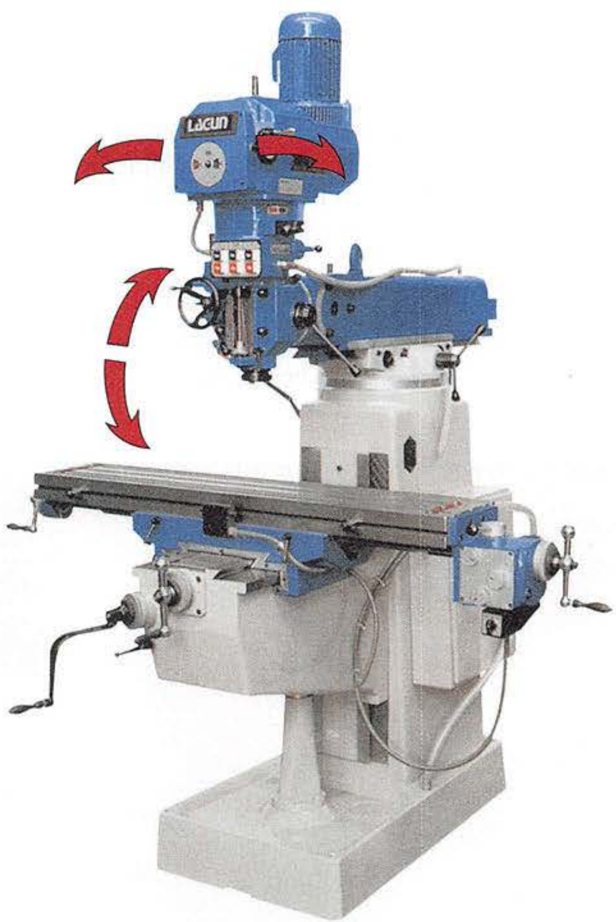
The milling machine family is versatile enough to perform so many different machining operations that it is not possible to cover all of them in a book of this type. This chapter describes the basic milling operations.

18.1 Vertical Milling Machine

The vertical milling machine can perform milling, drilling, boring, and reaming operations. It differs from the horizontal mill in that the spindle is mounted in a vertical position.

18.1.1 Angular Head Adjustments

The spindle head swivels 90° left or right for machining at any angle, **Figure 18-1**. The ram on which it is mounted can be adjusted in and out. On many vertical mills, it also revolves 180° on a horizontal plane. Both swivels are graduated in degrees with a vernier scale to ensure accurate angular settings.



Republic-Lagun Machine Tool Co.

Figure 18-1. Angular head adjustments are possible on many vertical milling machines.

18.1.2 Cutters for Vertical Milling Machines

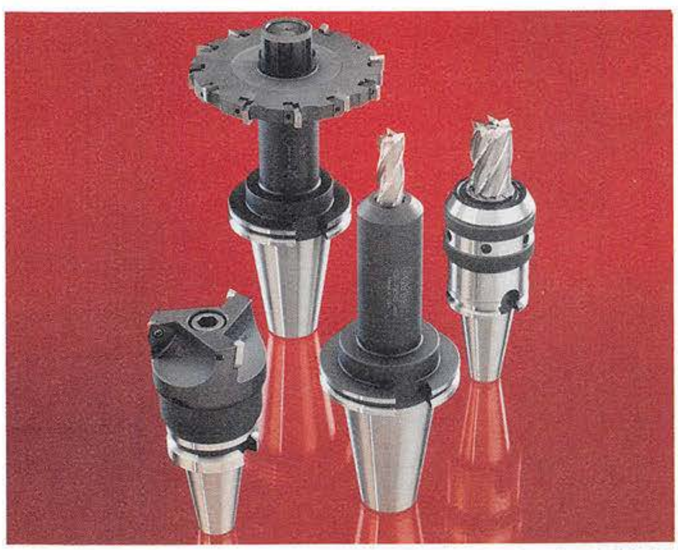
Adapters are available that permit the use of side and angle cutters, **Figure 18-2**. However, the cutters most often used in vertical milling machines are face mills and end mills.

Taper shank end mills and drills are fitted in an adapter, **Figure 18-3A**. Some machine spindles have a Brown & Sharpe (B&S) taper. When taper shanks are large enough, they are mounted directly, **Figure 18-3B**. When a taper is too small to fit directly into the spindle, a sleeve must be used. Straight shank end mills are held in a spring collet, **Figure 18-3C**, or in an end mill adapter, **Figure 18-3D**. Small drills, reamers, and similar tools are held in a standard Jacobs chuck fitted to the spindle by one of the above methods.

18.2 Vertical Milling Machine Operations

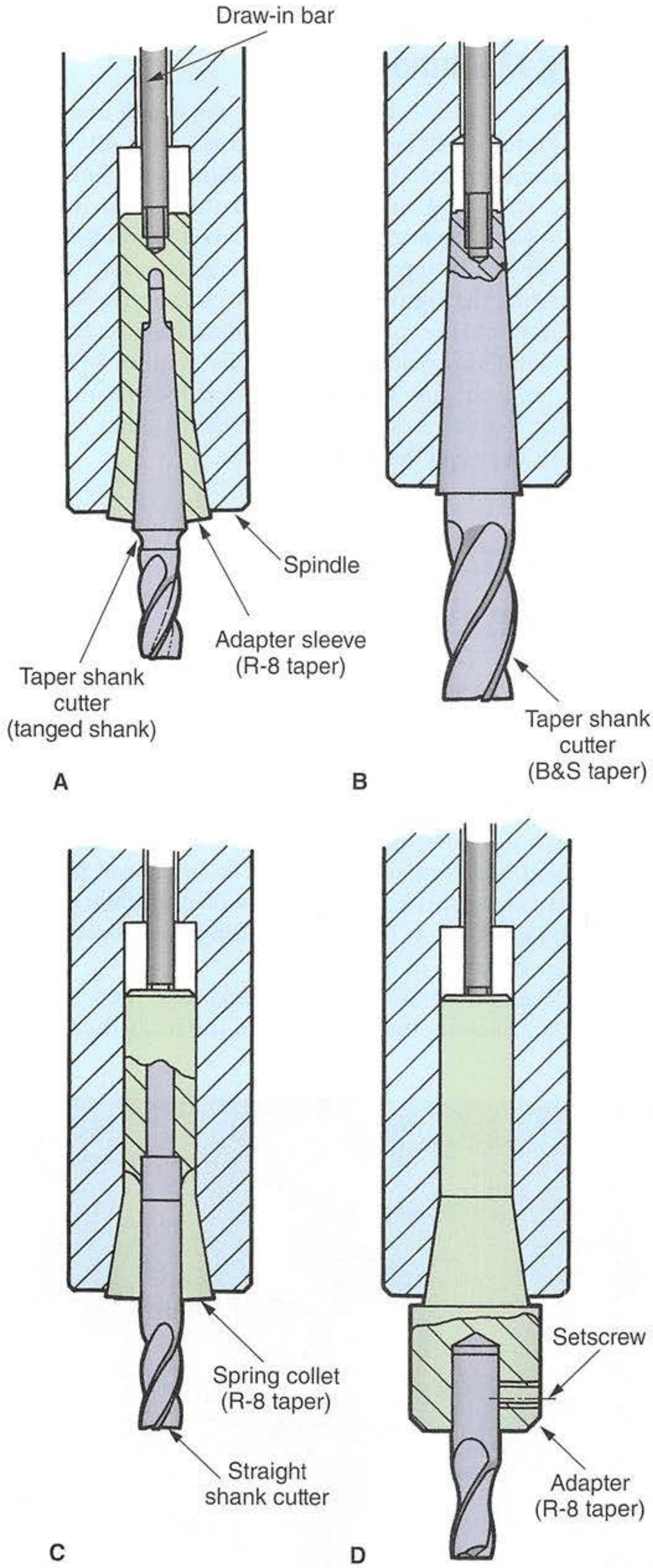
In addition to the usual precautions that must be observed when preparing a machine tool for a job, the spindle head alignment on a vertical milling machine must be checked. Make sure that the spindle head is at an exact right angle (perpendicular) to the worktable. If the spindle is not perpendicular, it is not possible to machine a flat surface. The surface will be irregular or "dished." Spindle head perpendicularity can be checked with the use of a dial indicator, as shown in **Figure 18-4**. The device holding the indicator may be made in the shop or purchased.

If a vise is used to mount the work, wipe the vise base and worktable clean. Inspect for burrs and nicks that would



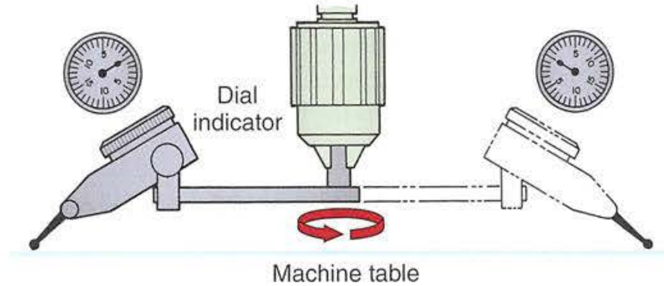
Parlec, Inc.

Figure 18-2. Adapters permit arbor cutters and other types of cutters to be mounted on a vertical milling machine.



Goodheart-Willcox Publisher

Figure 18-3. Four of the most common methods to mount end mills in a vertical milling machine. A—Adapter sleeve with taper shank cutter. B—B&S taper mounted directly in the spindle. C—Spring collet with straight shank cutter. D—Adapter with setscrew on straight shank cutter.



Goodheart-Willcox Publisher

Figure 18-4. A dial indicator can be used as shown to check whether the milling head is perpendicular to the table.

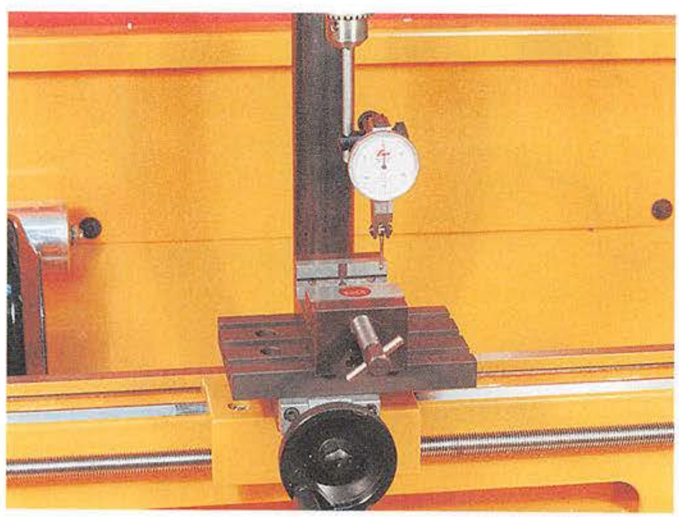
prevent the vise from seating properly on the table. Bolt the vise firmly to the machine.

If extreme accuracy is required, the next step is to align the vise with a dial indicator, **Figure 18-5**. However, for many jobs the vise can be aligned with a square, as shown in **Figure 18-6**. Angular settings can be made with a protractor, **Figure 18-7**, or by using the degree divisions on the base of a swivel vise.

If you have not already done so, wipe the vise jaws and bottom clean of chips and dirt. Place clean parallels in the vise and place the work on them. Tighten the jaws and tap the work onto the parallels with a mallet or soft-face hammer. Use thin paper strips to check whether the work is firmly on the parallels, **Figure 18-8**. Never strike the vise handle with a hammer or mallet to put additional holding pressure on the jaws. If the workpiece is rough, protect the vise jaws and parallels with soft metal strips.

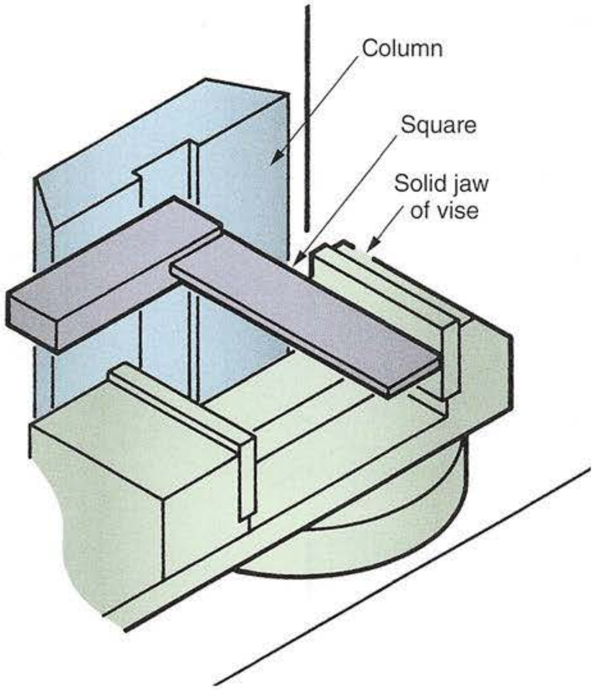
18.2.1 Checking for Backlash

This is also a good time to check the axes for machine backlash. ***Machine backlash*** is a hesitation or failure to begin



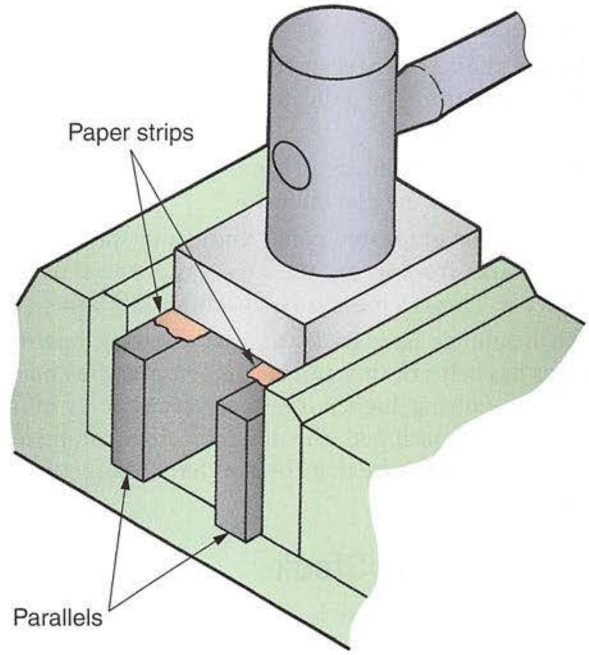
Goodheart-Willcox Publisher

Figure 18-5. Use of a dial indicator permits extreme accuracy in aligning a solid vise jaw.



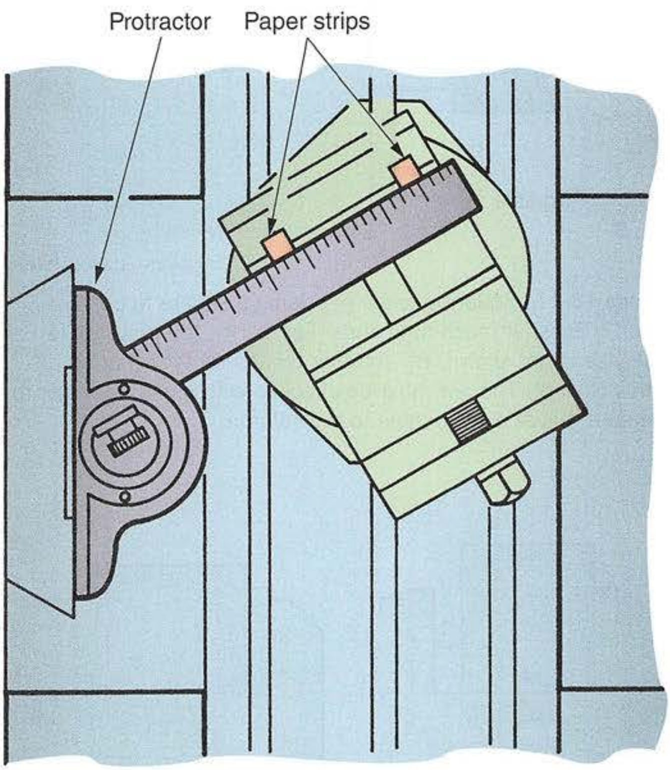
Goodheart-Willcox Publisher

Figure 18-6. Procedure for squaring a solid vise jaw using a machinist's steel square.



Goodheart-Willcox Publisher

Figure 18-8. Setting work on the parallels with a soft-face hammer. Use thin paper strips to check whether the work is firmly on the parallels.



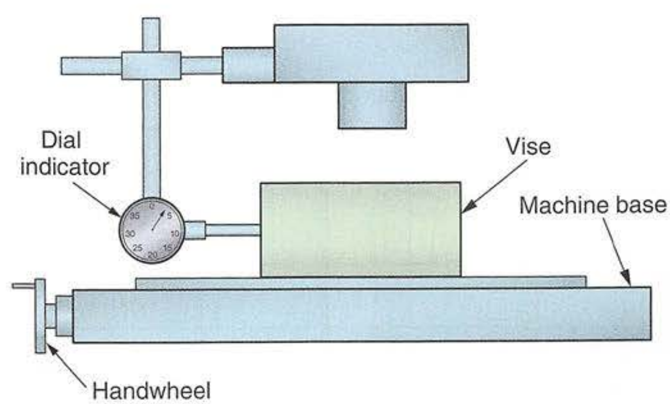
Goodheart-Willcox Publisher

Figure 18-7. Angular settings can be made with a protractor head and the steel rule of a combination set. Use paper strips to help determine whether the setting is accurate.

moving immediately when an axis is reversed. It is caused by the clearance between the drive screw and the mating nuts that move the axes. Over time, wear increases machine backlash.

Machine backlash can easily be checked by moving the axis being inspected to one end of its travel. Follow this procedure (**Figure 18-9**):

1. Mount a dial indicator onto a fixed component on the machine and zero the indicator against the table or a tool mounted to the table, such as a machine vise.



Goodheart-Willcox Publisher

Figure 18-9. Setup for checking machine backlash.

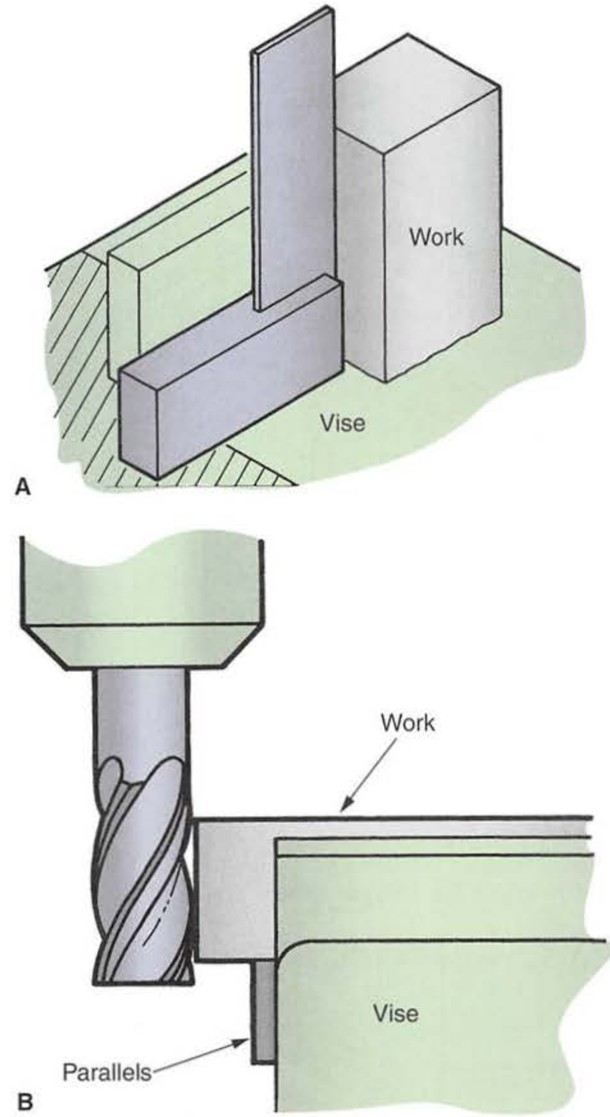
2. Zero one of the graduated collars inside the handwheel of the axis.
3. Rotate the handwheel in the direction that moves the table away from the indicator until the table reaches the end of its travel.
4. Return the table to its original position until the tool zeroes against the dial indicator.
5. Look at the graduated collar and count the number of tick marks from where the axis was at the start of the process. This is a measure of machine backlash.

All machines have backlash, and knowing how much each axis has helps the machinist compensate when machining parts requiring higher levels of precision. If multiple pieces must be machined, the dial indicators can be left in place and used to ensure that the axes have been returned to the proper location.

18.2.2 Squaring Stock

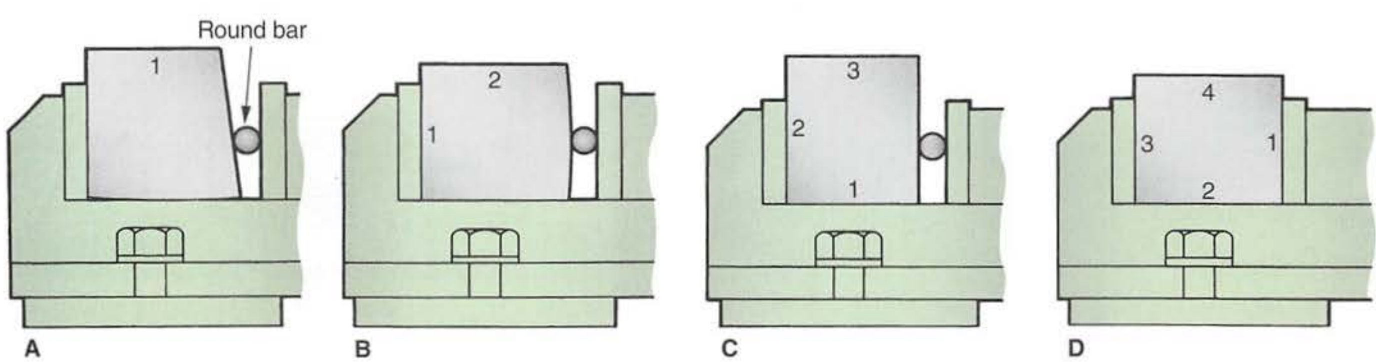
A specific sequence must be followed to machine several surfaces of a piece square with one another, as shown in **Figure 18-10**. Follow these steps:

1. Machine the first surface.
2. Remove the burrs and place the first machined surface against the fixed vise jaw. Insert a piece of soft metal rod between the work and movable jaw if that portion of the work is rough or not square.
3. Machine the second surface.
4. Remove the burrs and reposition the work in the vise to machine the third side. This side must be machined to dimension. Take a light cut and use a micrometer to measure for size. The difference between this measurement and the required thickness is the amount of material that must be removed.
5. Repeat the above operation to machine the fourth side.
6. If the piece is short enough, the ends may be machined by placing it in a vertical position with the aid of a square, **Figure 18-11A**. Otherwise, it may be machined as shown in **Figure 18-11B**.



Goodheart-Willcox Publisher

Figure 18-11. Squaring ends. A—Using a square to position short pieces for machining ends. For clarity, the movable jaw of the vice is not shown. B—Another technique for squaring the ends of work. The jaw must be checked with a dial indicator to be sure it is at a right angle to the column.



Goodheart-Willcox Publisher

Figure 18-10. Sequence for squaring work on a milling machine. A—Square the top. B—Square one side. C—Square the bottom. D—Square the other side.

SAFETY NOTE

Always stop the machine before attempting to make measurements.

18.2.3 Machining Angular Surfaces

Angular surfaces (bevels, chamfers, and tapers) may be milled by tilting the spindle head assembly to the required angle. They may also be made by setting the work at the specified angle in a vise. See **Figure 18-12**. A fixture, **Figure 18-13**, is often used when many similar pieces must be milled. Compound angles (angles on two planes) are made in a universal vise.

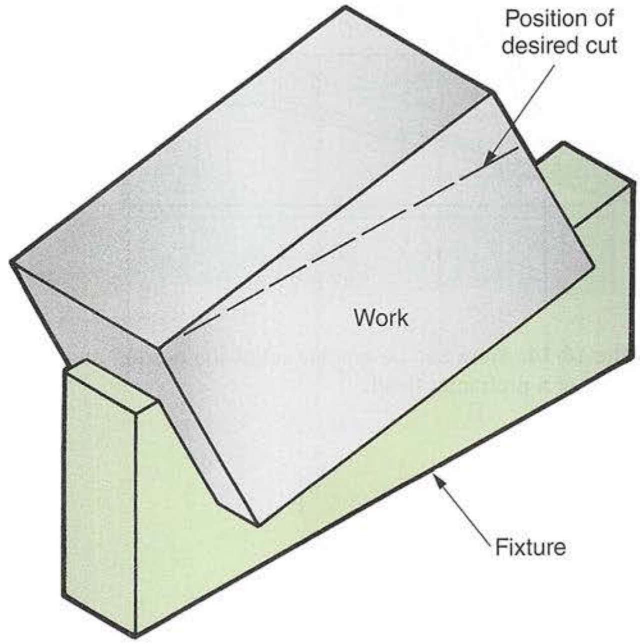
When the pivoted spindle head is used for angular cutting, it is essential to align the vise with a dial indicator. Make a layout of the desired angle on the work and clamp it in the vise. Position the cutter and machine to the line.

Work mounted at an angle in the vise for machining must be set up carefully. Alignment may be made with a protractor head fitted with a spirit level, **Figure 18-14**, or with a surface gage.

18.2.4 Milling a Keyseat or Slot

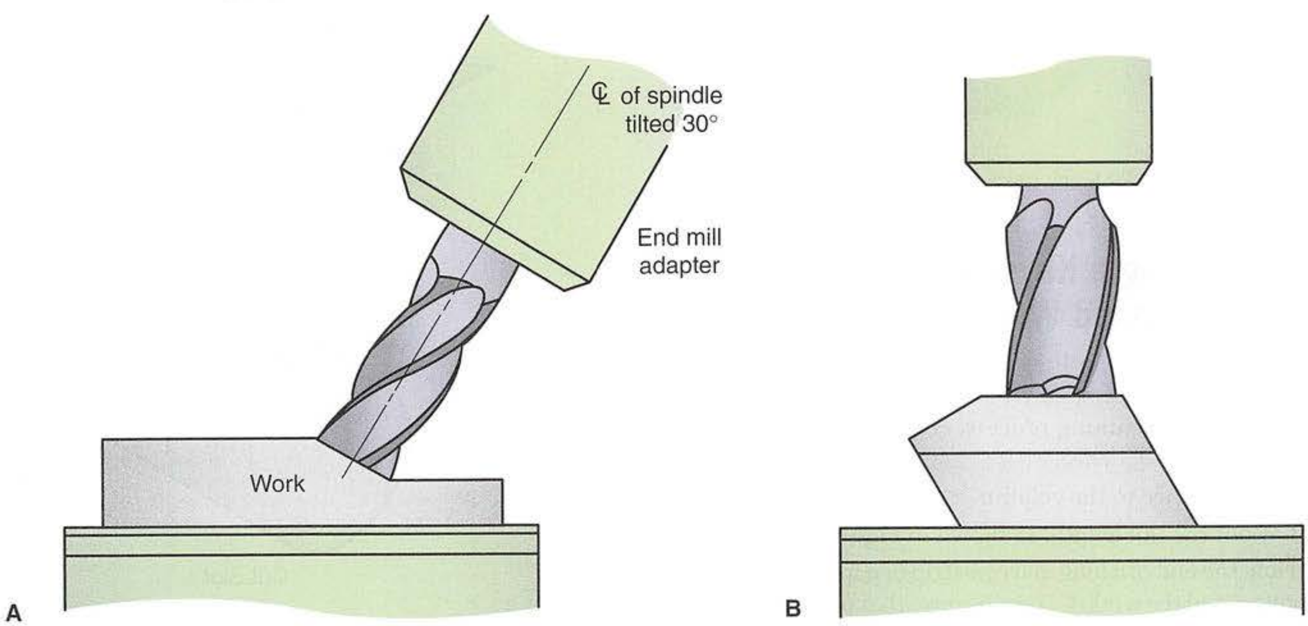
An end mill may be used to cut a keyseat or slot. After aligning the vise with a dial indicator, clamp the workpiece in the vise or to the machine table. If you mount it directly to the table, place a piece of paper between the table and the work to seat the work more solidly and prevent slippage.

Use a sharp cutter that is equal in diameter to the keyseat or slot. To machine a blind keyseat or slot, use a two-flute end mill, **Figure 18-15**. Otherwise, use a multiflute end mill.



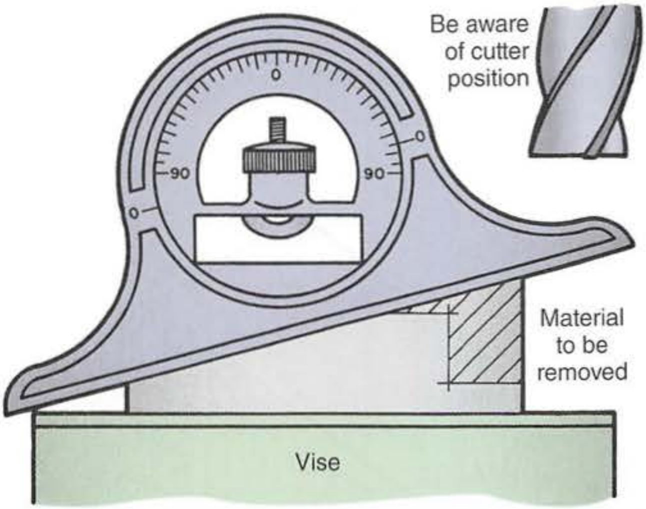
Goodheart-Willcox Publisher

Figure 18-13. If many pieces are to have angular surfaces machined, considerable time can be saved by using a fixture.



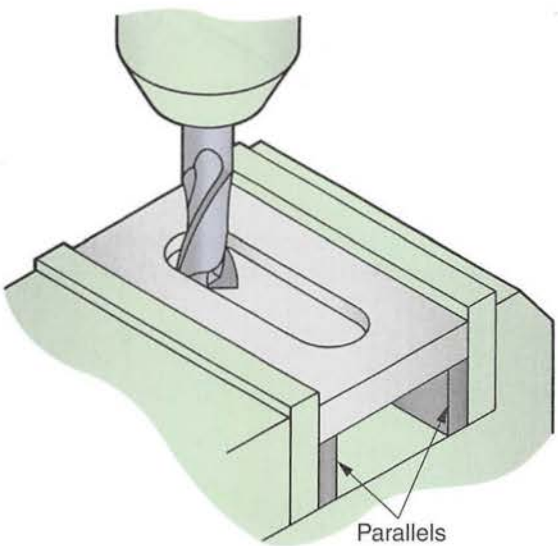
Goodheart-Willcox Publisher

Figure 18-12. Cutting an angular surface. A—Cutting an angular surface with the spindle head pivoted to the desired angle. B—Cutting an angular surface by positioning the work at the desired angle in a vise.



Goodheart-Willcox Publisher

Figure 18-14. Work can be quickly set at the desired angle with the aid of a protractor head.



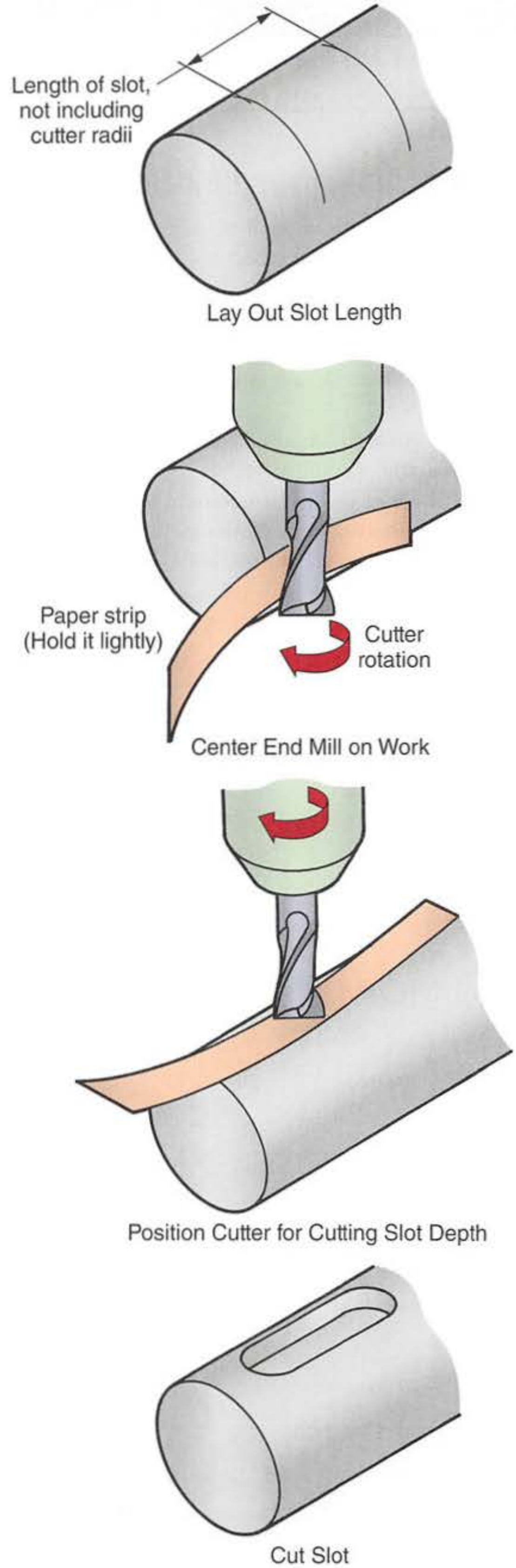
Goodheart-Willcox Publisher

Figure 18-15. Blind slot being machined with a two-flute end mill.

18.2.5 Cutting a Keyseat or Slot on Round Work

After setting up the milling machine, secure the work in a vise, between centers, in V-blocks, or in a fixture. Before beginning the machining process, center the end mill precisely using this sequence:

1. Lock the knee to the column.
2. Lay out the slot length, as shown in **Figure 18-16**.
3. Hold the end of a long, narrow strip of paper between the cutter and the work. Carefully move the cutter toward the work until the paper strip is pulled lightly from your fingers by the rotating cutter. Pay close attention when using this alignment technique. Use a paper strip long enough



Goodheart-Willcox Publisher

Figure 18-16. Procedure for using a paper strip to position the cutter on the exact center of round stock for cutting a keyseat. Keep your fingers clear of the rotating cutter!

to keep your fingers well clear of the cutter. Release the paper as soon as you feel the cutter "grabbing" it.

4. Unlock the knee from the column and lower the table until the cutter is slightly above the work. Move the cutter inward half the work diameter, plus half the cutter diameter, plus the paper thickness.
5. Using another long, narrow strip of paper, use the same technique to get the required depth.

Correct keyseat depth may be obtained from tables in a machinist's handbook.

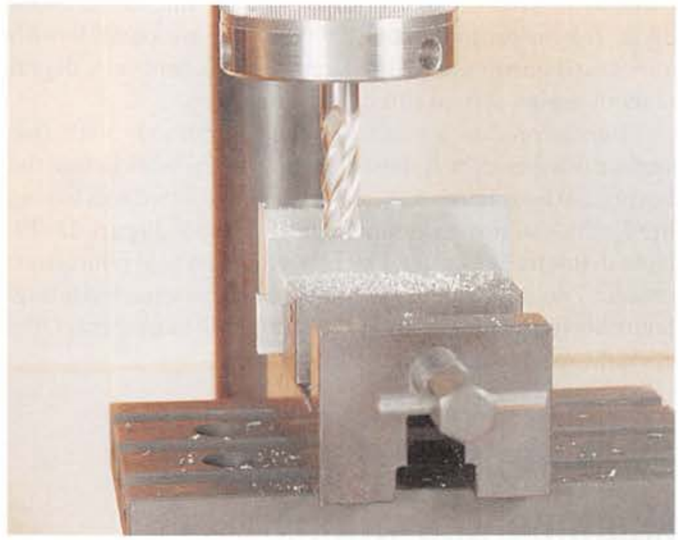
18.2.6 Machining Internal Openings

Internal openings are easily machined with a vertical milling machine. A two-flute end mill must be used if the cutter is to make the initial opening. It can be fed directly into the material in much the same manner as a drill.

When the slot is wider than the cutter diameter, the direction of feed in relation to cutter rotation becomes important. Feed direction should be against cutter rotation, **Figure 18-17**. This applies only when the cutter is removing metal from one side of the opening.

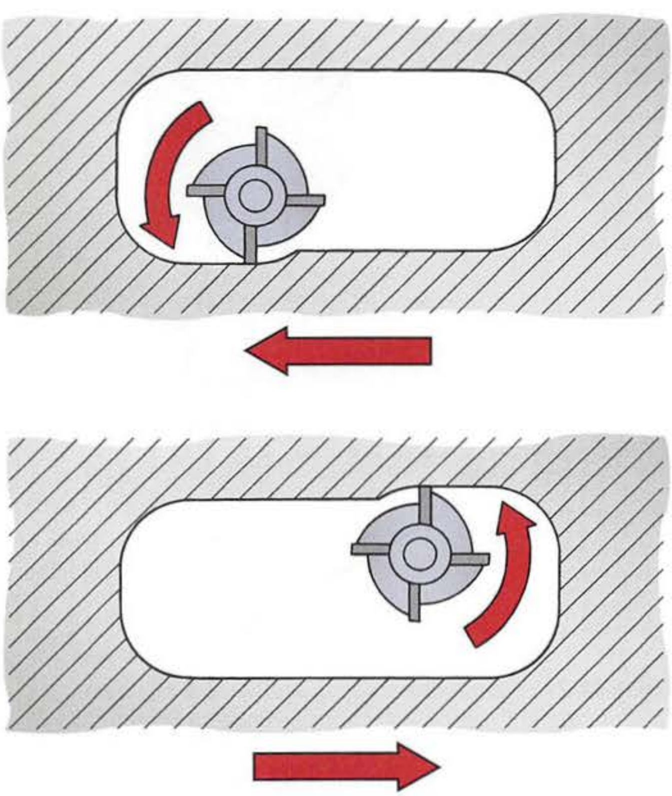
18.2.7 Machining Multilevel Surfaces

Milling a multilevel surface is probably the easiest of the milling operations, **Figure 18-18**. A layout of the various levels is made on the work's surface. Cuts are made until the lines are reached. For accuracy, the cut must be checked with a depth micrometer, **Figure 18-19**. Remove any burrs before making the measurement. Make the necessary table adjustments accordingly.



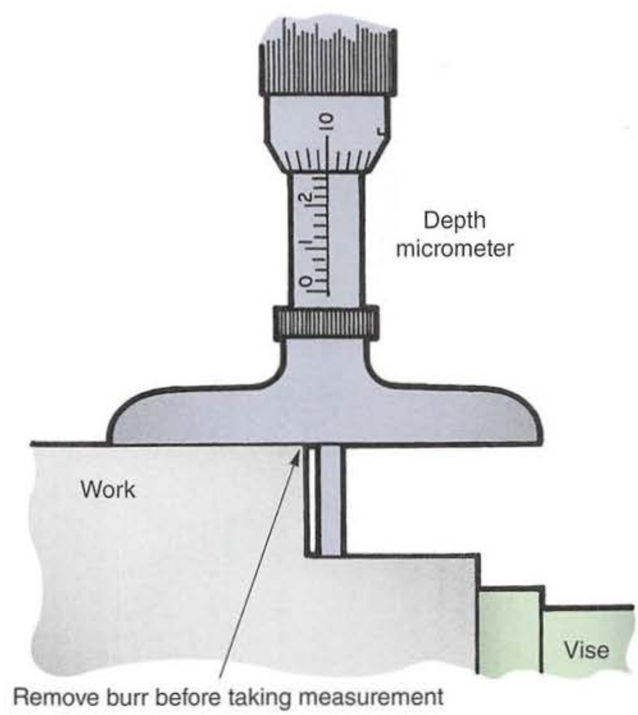
Goodheart-Willcox Publisher

Figure 18-18. A stepped (multilevel) surface being milled.



Goodheart-Willcox Publisher

Figure 18-17. When removing metal from one side of an internal opening, remember that feed direction is always against cutter rotation.



Goodheart-Willcox Publisher

Figure 18-19. Check the depth of the cut with a depth micrometer before making final cutter adjustments. Remove any burrs before making the measurement. Always stop the machine before removing the burrs and making a measurement.

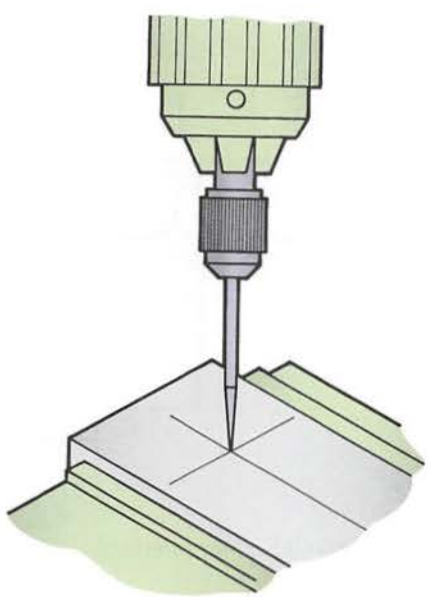
18.2.8 Drilling, Reaming, and Boring

Holes may be located to very close tolerances for drilling, reaming, or boring on a vertical milling machine. The first hole can be located with a wiggler, **Figure 18-20**, a centering scope, **Figure 18-21**, an edge finder, **Figure 18-22**, or a coaxial indicator. It is then possible to locate any remaining holes within 0.001″ (0.025 mm) using the micrometer feed dials. Tolerances of 0.0001″ (0.0025 mm) are possible with a measuring rod and dial indicator attachment or a digital readout gaging system fitted to the machine.

Boring produces machined holes accurately with fine surface finishes. A hole must be drilled in the work before the boring can be started. A single-point tool is fitted to a boring head, which in turn is mounted in the spindle, **Figure 18-23**. Hole diameter is obtained by offsetting the tool point from center. The adjustment is graduated for a direct reading. Some boring head dials indicate actual tool movement. Others indicate actual material removal.

18.3 Horizontal Milling Machine Operations

Like the vertical milling machine, the horizontal mill is a versatile machine tool. Many different machining operations can be performed on it.



Goodheart-Willcox Publisher

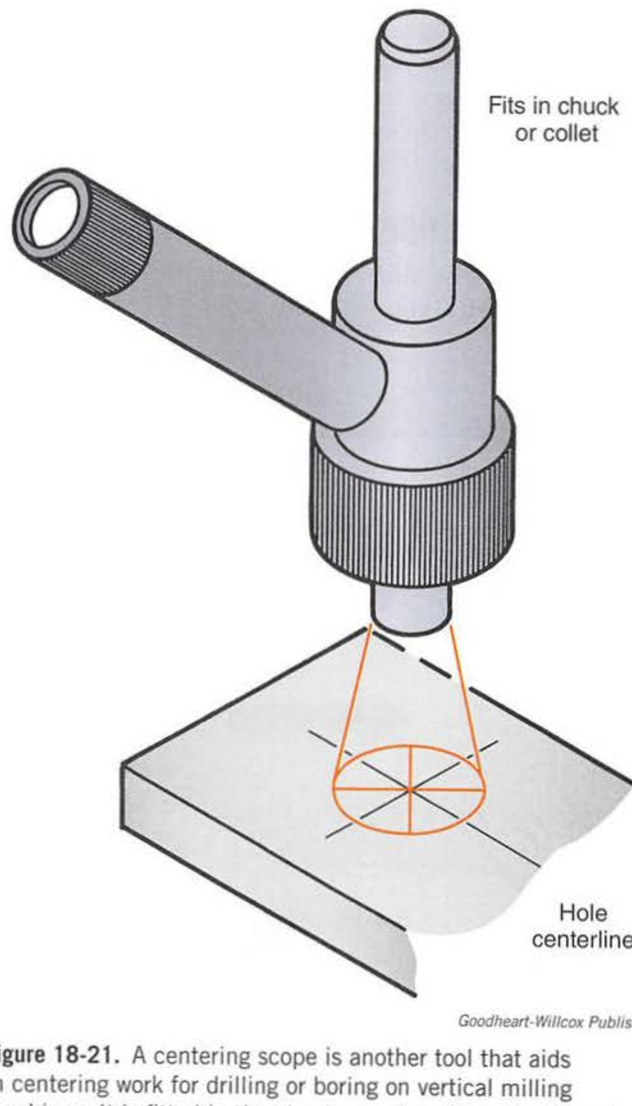
Figure 18-20. Accuracy can be ensured when locating holes on a milling machine by aligning the first hole with a "wiggler."

18.3.1 Milling Flat Surfaces

Study the part drawing carefully to determine which operation is to be performed, which cutter is best suited for the job, and the best way to hold the workpiece. Flat surfaces may be milled with a plain cutter or slab cutter mounted on an arbor (peripheral milling), or with an inserted-tooth face or shell milling cutter (face milling). The method used is determined by the size and shape of the work.

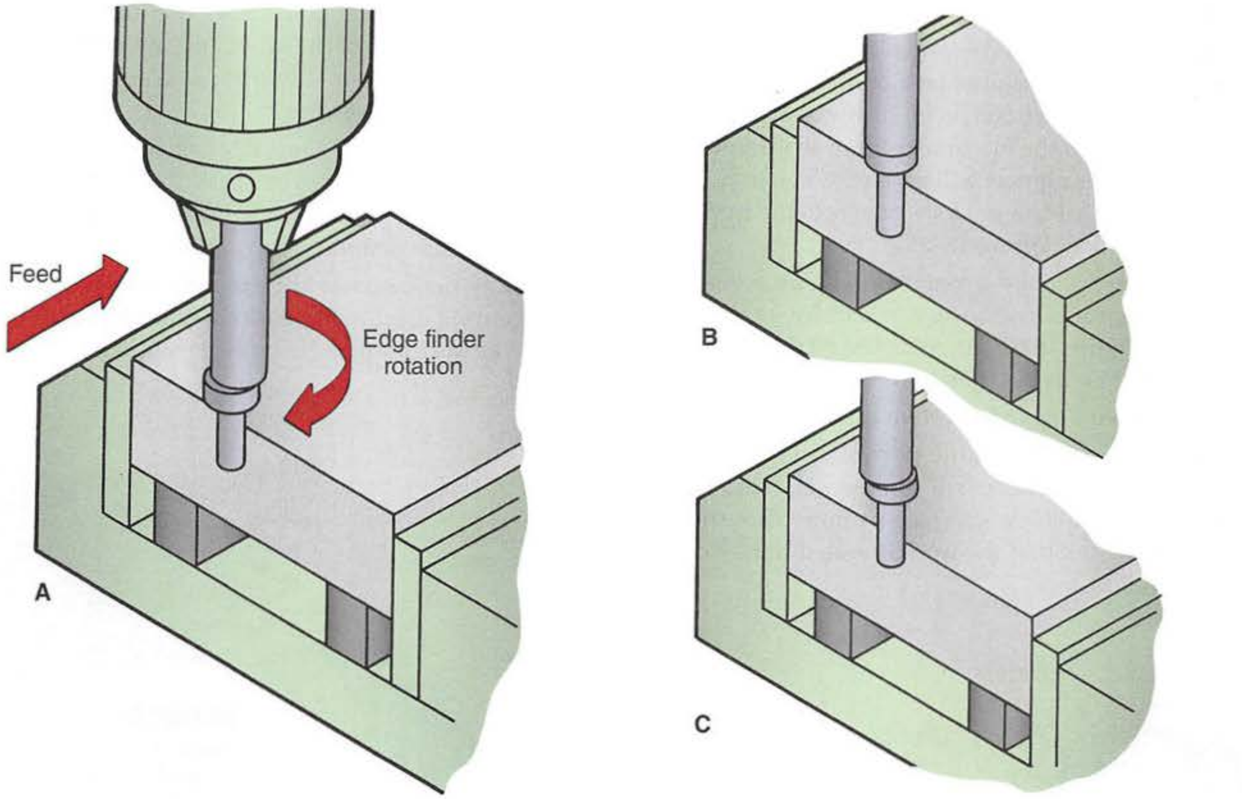
After you have selected the milling method and cutter, follow these steps:

1. Check and lubricate the machine. Wipe the worktable clean and examine it for nicks and burrs. Nicks or burrs prevent the workpiece or holding attachments from seating properly on the table.



Goodheart-Willcox Publisher

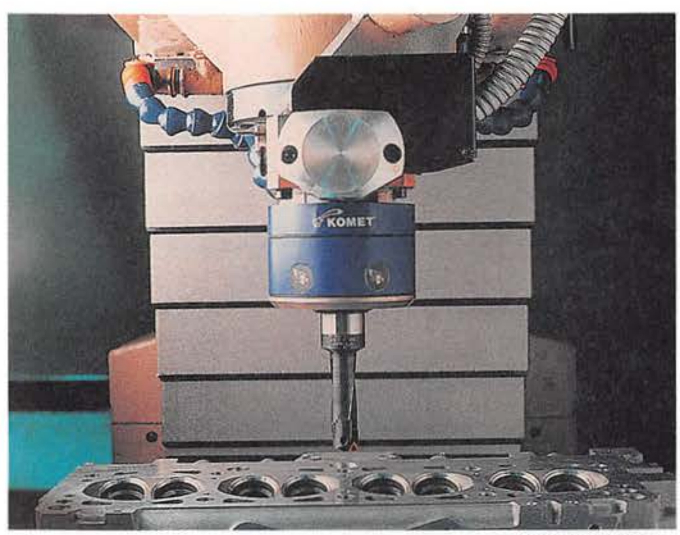
Figure 18-21. A centering scope is another tool that aids in centering work for drilling or boring on vertical milling machines. It is fitted in the chuck or collet. (The chuck must run true.) The machinist sights through the optical system and positions the work using the crosshairs in the scope. A centering scope is capable of locating true position to within ±0.0001″ (0.0025 mm).



Goodheart-Willcox Publisher

Figure 18-22. The edge finder is a precision positioning tool that locates the edge of the work in relation to the center of the spindle with 0.0002" (0.005 mm) accuracy. A—With the spindle rotating at moderate speed, and with the edge finder tip as shown, slowly feed the tip of the tool against the work. B—The edge finder tip gradually becomes centered with its shank. C—When the tip becomes exactly centered, it abruptly jumps sideways about 1/32" (0.8 mm). When this occurs, stop the table movement immediately. The center of the spindle will be exactly one-half tip diameter away from edge of work. Set the micrometer dial to 0 and, with the edge finder clear of the work, move the table longitudinally the required distance plus one-half the tip diameter. Follow the same procedure to get the traverse measurement.

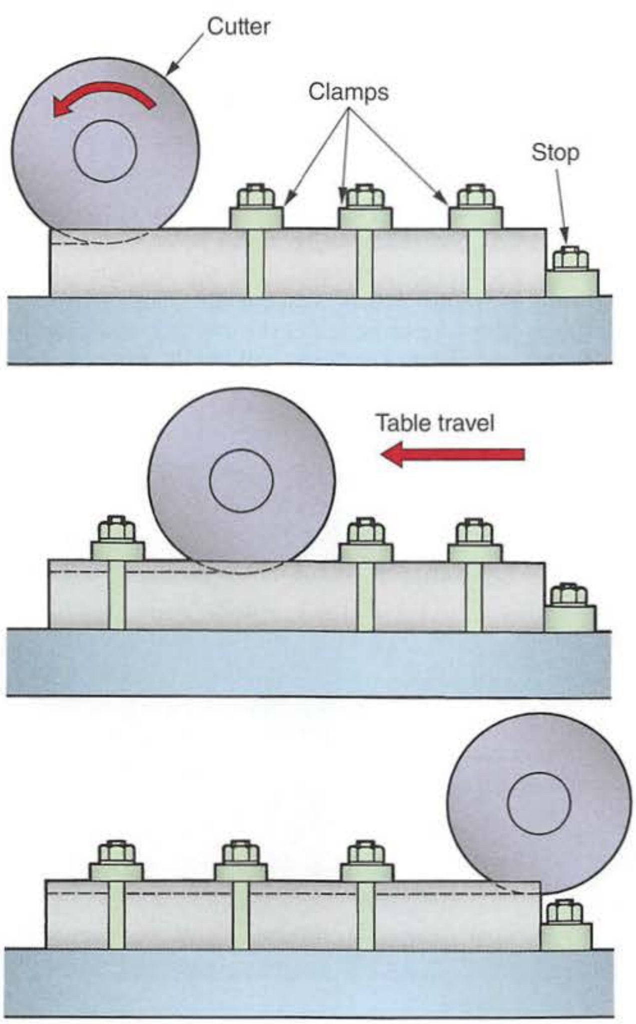
A Marposs Corp.



B Komet of America, Inc.

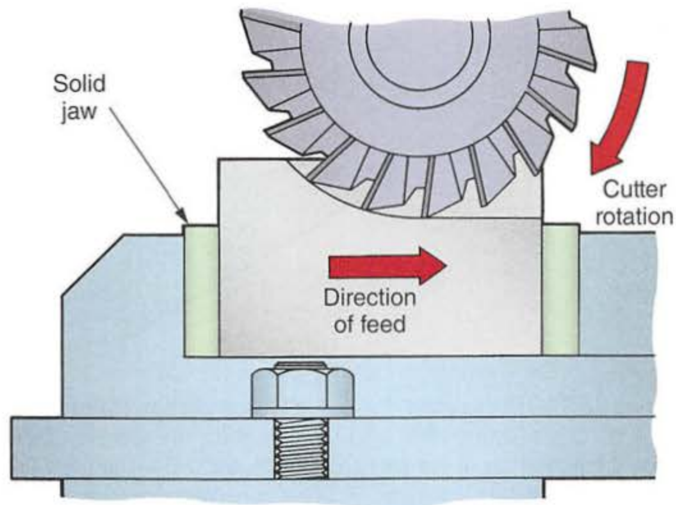
Figure 18-23. Boring heads. A—Micro-adjustable boring head with digital display. This unit has an accuracy of 0.00004" (0.001 mm), and has a decimal or metric readout display. B—Boring head with a programmable electronic controller interfaced to the machine control. It relates measurement data to preset tolerance limits and signals the tool to make size adjustments to compensate for tool wear. The boring head incorporates a small servomotor that adjusts the boring slide within extremely fine limits. Communication to and powering of the tool are entirely wireless.

2. Mount the work directly on the table, if possible. **Figure 18-24** illustrates one method of mounting long work. If the work cannot be mounted to the table, use a vise. Clean its base and bolt it firmly in place. Locate the vise as close to the machine column as workpiece shape and the arbor support will permit. When possible, pivot the vise so that the solid jaw supports the work against cutter rotation, **Figure 18-25**.
3. If extreme accuracy is required, align the vise using a dial indicator. Otherwise, a square or machine arbor will do, **Figure 18-26**. Angular vise settings can be made using the vise base graduations or a protractor.
4. Wipe the vise jaws and bottom clean.
5. Place clean parallels in the vise with the work seated on them. Tighten the jaws and tap the work onto the parallels with a mallet or soft-face hammer. Use thin paper strips to check that the work is seated tightly on top of the parallels.
6. Select as short an arbor as the job permits. Wipe the taper section of the arbor and the spindle opening with a dry cloth. Insert the arbor and draw it tightly into place with the draw-in bar.
7. When possible, the cutter should be wide enough to machine the area in one pass. It should have the smallest diameter possible, while being large enough to provide adequate clearance. See **Figure 18-27**.
8. Key the cutter to the arbor to prevent it from slipping during machining. Position it as close to the column as the work permits. To protect your hands when mounting



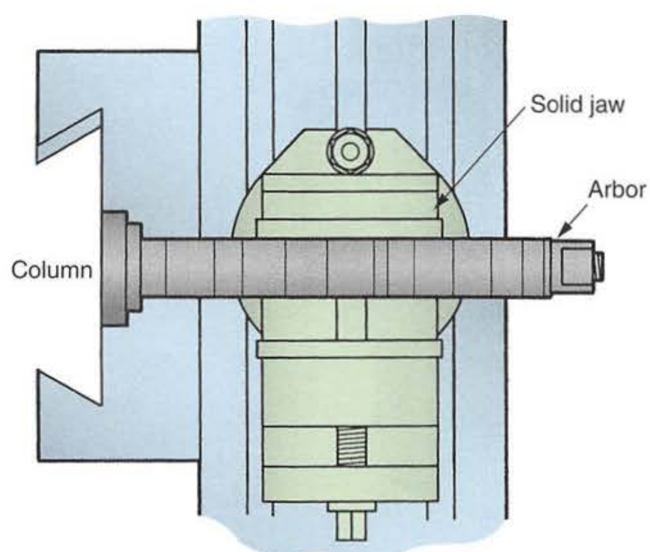
Goodheart-Willcox Publisher

Figure 18-24. A method used to mount long work. Reposition the clamps as the cut progresses across the workpiece.



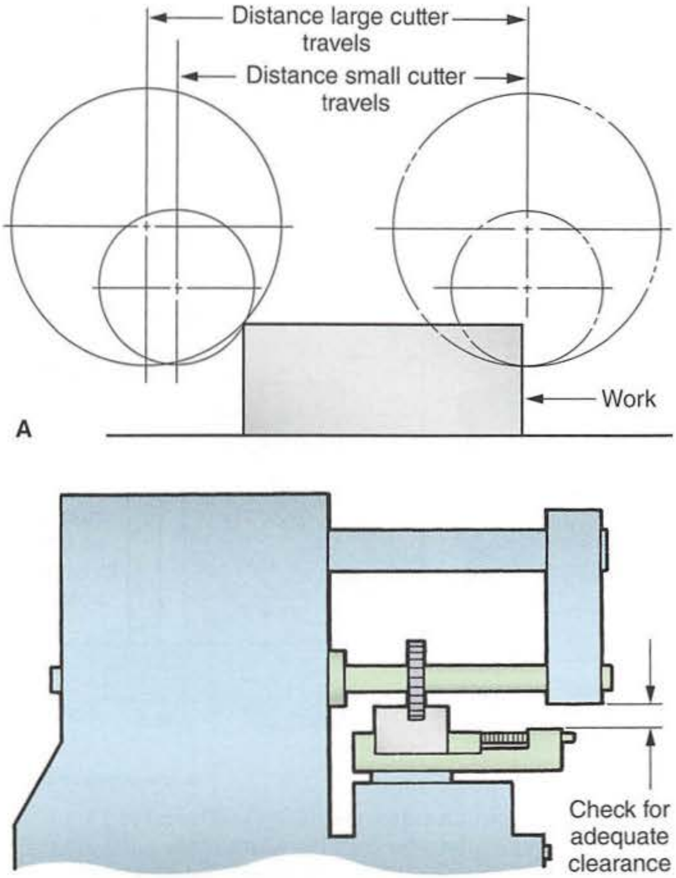
Goodheart-Willcox Publisher

Figure 18-25. Whenever the setup permits, the solid jaw of the vise should support the work against the cutter rotation.



Goodheart-Willcox Publisher

Figure 18-26. An arbor can be used to align a vise unless extreme accuracy is required.



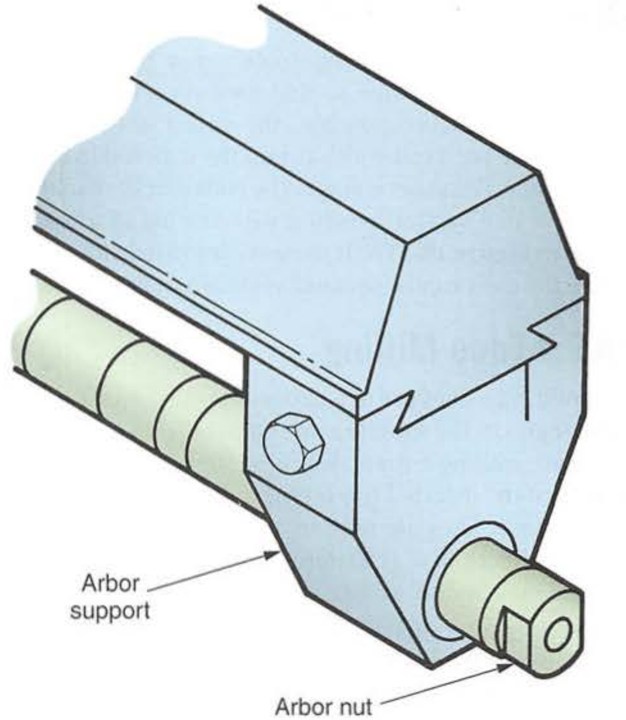
Goodheart-Willcox Publisher

Figure 18-27. Cutter diameter. A—A small-diameter cutter is more efficient than a large-diameter cutter because it travels less distance while doing the same amount of work. B—Use the smallest cutter diameter possible, but be certain it is large enough to allow adequate clearance.

a cutter on the arbor, use a piece of cloth or gloves. If you are using a helical slab mill, mount it so that the cutting pressure forces it toward the column.

9. Position and lock the arbor support into place, **Figure 18-28**, and then tighten the arbor nut.
10. Adjust the machine to the proper cutting speed and feed.
11. Turn on the machine and check cutter rotation and direction of power feed. If satisfactory, loosen all work-table and knee locks. Position the workpiece under the rotating cutter until the cutter just touches its surface. Set the micrometer dial to 0. Back the work away from the rotating cutter. Make a light cut with ample cutting fluid flooding the surface. Make a measurement, then raise the table the required distance.

If a previously machined surface requires additional machining, position the cutter by holding a long, narrow strip of paper with its loose end between the work and the cutter. Raise the table until the paper is pulled lightly from your fingers.



Goodheart-Willcox Publisher

Figure 18-28. Do not tighten the arbor nut until after the arbor support has been positioned and locked in place.

SAFETY NOTE

Pay close attention when positioning a cutter by the paper strip method. Use a long strip of paper, hold it lightly, and keep your fingers well clear of the rotating cutter.

Once the cutter has been positioned, it is only necessary to move the cutter clear of the work and raise the table the required distance, plus the thickness of the paper. Tighten all locks (except longitudinal) and feed the work into the cutter. As soon as cutting starts, turn on the coolant and power feed.

Unless there is an emergency, do not stop the work during the machining operation. Stopping causes a slight depression to be cut in the machined surface. Do not bring the rotating cutter back over the machined surface. Doing so creates ridges on the surface of the work.

Complete the cut and stop the cutter. Return the work to the starting position. Avoid feeding the work back to the starting position while the cutter is rotating because doing so will make a series of depressions on the newly machined surface.

Repeat the above operations if additional metal must be removed to bring the work to size.

SAFETY NOTE

Do not attempt to feel the machined surface while the cut is in progress or while the cutter is rotating.

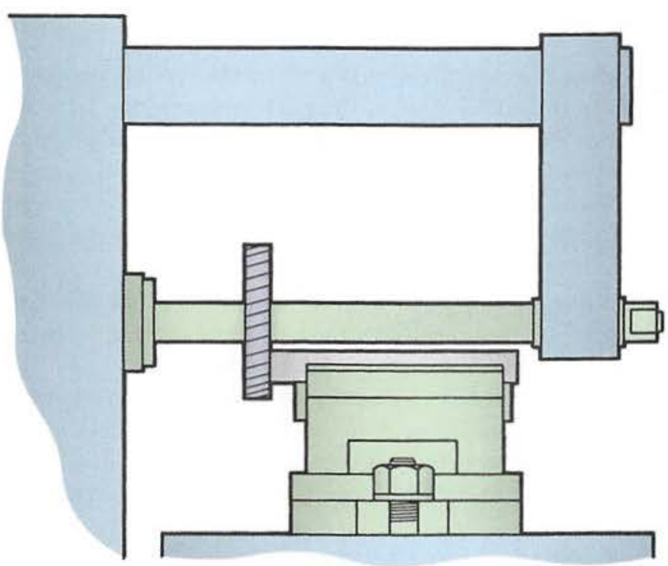
18.3.2 Squaring Stock

The sequence for squaring stock on a horizontal milling machine is the same as that used on a vertical milling machine. Whenever possible, the cutter should be wide enough to make a full-width cut on the material in one pass. If the material is short enough, the ends can be machined by placing it in a vertical position with the aid of a square, as shown in **Figure 18-11A**. If the work is too long for this technique, the ends can be squared as shown in **Figure 18-29**.

18.3.3 Face Milling

Face milling makes use of a cutter that machines a surface at right angles to the spindle axis and parallel to the face of the tool. Face milling cutters over 6″ (150 mm) in diameter are usually of the inserted tooth type and mount directly to the spindle nose. They are used to mill large, flat surfaces. Face mills smaller than 6″ (150 mm) are called *shell end mills* and are held on a Style C arbor.

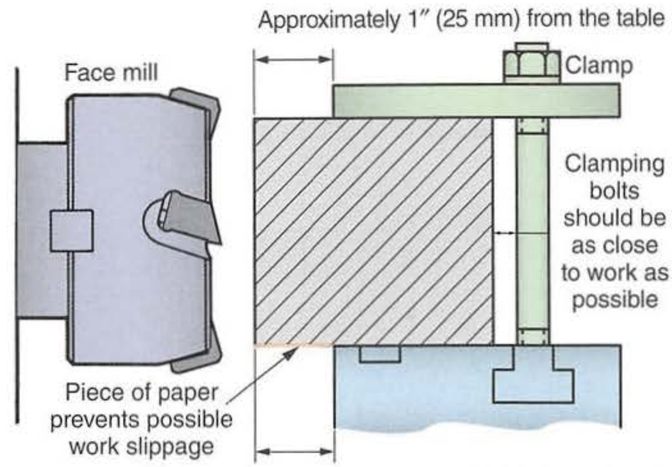
1. Select a cutter that is 3/4″ (20 mm) to 1″ (25 mm) larger in diameter than the width of the surface to be machined, **Figure 18-30**.
2. The work should project about 1″ (25 mm) beyond the edge of the table to provide adequate clearance. In face milling, it is frequently necessary to mount the work on an angle plate, **Figure 18-31**.
3. Adjust the machine for correct speed and feed.
4. Slowly feed the work into the cutter until it starts to remove material. Roughing cuts up to 1/4″ (6 mm) may be taken. Use adequate cutting fluid.



Goodheart-Willcox Publisher

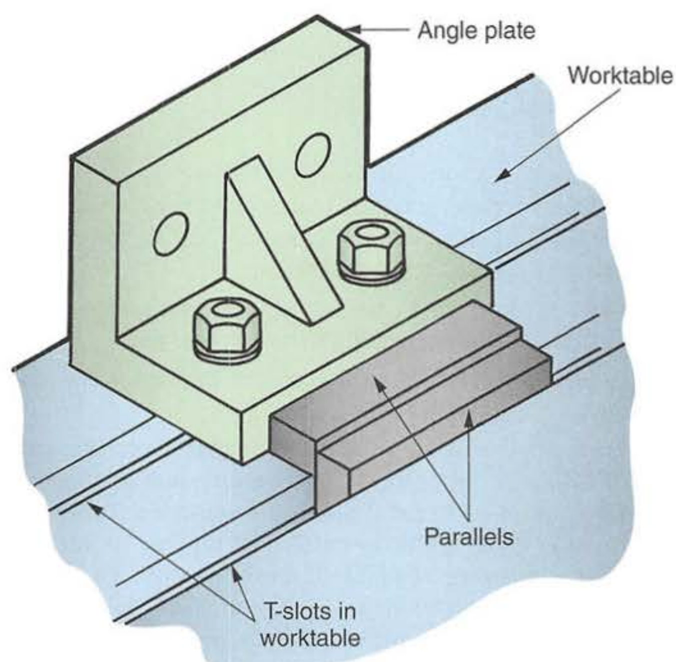
Figure 18-29. Another technique for squaring work ends. Check the solid vise jaw with a dial indicator to ensure that it is properly aligned. Be sure there is adequate clearance between the work and the arbor.

5. When the cut is complete, stop the cutter. Return the work to the starting position for additional machining, if needed.
6. Make the finishing cut and tear down the setup. Use a brush to remove chips. Clean and store the cutter.



Goodheart-Willcox Publisher

Figure 18-30. Select a cutter that is 3/4″ (20 mm) to 1″ (25 mm) larger in diameter than the width of the surface to be machined. The work should project approximately 1″ (25 mm) beyond the table edge to provide adequate clearance.



Goodheart-Willcox Publisher

Figure 18-31. An angle plate is often used to mount work for face milling. Check that the mounting clamps clear the cutter. Note the use of parallels to align the angle plate.

18.3.4 Side Milling

Side milling refers to any milling operation that involves the use of half side and side milling cutters. When cutters are used in pairs to machine opposite sides of a piece at the same time, the setup is called ***straddle milling***, **Figure 18-32**.

Cutters used for this operation should be kept in matched pairs. That is, they should be sharpened at the same time to maintain equal diameters. Shoulder width of the machined surface is determined by the thicknesses of the spacers between the cutters, **Figure 18-33**.

Spacers are available in a large selection of standard sizes. Additional sizes can be made by surface-grinding standard sizes to needed dimensions. Shim stock spacers can be used to build up standard spacers to a desired dimension.

Gang milling involves mounting several cutters on an arbor to machine several surfaces in a single pass, **Figure 18-34**. It is a variation of straddle milling. Gang milling is used when many identical pieces must be made.

Side milling cutters can also be utilized to machine grooves and keyseats. When used with a dividing head or rotary table, they can cut squares, hexagons, or other shapes on round stock.

Milling a Slot in Square or Rectangular Work

When milling a slot, the machine is set up in much the same manner as it is for milling flat surfaces. Use a dial indicator to check the vise to ensure accuracy. Exercise the same care in placing the side cutter on the arbor when using a slab cutter. You may use a plain side milling cutter if the slot is not too deep. Otherwise, use a staggered-tooth side milling cutter.

Make a layout on the end of the work. Then position the cutter using one of the following methods:

- Use a steel rule, **Figure 18-35**. Make a light cut partway up the piece and remove the burrs. Measure the cut depth with a depth micrometer. The difference

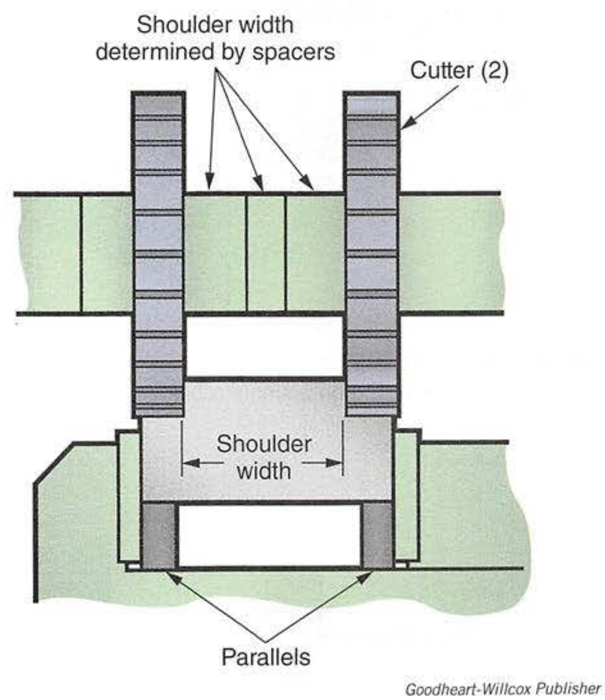


Figure 18-33. Arbor spacers are used to set the distance between cutters.

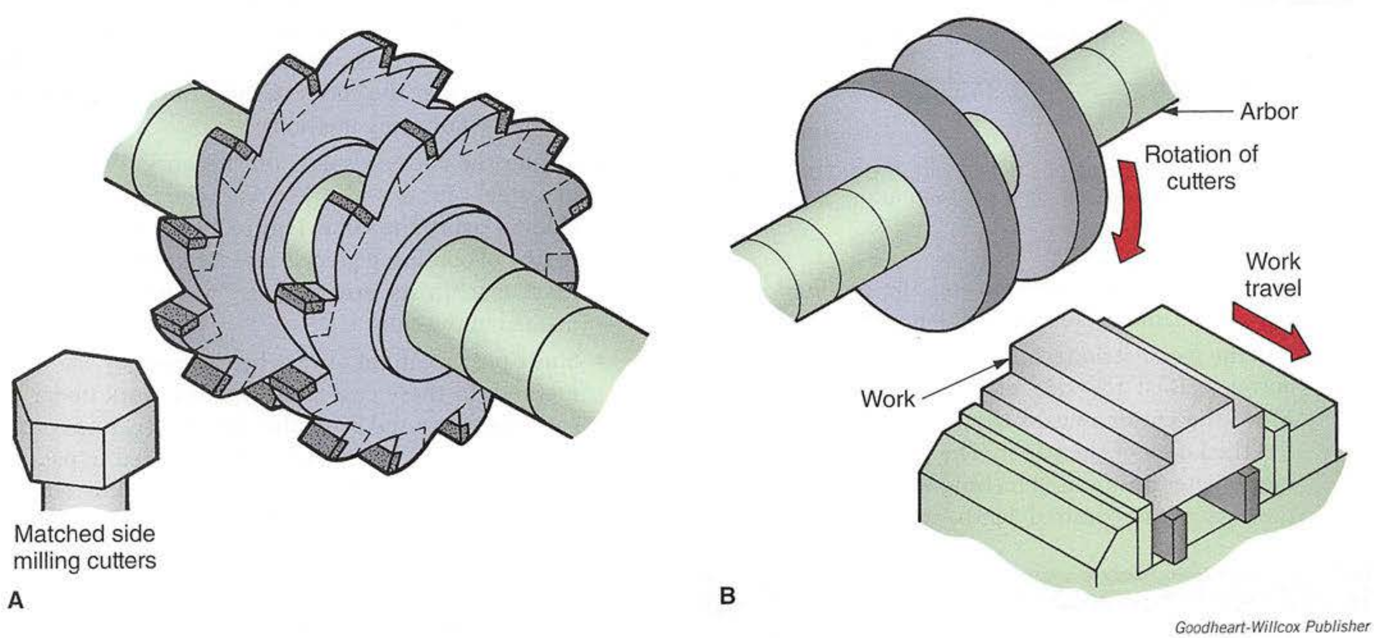


Figure 18-32. Straddle milling. A—This machining method uses a spacer between cutters. These two cutters are machining a hexagon on a machine part. B—An example of straddle milling on flatwork.

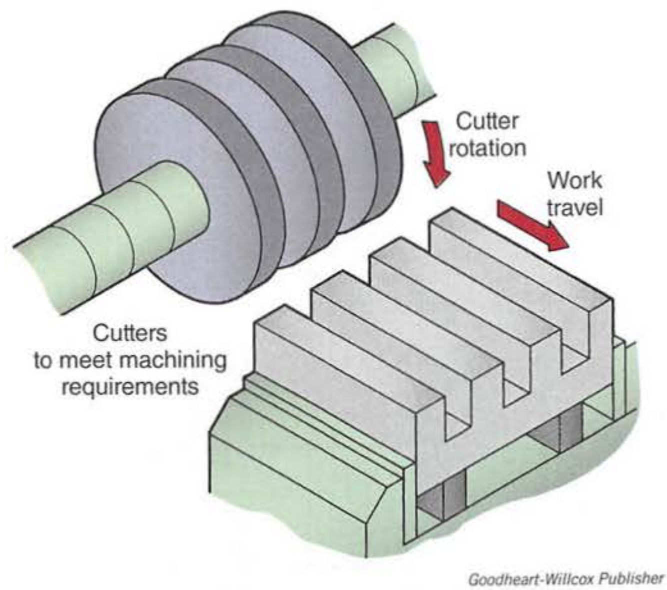


Figure 18-34. Gang milling involves the use of two or more milling cutters mounted on a single arbor.

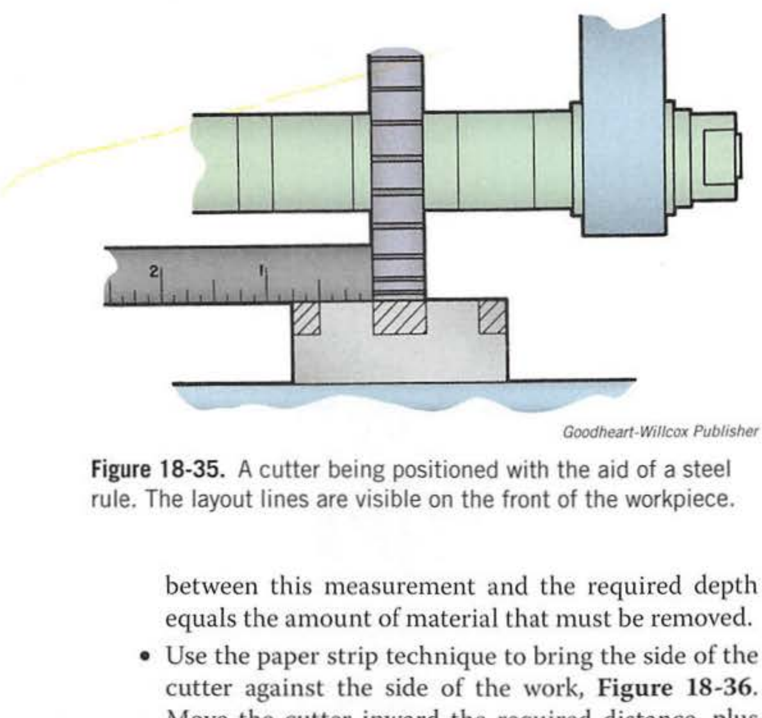


Figure 18-35. A cutter being positioned with the aid of a steel rule. The layout lines are visible on the front of the workpiece.

between this measurement and the required depth equals the amount of material that must be removed.

- Use the paper strip technique to bring the side of the cutter against the side of the work, **Figure 18-36.** Move the cutter inward the required distance, plus the thickness of the paper. The paper strip or depth micrometer positioning technique may be used to set the cutter to the desired depth.

SAFETY NOTE

When using the paper strip technique to position a cutter, remember to use a long paper strip and keep your fingers clear of the revolving cutter.

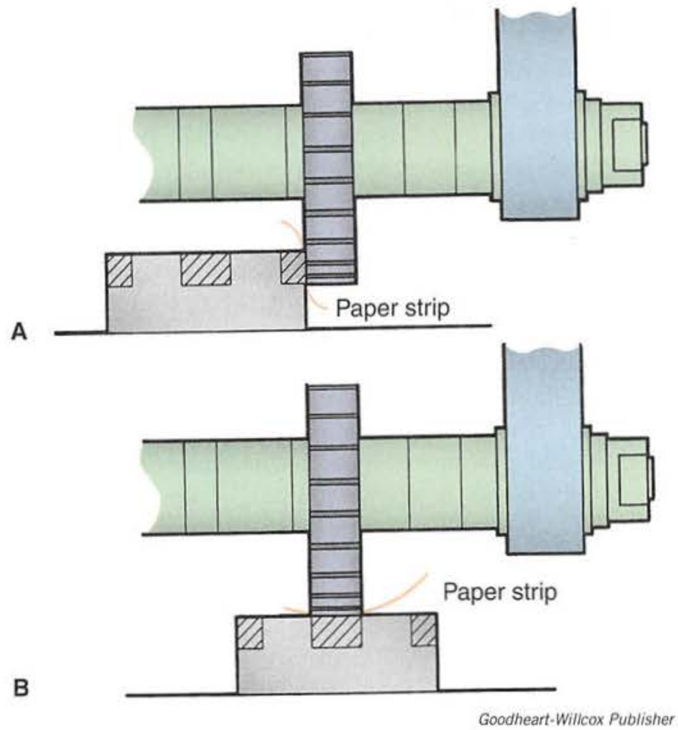


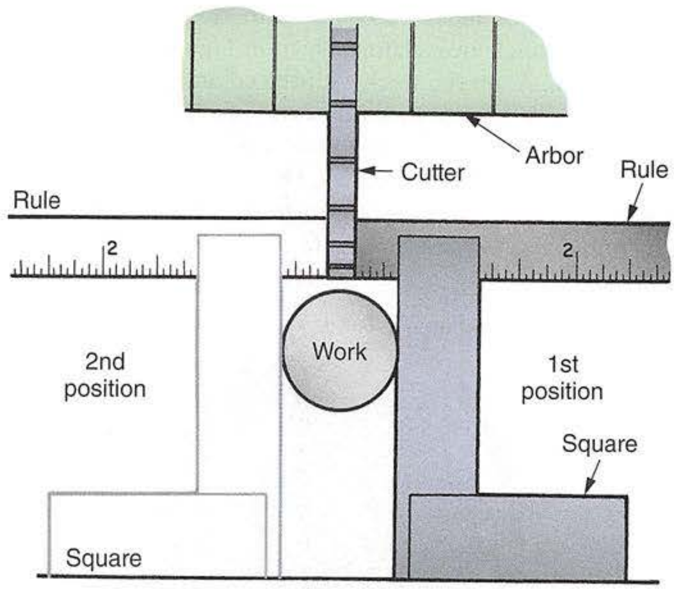
Figure 18-36. Paper strip method of positioning the cutter. A—Using a paper strip to secure the internal dimension. Read the micrometer to move the cutter the correct distance over the work. B—Using a paper strip to position the cutter for depth. Read the micrometer dial as the cutter is lowered.

Milling a Slot or Keyseat in Round Stock

Many situations require cutting keyseats for the standard square key in round stock. The keyseat must be kept precisely on center in order to align with the keyway in the mating piece.

After the milling machine has been set up and the work positioned in a vise, between centers, in V-blocks, or in a fixture, you must center the cutter. Center the cutter precisely using one of the following methods:

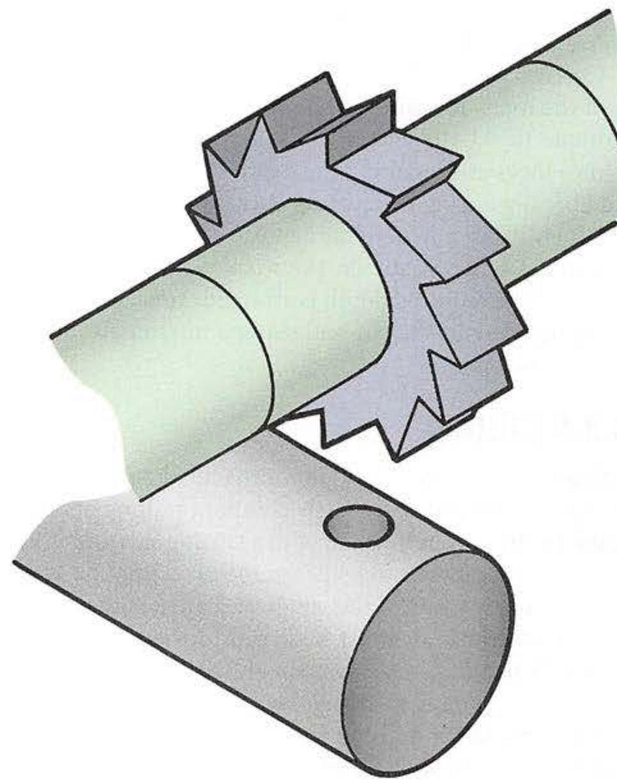
- Center the cutter visually on the work. With the aid of a steel square and rule, adjust the table until both sides measure the same, **Figure 18-37.** Due to the difficulty of obtaining precise measurements with a rule, most machinists prefer to use a depth micrometer instead of the rule.
- Short pieces cannot always be centered by the above method. In these cases, position the work under the rotating cutter and bring the work lightly into contact with the cutter. Use traverse (in/out) feed to pass the work under the cutter. Because the work is circular in shape, an oval cut will result, and the oval will be perfectly centered. To center the cutter, position it on the oval, **Figure 18-38.**
- Use the narrow paper strip technique to center the cutter. Hold the strip between the work and the cutter. Carefully move the work toward the cutter until it causes the paper to be lightly pulled from between



Goodheart-Willcox Publisher

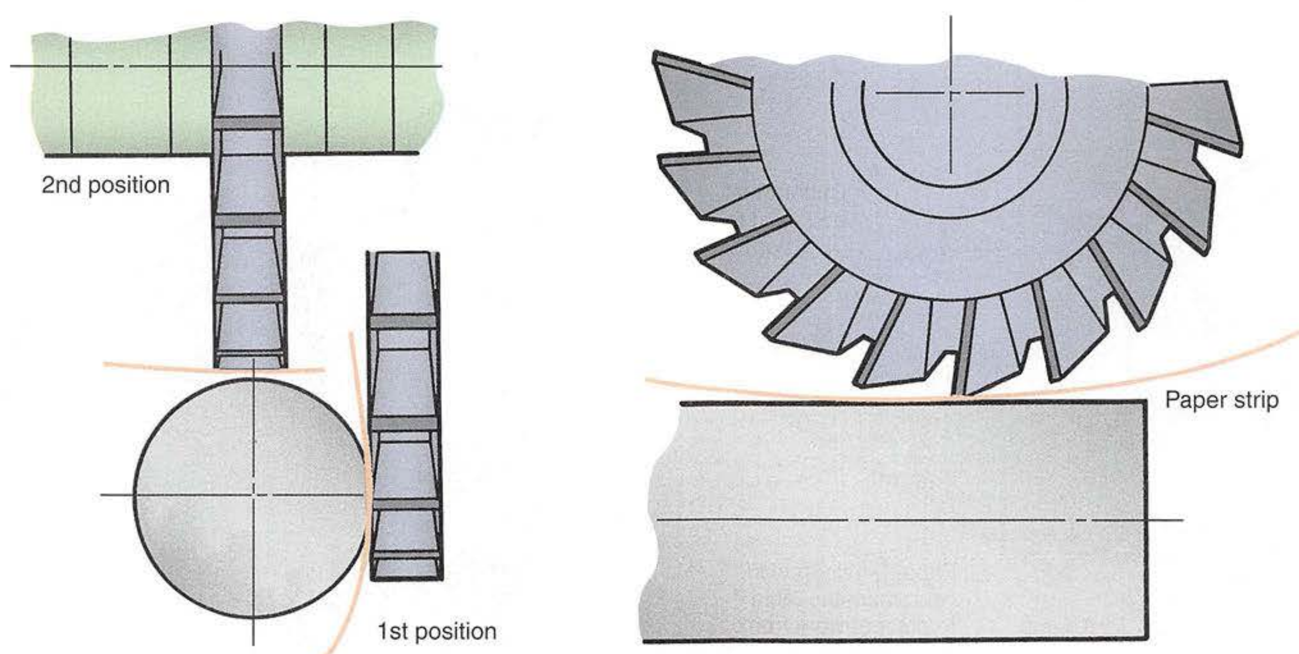
Figure 18-37. Centering a cutter on round stock with a steel rule and a machinist's square.

your fingers. Lower the table until the cutter is slightly above the work. Move the cutter inward half the diameter of the work, plus half the cutter thickness, plus the paper thickness, **Figure 18-39**. The same technique may be used to center a Woodruff keyseat cutter.



Goodheart-Willcox Publisher

Figure 18-38. Positioning a cutter on center using an oval made in the work with the cutter as a guide.



Goodheart-Willcox Publisher

Figure 18-39. Using the paper strip technique to position round stock. Use a long strip of paper. Hold the paper lightly between your fingers and keep your fingers well clear of the revolving cutter.

After you have centered the cutter, lock the saddle to prevent traverse table movement.

Correct keyseat depth can be obtained from tables in one of the many machinist's handbooks. Use the paper strip technique to set the cutter to the required depth. Tighten the knee locks after making the depth setting. Apply cutting fluid liberally during the cutting operation.

When using a Woodruff keyseat cutter, you must also position it longitudinally on the work. Slowly feed into the piece until the required depth is attained. You can check this by placing a key in the cut and using a micrometer to measure the section.

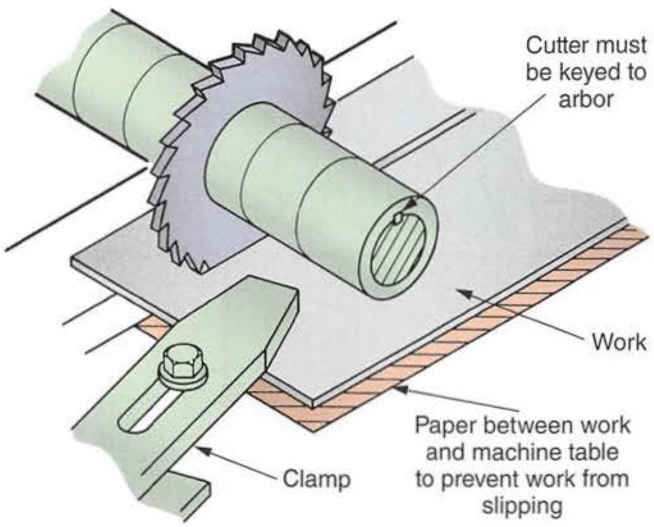
18.3.5 Slitting

Slitting thin stock into various widths for the production of flat gages or templates is a fairly common milling operation, **Figure 18-40**. It is performed with a slitting saw and is likely to give considerable trouble if extreme care is not exercised.

Use a slitting saw of the smallest diameter that permits adequate clearance. It must be keyed to the arbor, and the key should also fit into spacers on either side of the saw. For best results, mount the slitting saw for climb milling (so that the work and cutter move in the same direction at the point of contact), **Figure 18-41**. Cutting pressure is downward and tends to press the work onto the table or holding device.

Adjust the table gibs until you feel a heavy drag when you move the table by hand. This removes table "play" and prevents the saw from jumping in the cut.

If the section is narrow enough, the piece may be clamped in a vise, **Figure 18-42**. It should be well supported on parallels. Do not permit the parallels to project out into the cutter path.

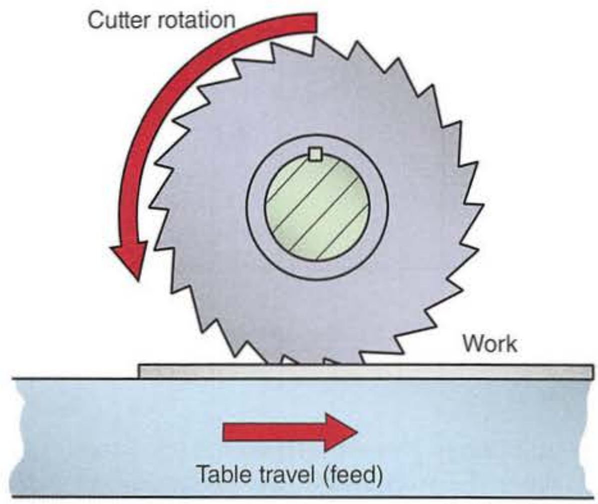


Goodheart-Willcox Publisher

Figure 18-40. Typical slitting operation setup. The work must be positioned over a table slot and clamped securely. Be sure the clamping bolts clear the arbor.

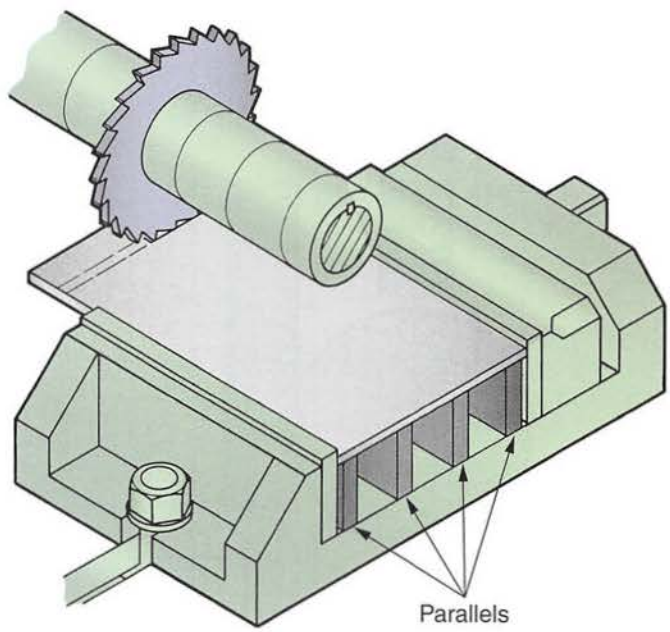
Long strips must be clamped to the worktable. The shop-made angle iron clamp shown in **Figure 18-43** is recommended. Align the work with the column face and position it to make the cut over the center of a table T-slot.

A sheet of paper between the work and table prevents the metal from slipping during the slitting operation. Set the cutter to a depth equal to the work thickness plus 1/16" (1.5 mm). Always use a sharp saw.



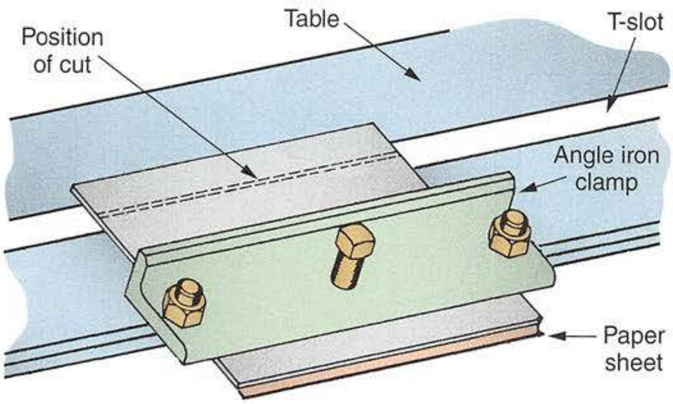
Goodheart-Willcox Publisher

Figure 18-41. Cutter rotation and feed direction for best slitting results. Use a slow feed.



Goodheart-Willcox Publisher

Figure 18-42. Narrow slitting work can be held in a vise. Be sure the parallels do not project into the cutter path.



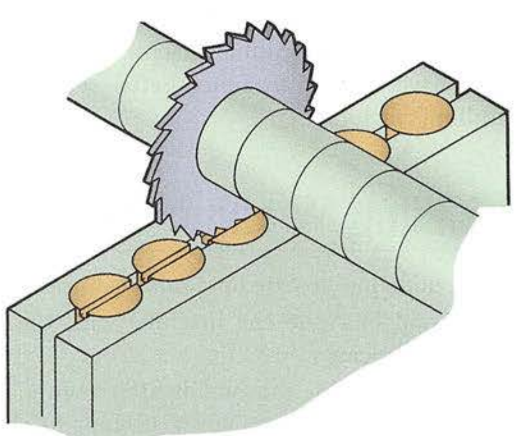
Goodheart-Willcox Publisher

Figure 18-43. A worktable clamp made from angle iron. The paper sheet prevents work movement. Position the work so that the cut is made over a T-slot.

18.3.6 Slotting

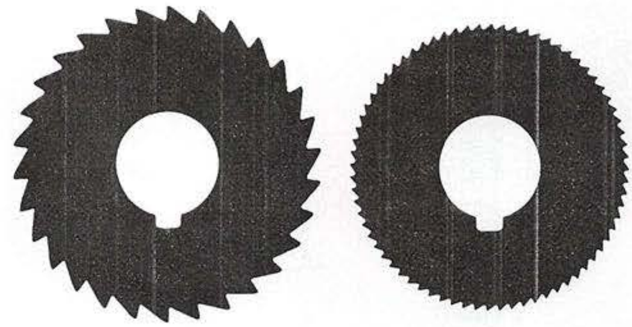
Slotting is similar to slitting, except that the cut is made only partway through the work, **Figure 18-44**, and is performed with a slotting saw. The slot in a screw head is an example of slotting.

1. Mount the slotting saw as for conventional milling. Use a sharp saw with a width suitable for the job. Note the difference between a slotting saw and a slitting saw, **Figure 18-45**.
2. Set the machine for the correct cutting speed. Use the slowest feed possible, increasing feed rate if conditions warrant.
3. Align the vise and mount the work.
4. Position the saw and make a light cut. Check the trial cut and make adjustments if necessary. Stop the machine before making measurements or adjustments.
5. Adjust the work for proper cut depth.



Goodheart-Willcox Publisher

Figure 18-44. Screw heads being slotted with a slitting saw. Slotting saws have many more teeth than slitting saws of a similar diameter.



Goodheart-Willcox Publisher

Figure 18-45. The number and size of the teeth in a slitting saw and slotting saw differ.

6. Apply cutting fluid and make the cut.

SAFETY NOTE

Avoid standing directly in line with the saw. Despite all precautions, saws shatter occasionally and can cause serious injury.

18.3.7 Drilling and Boring

The machinist often finds it necessary to produce accurately spaced and drilled holes in work. The milling machine offers a convenient way to make these holes in a specified and precise alignment. Small drills are held in a standard Jacobs chuck mounted in the machine spindle or in collet chucks, **Figure 18-46**. Larger taper shank drills are fitted into an adapter sleeve.

Boring permits large holes to be machined to close tolerances. It is done with a single-point cutting tool fitted in a boring head. Boring heads with a taper shank are mounted directly in the spindle, **Figure 18-47**. Those with a straight shank are held in a collet or adapter.

A wiggler, **Figure 18-48**, helps align the machine for drilling the hole prior to boring, or when holes are small enough to be drilled and reamed. A dial indicator must be used to realign previously made holes for boring to final size, **Figure 18-49**.

18.3.8 Advantages of Horizontal Milling

Much of the work that can be done on a horizontal mill can also be done using a vertical mill. However, the horizontal mill is often a better choice when large amounts of material need to be removed from large pieces of stock. The horizontal mill may lack the precision of some vertical machines, but it has the power and rigidity to remove larger amounts of material in the same amount of time. In addition, various contour and profile cutters can be stacked on the spindle of a horizontal mill, allowing the mill to machine profile designs on flat surfaces in less time than can be done with a vertical


Parlec, Inc.

Figure 18-46. Drills can also be mounted in collets. Collet holders are available in a range of sizes.

Parlec, Inc.

Figure 18-47. Boring tool holder and boring heads that mount directly into the spindle.

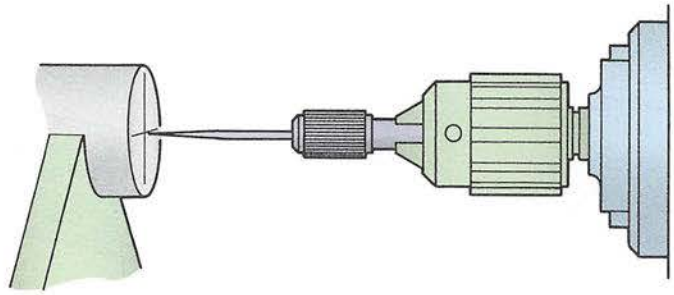
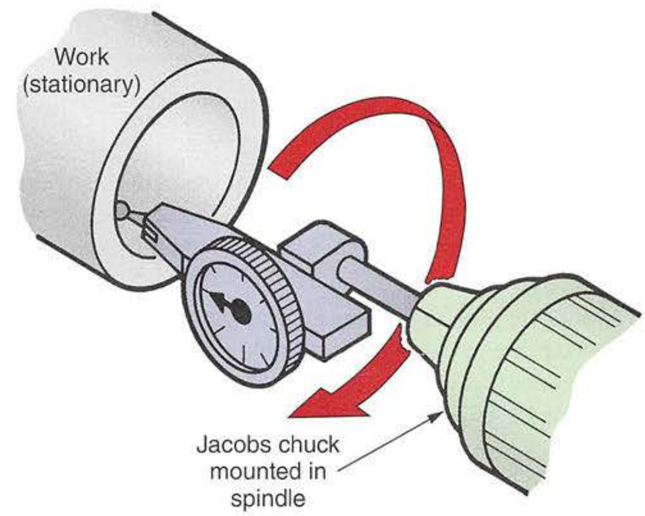

Goodheart-Willcox Publisher

Figure 18-48. Holes can be located with a wiggler.



Goodheart-Willcox Publisher

Figure 18-49. Existing holes can be realigned on a horizontal milling machine with the aid of a dial indicator.

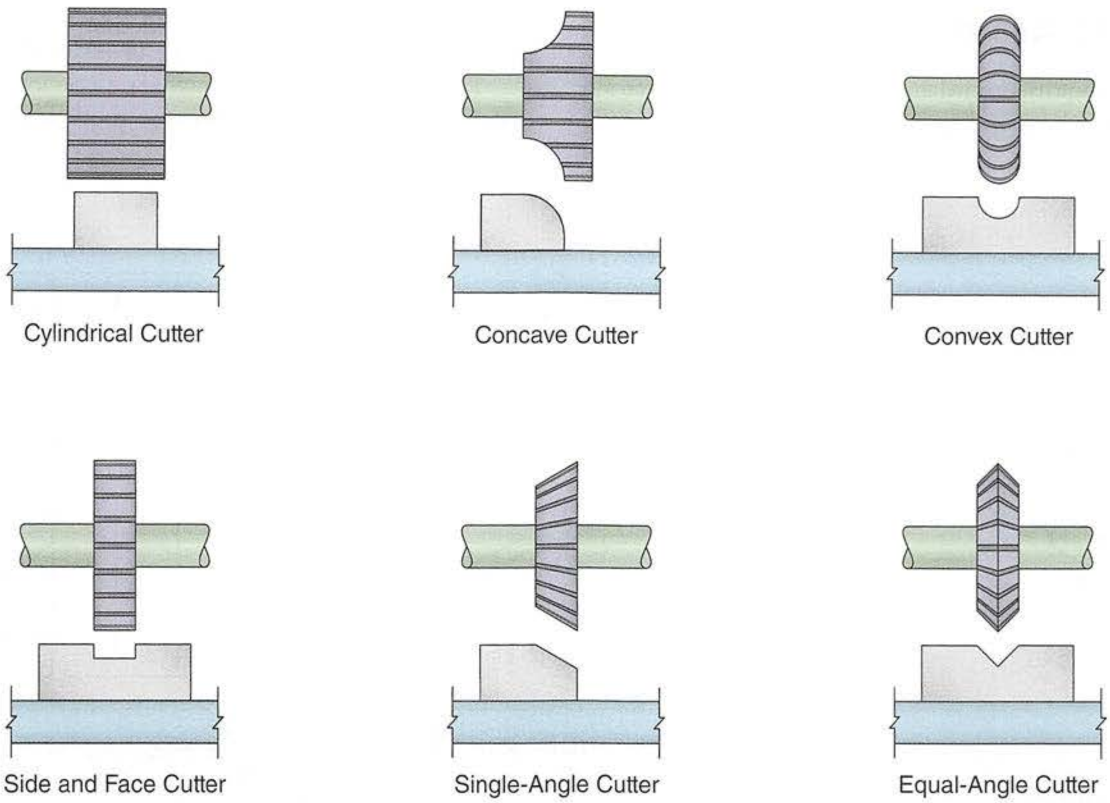
mill. **Figure 18-50** provides several examples of contour and profile cutters.

Many newer machine shops have not invested in what many consider an obsolete technology. However, those shops that still have horizontal milling machines keep them for those jobs for which they are much better suited than the vertical milling machine.

18.4 Milling Machine Care

Follow these guidelines to care for a milling machine:

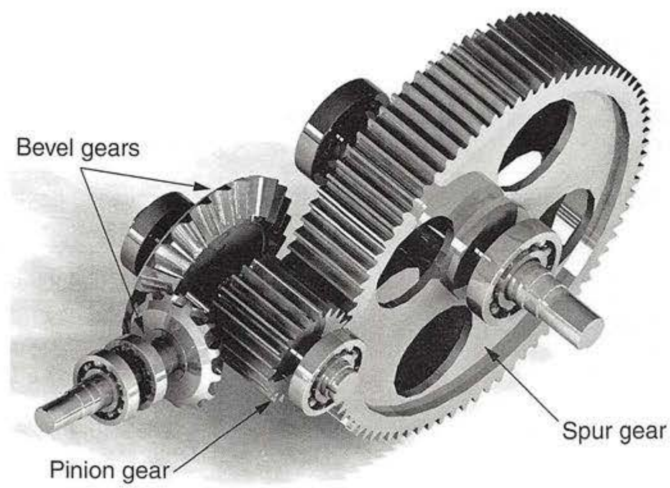
- Check and lubricate the machine with the recommended lubricants.
- Clean the machine thoroughly after each job. Use a brush to remove chips. Never attempt to clean the machine while it is running. Never use your hands.
- Keep the machine clear of tools.
- Check each setup for adequate clearance between the work and the various parts of the machine.



Goodheart-Willcox Publisher

Figure 18-50. Contour and profile cutters that can be stacked on the spindle of a horizontal mill.

- Never force a cutter into a collet or holder. Check to see why it does not fit properly.
- Use a sharp cutter. Protect your hands when mounting it.
- Check the machine to determine whether it is level. Checking should be done at regular intervals.
- Have all guards in place before attempting to operate a milling machine.
- Check coolant level and condition if the reservoir is built into the milling machine. Change it if it becomes contaminated.
- Start the machining operation only after you are sure that everything is in satisfactory working condition. It may be necessary to make special fixtures to hold odd-shaped or difficult-to-mount work.
- Use only attachments designed for the machine.



Eric Milos/Shutterstock.com

Figure 18-51. A few of the many types of gears available. A large spur gear, a pinion gear, and two bevel gears.

18.5 Cutting a Spur Gear

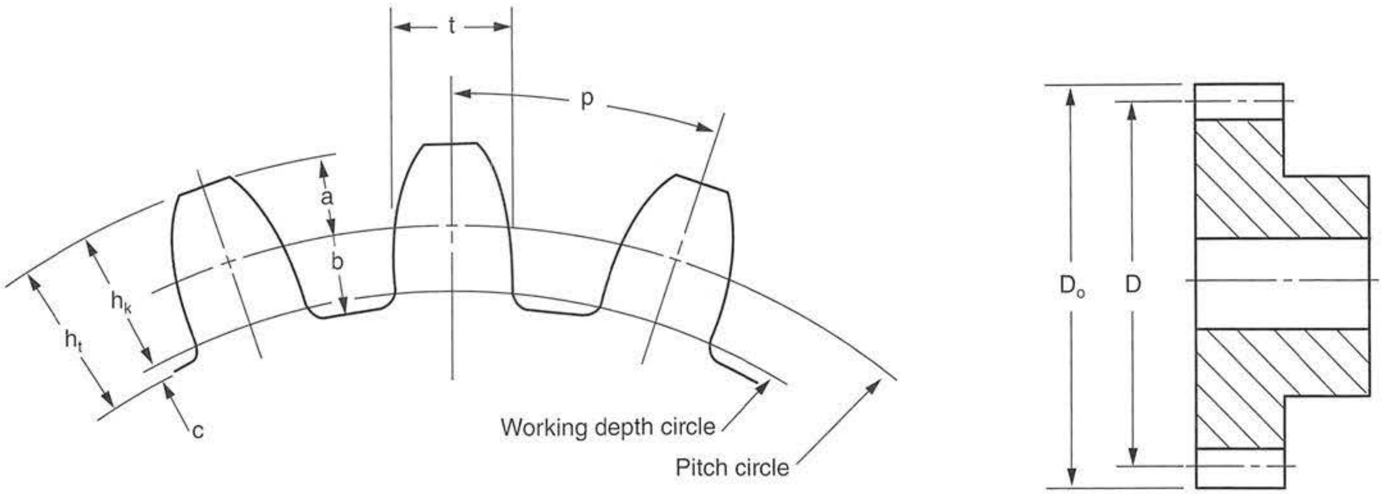
A gear, **Figure 18-51**, is a toothed wheel, usually fitted to a shaft. It typically engages another toothed wheel to smoothly transmit power or motion at a specific ratio between the shafts. The teeth are shaped so that contact between the mating gears is continually maintained while they are in operation.

The ***spur gear*** has teeth that run straight across the face and are perpendicular to the sides. It is the simplest gear and is widely used. Knowledge of gear nomenclature (terminology) is necessary to calculate the data needed to machine a spur gear. Inch-based gears and metric-based gears are not interchangeable.

18.5.1 Gear Nomenclature

Gear cutting requires knowledge of gear nomenclature to aid in determining the proper gear cutter to use, the depth of the teeth, and the dividing head setup. The various gear parts and formulas are shown in **Figure 18-52** and **Figure 18-53**.

The ***pitch circle*** is an imaginary circle located approximately half the distance from the roots and tops of gear



Spur Gear Parts		
Gear Part	**Symbol**	**Definition**
Number of teeth on the gear	N	
Pitch circle		An imaginary circle located approximately half the distance from the roots and tops of gear teeth. It is tangent to the pitch circle of the mating gear.
Pitch diameter	D	The diameter of the pitch circle.
Diametral pitch	P	The number of teeth per inch of pitch diameter.
Circular pitch	p	The distance measured on the pitch circle between similar points on adjacent teeth.
Tooth thickness	t	Thickness of the tooth at the pitch circle. The dimension used in measuring tooth thickness with a vernier gear tooth caliper.
Addendum	a	The distance the tooth extends above the pitch circle.
Dedendum	b	The distance the tooth extends below the pitch circle.
Working depth	h_k	The sum of the addendums of the two mating gears.
Whole depth	h_t	Total depth of a tooth space, equal to the addendum (a) plus dedendum (b), or to the depth to which each tooth is cut.
Clearance	c	The difference between the working depth and the whole depth of a gear tooth. The amount by which the dedendum on a given gear exceeds the addendum of the mating gear.
Center distance	C	Distance between the centers of two mating gears.
Outside diameter	D_o	Diameter or size of gear blank.
Pressure angle	θ	The angle of pressure between contacting teeth of mating gears. It represents the angle at which the forces from the teeth on one gear are transmitted to the mating teeth of another gear. Pressure angles of 14½°, 20°, and 25° are standard. However, the 20° is replacing the older 14½°.

Goodheart-Willcox Publisher

Figure 18-52. Gear nomenclature.

Formulas for Spur Gear Calculations		
To Find	**Rule**	**Formula**
Diametral pitch	Divide 3.1416 by the circular pitch.	P = 3.1416 / p
Circular pitch	Divide 3.1416 by the diametral pitch.	p = 3.1416 / P
Pitch diameter	Divide the number of teeth by the diametral pitch.	D = N / P
Outside diameter	Add 2 to the number of teeth and divide the sum by the diametral pitch.	$D_o = N + 2 / P$
Number of teeth	Multiply the pitch diameter by the diametral pitch.	$N = D \times P$
Tooth thickness	Divide 1.5708 by the diametral pitch.	T = 1.5708 / P
Addendum	Divide 1.0 by the diametral pitch.	a = 1.0 / P
Dedendum	Divide 1.157 by the diametral pitch.	b = 1.157 / P
Working depth	Divide 2 by the diametral pitch.	$h_k = 2 / P$
Whole depth	Divide 2.157 by the diametral pitch.	$h_t = 2.157 / P$
Clearance	Divide 0.157 by the diametral pitch.	c = 0.157 / P
Center distance	Add the number of teeth in both gears and divide the sum by two times the diametral pitch.	$C = N_1 + N_2 / 2P$
Length of rack	Multiply the number of teeth in the rack by the circular pitch.	$L = N \times p$

Goodheart-Willcox Publisher

Figure 18-53. Chart formulas used for gear calculations.

teeth. It is tangent to the pitch circle of the mating gear. The pitch diameter (D) is the diameter of the pitch circle.

Diametral pitch (P) is the number of teeth per inch of pitch diameter. ***Circular pitch*** (p) is the distance measured on the pitch circle between similar points on adjacent teeth.

Tooth thickness (t) is the thickness of the tooth at the pitch circle. This dimension is used when measuring the tooth thickness with a vernier gear tooth caliper. The ***addendum*** (a) is the distance the tooth extends above the pitch circle. The ***dedendum*** (b) is the distance the tooth extends below the pitch circle.

The working depth (h_k) is the sum of the addendums of the two mating gears. The whole depth (h_t) is the total depth of a tooth space, equal to the addendum plus dedendum, or to the depth to which each tooth is cut.

Clearance (c) is the difference between the working depth and the whole depth of a gear tooth. The clearance is the amount by which the dedendum of one gear exceeds the addendum of the mating gear.

18.5.2 Calculations for Milling a Spur Gear

No one gear cutter can be used to cut all gears. Gear cutters are made with eight different forms depending on the number of teeth for which the cutter is to be used. **Figure 18-54** illustrates the comparative sizes for gear teeth. The cutter range is as follows:

No. of Cutter	Range of Teeth
1	135 to a rack
2	55 to 134
3	35 to 54
4	26 to 34
5	21 to 25
6	17 to 20
7	14 to 16
8	12 to 13

With the information furnished, it is possible to calculate the data needed to cut a simple inch-based spur gear.

Example Problem: Calculate the data needed to cut a 40 tooth, 10 diametral pitch gear.

1. Diameter (D_o) of gear blank needed.
   Having:

   Diametral pitch (P) = 10

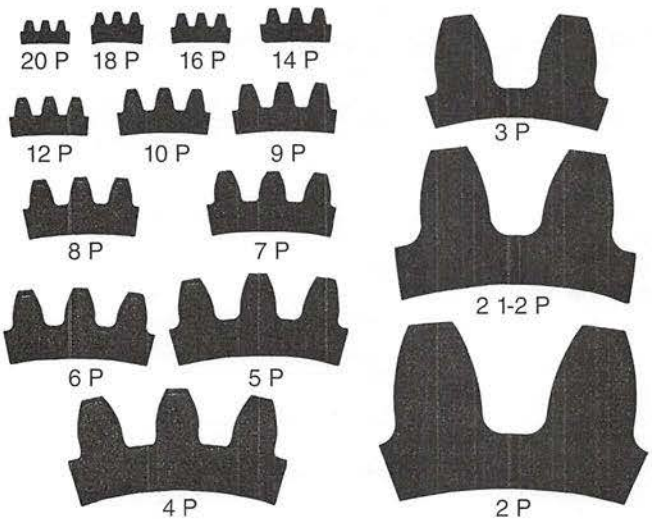
   Number of teeth (N) = 40

   Formula:

$$D_o = \frac{N + 2}{P} = \frac{40 + 2}{10} = \frac{42}{10}$$

$$= 4.200$$

$$= 4.200'' \text{ diameter}$$



Goodheart-Willcox Publisher

Figure 18-54. Comparative sizes of gear teeth. Diametral pitch is shown.

2. Whole depth of tooth (h_t) needed. This will be the depth of the cut.
   Having:

   Diametral pitch (P) = 10

   Formula:

   $$h_t = \frac{2.157}{P} = \frac{2.157}{10} = 0.216$$
   $$= 0.216''$$

3. The dimension of the addendum (a) is needed to measure the gear tooth to determine whether it is being machined to specifications.
   Having:

   Diametral pitch (P) = 10

   Formula:

   $$a = \frac{1}{P} = \frac{1}{10} = 0.100$$
   $$= 0.100''$$

4. Tooth thickness (t) is needed to determine whether the gear is being machined to specifications.
   Having:

   Diametral pitch (P) = 10

   Formula:

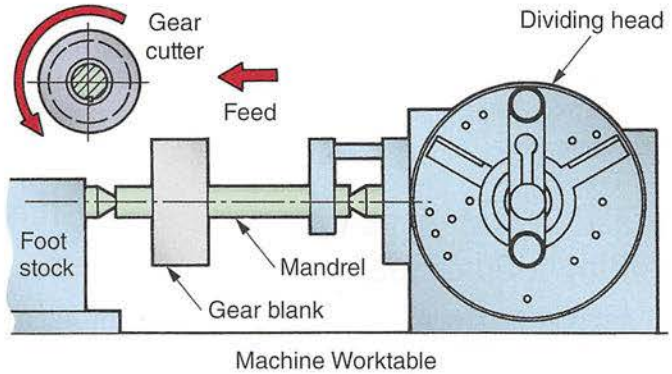
   $$t = \frac{1.5708}{P} = \frac{1.5708}{10} = 0.157$$
   $$= 0.157''$$

5. According to the gear cutter chart, a No. 3 cutter, with a range of 35 to 54 teeth, must be used to cut 40 teeth.
6. Using a 40:1 ratio dividing head for this job means that the index crank must be turned through one complete revolution to position the gear blank for each cut. A 5:1 ratio dividing head would require the use of an index plate that would permit a setting of one-eighth turn for each cut.

18.5.3 Cutting the Gear

Following a few simple precautions can greatly reduce the possibility of an inaccurately machined gear. Follow these guidelines:

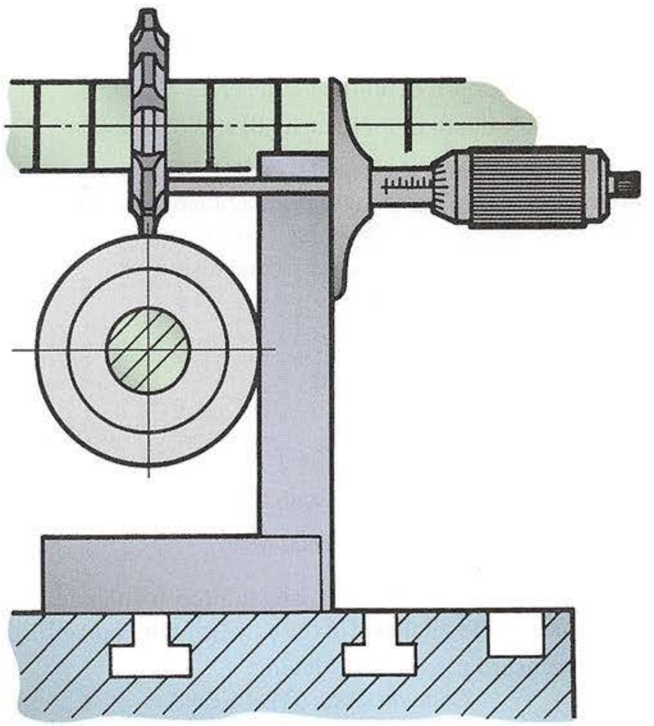
1. Set up the milling machine as previously described. Check the center alignment of the dividing head and foot stock.
2. Press the gear blank onto a mandrel and mount the unit on the dividing head. Cutting is done toward the dividing head, **Figure 18-55**.
3. Use a dial indicator on the gear blank longitudinally and turn the blank through one complete revolution. Make any adjustments that are necessary.
4. Center the cutter on the gear blank. Use a depth micrometer and a steel square, **Figure 18-56**. Position the cutter for depth and use the paper strip technique. Set the micrometer dial to 0, and then raise the table to within 0.040″ of finished depth. For the example problem given earlier, it would be raised to make a cut of 0.216″ – 0.040″ = 0.176″.
5. Move the work until the cutter just begins removing metal. Back it away from the cutter. Using the dividing head, bring the next cut into position. Repeat this sequence around the gear blank until you are back to the original cutting position. If there is exact alignment with the first cut, you are ready to cut the gear.
6. Make the roughing cuts. Use liberal quantities of cutting fluid.
7. The finish cut requires more care. Set the vertical scale on a gear tooth vernier caliper to the distance calculated for the addendum (a), **Figure 18-57**. Raise the work to within a few thousandths of the calculated whole depth of the tooth (h_t) and make cuts at two positions. Make your measurement and adjust until the reading equals



Goodheart-Willcox Publisher

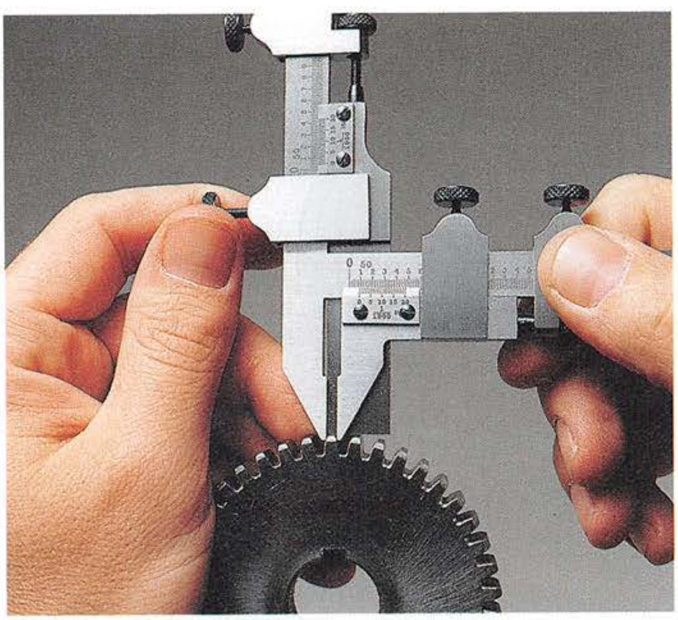
Figure 18-55. Cutting is done toward dividing head.

the distance calculated for tooth thickness (t). Make the finish cuts. Press the completed gear from the mandrel. Remove all burrs and cut the keyway, if required.



Goodheart-Willcox Publisher

Figure 18-56. A depth micrometer can be used to center the gear cutter on a gear blank.



L. S. Starrett Co.

Figure 18-57. This machinist is measuring a gear tooth using a gear tooth vernier caliper. The tool is read in the same manner as a vernier caliper and vernier height gage.

Spur gears can also be measured using Van Keuren wires, **Figure 18-58**. A table furnished with the wire set specifies the wire diameter to use according to the diametral pitch of the gear. The table also includes dimensions for checking external spur gear measurement (M) over the wires. Measurement is made with outside micrometers.

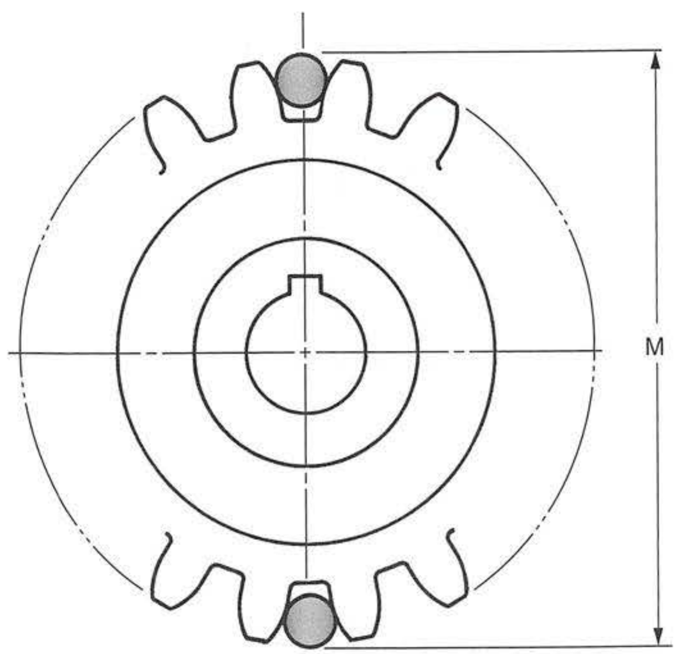
Spiral gears and helical gears are cut on a universal type milling machine utilizing a universal dividing head geared to the table lead screw.

18.6 Cutting a Bevel Gear

Bevel gears, **Figure 18-59**, are used to change the angular direction of power between shafts. The teeth may be either straight or curved. The procedure for cutting a straight tooth bevel gear is explained and illustrated in this section.

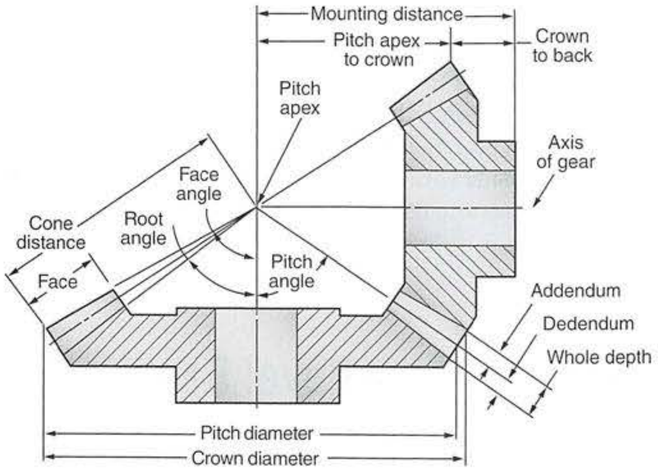
Since tooth space at the pitch diameter is narrower at the small end than at the large end, special form-relieved cutters have been designed to cut bevel gears. To achieve the required gear tooth dimensions, additional material must be removed from the flanks of the teeth after the preliminary roughing operation.

Measurements of the finished gear are made of the blank size and shape, tooth thickness, and depth. However, there is no simple method for checking tooth surfaces. Final inspection is made by running the mating gears and checking for quietness and shape of the tooth contact.



Goodheart-Willcox Publisher

Figure 18-58. Gears can also be measured using Van Keuren wires. Tables furnished with the wire set provide the information needed to measure gears with an even number and odd number of teeth.



Goodheart-Willcox Publisher

Figure 18-59. Nomenclature of bevel gears. The smaller gear is called the *pinion.*

18.6.1 Calculations for Milling a Bevel Gear

The following information was obtained using the dimensions shown in **Figure 18-60** and the formulas given in this chapter.

- Cone distance = 3.535″
- Face width = 1.000″
- Pitch diameter at large end = 5.000″
- Pitch diameter at small end = 3.585″
- Circular pitch at large end = 0.5236″
- Circular pitch at small end = 0.3756″
- Addendum at large end of gear = 0.166″
- Addendum at small end of gear = 0.1195″
- Dedendum at large end of gear = 0.193″
- Dedendum at small end of gear = 0.139″
- Whole depth of tooth at large end = 0.3595″
- Whole depth of tooth at small end = 0.2585″

The tooth parts at the small end of the gear are in exact proportion to those at the large end. Dimensions at the small end can be found by multiplying the dimensions at the large end by the ratio of the respective cone distances.

Determining the ratio:

$$\frac{C_s}{C_r}$$

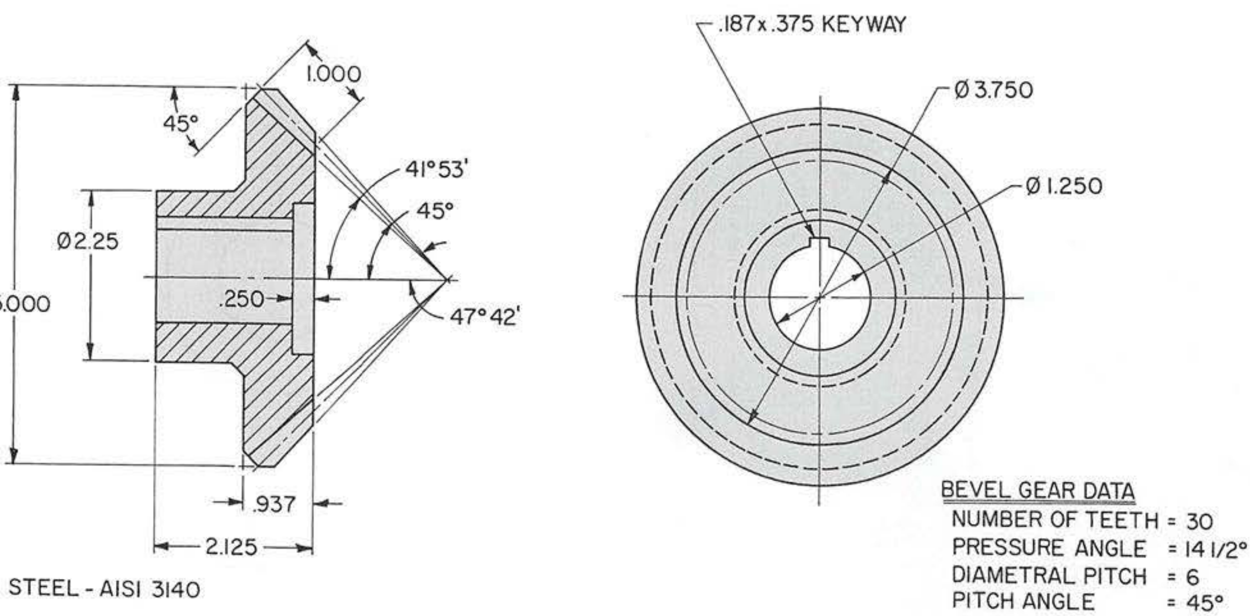
Where:

$$C_s = C_r - F$$
$$C_r = \text{Cone distance}$$
$$F = \text{Face width}$$

Example Problem: Calculate the tooth thickness and tooth space at the small end of the bevel gear in **Figure 18-60**.

1. Bevel gear ratio needed. Having:

$$\text{Cone distance } (C_r) = 3.535''$$
$$\text{Face width } (F) = 1.000''$$



Goodheart-Willcox Publisher

Figure 18-60. Dimensions of the bevel gear to be cut.

Formula:

$$C_s = C_r - F_s = 3.535'' - 1.000'' = 2.535''$$

$$\frac{C_s}{C_r} = \frac{2.535''}{3.535''} = 0.717$$

Ratio:

$$\frac{C_s}{C_r} = 0.717$$

2. Tooth thickness and tooth space at the small end (t_s) needed. Having tooth thickness and tooth space at the large end:

$$t_L = 0.2618''$$

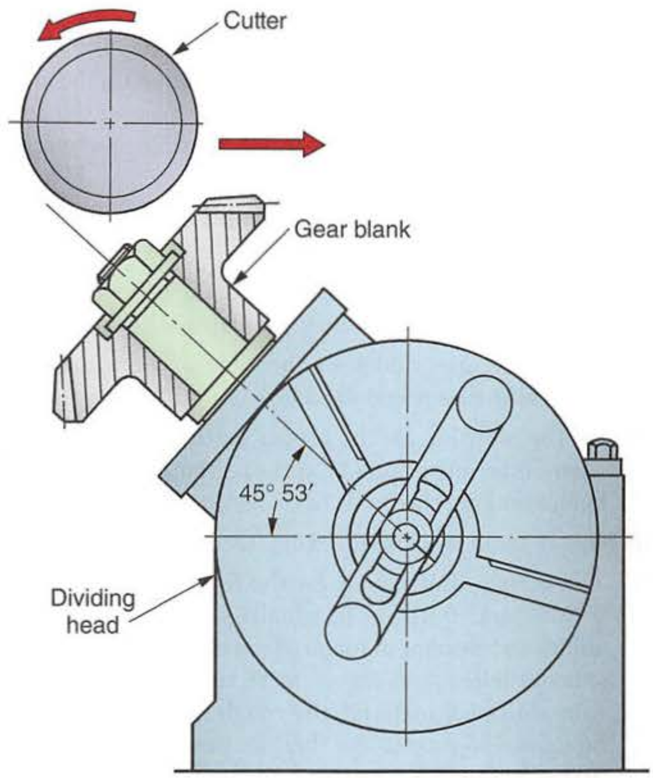
Formula:

$$t_s = t_L \times \frac{C_s}{C_r} = 0.2618'' \times 0.717 = 0.1877$$

$$= 0.1877''$$

18.6.2 Preparing to Cut a Bevel Gear

1. Mount the dividing head and tilt it to 45° 53′. See **Figure 18-61**.
2. Calculate the correct index plate to cut 30 teeth.
3. Secure the gear blank on an arbor.



Goodheart-Willcox Publisher

Figure 18-61. Tilt the dividing head to the required angle (45° 53′) and mount the gear blank.

CAREER CONNECTION

Machinist

What does a machinist do?

A machinist uses manual or CNC machinery to machine precision metal parts. Machinists working on manual machines may use grinders, lathes, milling machines, and other machines. CNC machines automate these processes after the machinist sets their cutting path, cutting speed, and feed rate.

What education and skills are needed to be a machinist?

High school diplomas are required for most machinist jobs, and students should prepare themselves further with courses in the following areas:

- Advanced math, such as trigonometry and geometry.
- Technical subjects, such as print reading, metalworking, and drafting.
- Computer courses in preparation for CNC programming.

More advanced positions require more advanced education. Apprenticeship programs are available, as well as training programs from community colleges and technical schools.

What is it like to be a machinist?

The demands placed on machinists vary by industry. Machinists working on factory floors respond to orders for customized parts as machines need repair. Other machinists may oversee the production of large batches of a single part.

According to the *Occupational Outlook Handbook*, the median annual wage for machinists has been $40,500 in recent years.

4. Mount the gear blank and arbor in the dividing head with the large end of the gear toward the dividing head.
5. Select the proper bevel cutter and mount it on the arbor. Cutting is toward the dividing head. Position the arbor bearing as close to the cutter as possible, allowing for adequate clearance.
6. Center the cutter on the gear blank. Lock the cross slide. Set the graduated collar to 0.
7. Position the cutter for depth, using the paper strip technique. Set the knee graduated collar to 0.
8. Clear the cutter and raise the table to the whole tooth depth at the large end (0.3595″).
9. Set the machine for the proper cutting speed and feed. Many machinists prefer to make the cut in two runs before making the final cut.
10. Cut all teeth by plain indexing. Use adequate coolant.
11. The correct dimensions for the finished gear teeth are 0.2618″ and 0.1878″. To obtain these dimensions, an additional amount of material must be removed on each side of the teeth, as shown in **Figure 18-62**. To remove this additional material, the rough finished blank must be rotated slightly (2° for this gear) and the table set over (0.044″). The following formulas were used to make the calculations.

Determining angle of roll:

$$A = \frac{57.3}{D}\left[\frac{p}{2} - \frac{C_r}{W}(T_L - T_s)\right]$$

Where:

A = Angle of roll in degrees
D = Pitch diameter at large end of gear
p = Circular pitch at large end of gear
C_r = Pitch cone distance at large end of gear
T_s, T_L = Chordal thickness of gear cutter corresponding to pitch line at small and large ends of gear, respectively
57.3 = Degrees per radian
W = Width of gear tooth face

Determining table setover:

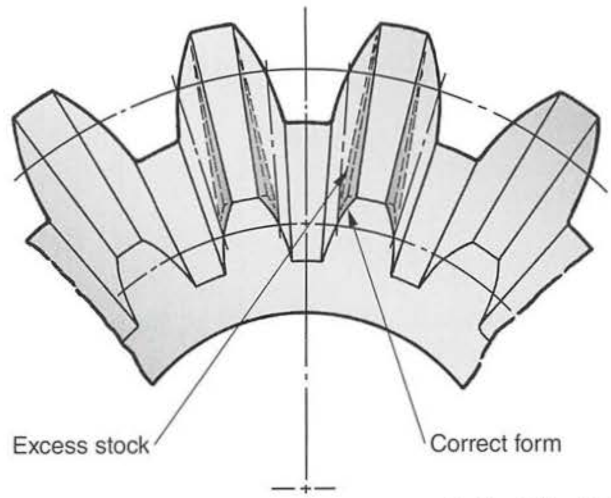
$$n = \frac{T_L}{2} - T_L - \frac{T_S}{2} \times \frac{C_r}{W}$$

Where:

n = Table setover

The direction of roll and setover must always be made in opposite directions. See **Figure 18-63**. Remove all backlash before making work movements.

12. Finish machining the gear. Remove all burrs. Measure tooth thickness at the small and large ends. It may be necessary to remove a slight amount of material at the small end to get proper meshing. Refer to **Figure 18-62**.



Goodheart-Willcox Publisher

Figure 18-62. Excess material that must be removed when using the described technique to cut a bevel gear.

This method of making bevel gears does not produce a tooth form that is accurate throughout the length of the tooth face. This is especially true at the small end of the gear even though the tooth form is correct at the large end. Remove this small amount of excess material by rotating the blank through a small angle with the dividing head and taking light cuts until proper meshing is attained.

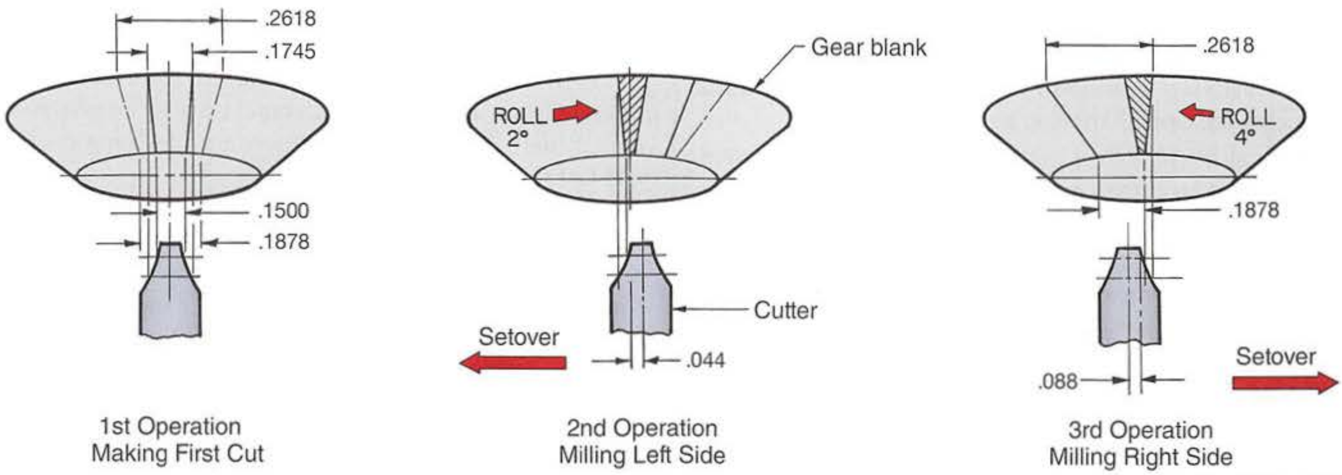
18.7 Thread Milling

For years, threads were milled on the lathe and special thread milling machines. The technique was used to produce coarse pitch threads that required the removal of relatively large quantities of metal.

Today, CNC machining centers with helical interpolation can mill high-quality threads in holes ranging from 3/8″ (9.5 mm) in diameter to diameters of almost any size. Helical interpolation creates tool movement in a helical path and requires simultaneous movement in the X, Y, and Z axes. In thread milling, circular movement in the X and Y axes create thread diameter. Thread pitch is created by linear movement in the Z axis. See **Figure 18-64**. Right-hand and left-hand internal and external threads can be milled.

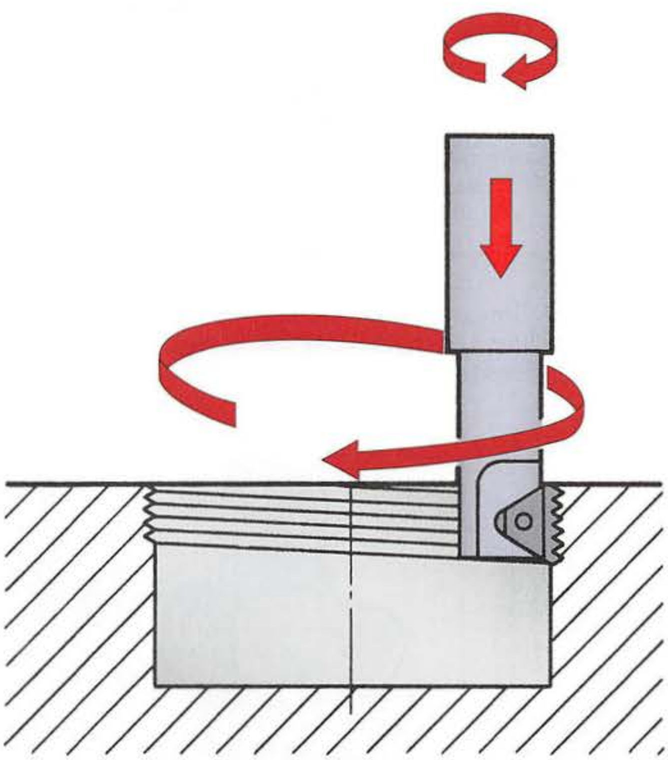
Thread milling is ideal for generating precision threads in work that is not symmetrical or is too large to be mounted on a lathe. Large threads are very expensive to generate by conventional methods. Since most threads can be milled in a single pass of the cutting tool, the process saves costs by greatly reducing production time.

Thread milling cutters are available in a wide range of sizes. See **Figure 18-65**. Most cutters have carbide inserts. The type of tool used is determined by thread size and depth, class of fit, type of material machined, and its relative hardness. Internal threads and external threads can be milled using the same tool.



Goodheart-Willcox Publisher

Figure 18-63. Sequence to be followed when milling the teeth of a bevel gear.



Goodheart-Willcox Publisher

Figure 18-64. In thread milling, helical interpolation creates tool movement in a helical path. Thread diameter is created by the circular movement in the X and Y axes. The simultaneous linear movement in the Z axis creates the pitch of the thread.

Greenfield Industries

Figure 18-65. Typical thread milling cutters. The pitch of the cutter teeth must match the thread pitch.

18.8 Milling Machine Safety

Follow these guidelines when working with a milling machine:

- Avoid performing any machining operation on the milling machine until you are thoroughly familiar with how it should be done.
- Some materials that are machined can produce chips, dust, and fumes that are dangerous to your health. Never machine materials that contain asbestos, fiberglass, beryllium, or beryllium copper unless you are fully aware of the precautions that must be taken.
- Make sure there is adequate ventilation when performing jobs where dust and fumes are a hazard.
- If the area where you work is extremely noisy, wear hearing protectors. Take no chances. Protect your hearing and sight at all times in the shop.
- Maintain cutting fluids properly. Discard them when they become rancid or contaminated. Never pour used coolants or solvents down the drain.
- Carefully read instructions when using synthetic oils, solvents, and adhesives. Many are dangerous if

not handled correctly. Return all oils and solvents to proper storage. Always wipe up spills.

- Never start a cut until you are sure there is adequate clearance on all moving parts.
- Be sure the cutter rotates in the proper direction. Expensive cutters can be quickly ruined.
- Carefully store milling cutters, arbors, collets, adapters, and other tools after use. They can be damaged if not stored properly.
- Exercise care when handling long pieces of metal. Accidentally contacting a light fixture or busbar can cause severe electrical burns and even electrocution.

18.9 Industrial Applications

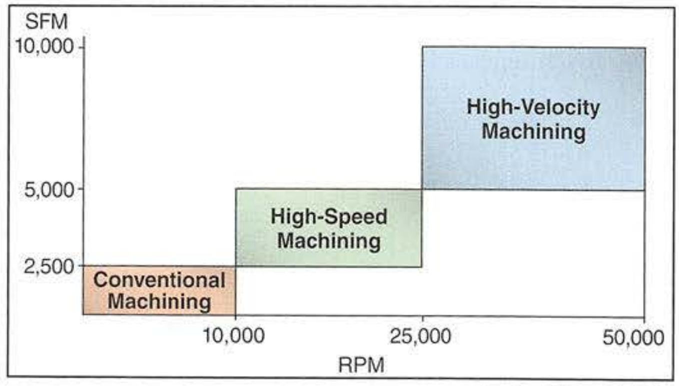
The milling machines found in industry operate on the same basic principles as those found in training programs. In many cases, the same equipment is used, **Figure 18-66.**

Milling machines in use today range in size from small tabletop models to machines capable of handling work that weighs many tons. Most milling machines have CNC capability.

18.10 High-Velocity Machining

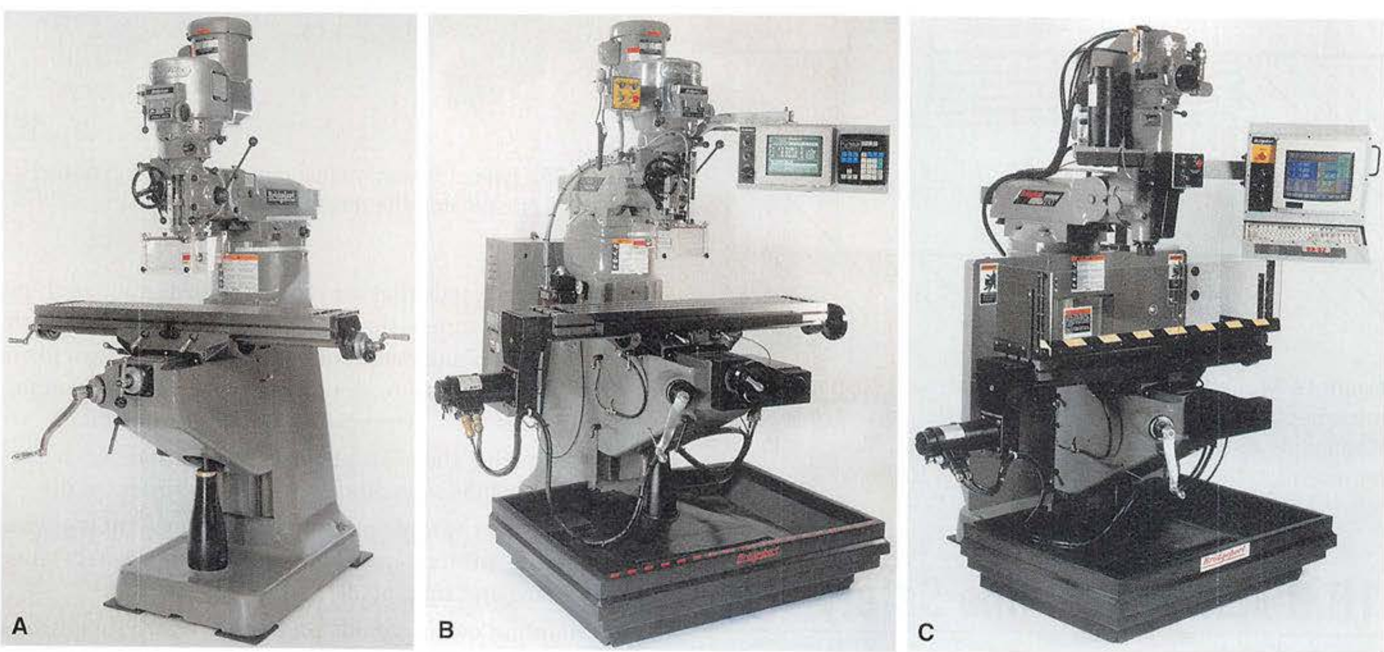
High-velocity machining is often referred to as *high-speed machining*, but the term *high-velocity machining* is preferred because it implies more than high spindle speeds. See **Figure 18-67.** In addition to high spindle speeds (25,000 rpm and higher), high-velocity machining also involves rapid feed rates (600 ipm and higher), high rapid traverse rates to reduce noncutting time, high acceleration and deceleration rates, and fast tool changes.

To meet these requirements, special bearings, motors, highly responsive servo systems, spindle cooling techniques, and rapid chip removal methods had to be developed. Cutting tools and toolholders must also be carefully balanced. CNC programs for high-velocity machining also require large amounts of computer data.



Goodheart-Willcox Publisher

Figure 18-67. This graph illustrates how the three machining techniques differ. The high-velocity machining range is expected to increase as new equipment is developed.



Bridgeport Machines, Inc.

Figure 18-66. Three types of vertical milling machines found in both industry and training programs. A—Manually operated vertical milling machine. B—CNC 2-axis vertical milling machine. C—CNC 3-axis vertical milling machine.

High-velocity machining is used primarily by the aerospace industry. The one-piece aluminum structural components used in aerospace craft are "hogged," or carved from large sections of metal. See **Figure 18-68**. Most weigh less than 5% as much as the aluminum plate from which they are machined. These parts are stronger, lighter, and easier to join to other sections. They are less expensive to produce than similar parts fabricated from many smaller sheet metal pieces by conventional techniques. Contoured surfaces can be finished so smoothly that little or no additional finishing work is needed. Dies and other expensive tooling are eliminated. The safety factor is also increased because no riveting is required—each rivet hole is a source of potential structural weakness.

MAG IAS, LLC

Figure 18-68. Typical aircraft structural element carved from a solid aluminum section with a 5-axis horizontal profiler. It is much stronger than a similar part fabricated from aluminum sheet. Many of these sections weigh less than 5% as much as the aluminum plate from which they are cut.

Chapter Review

Summary

- Squaring, machining flat and angular surfaces, machining keyseats and slots, and face and side milling are all common milling operations.
- Vertical milling machines are capable of milling, drilling, boring, and reaming operations.
- Horizontal milling machines are capable of milling, slitting, slotting, drilling, and boring operations.
- Proper care of a milling machine includes checking the machine at regular intervals, lubricating it with the recommended lubricants, and checking the coolant level and condition.
- Cutting spur and bevel gears requires knowledge of gear nomenclature, gear cutters, and specific calculations.
- Chips, dust, fumes, cutting fluids, and noise are all potential hazards when milling.
- Many milling machines found in industry are similar to training machines.
- High-velocity machining is used primarily by the aerospace industry.

Review Questions

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

1. A(n) _____ scale on the spindle head of a vertical milling machine ensures accurate angular settings.
2. _____ mills are the cutters normally used on a vertical milling machine.
   A. Face
   B. Spur
   C. End
   D. Both A and B.
   E. Both A and C.
3. The most accurate way to align a vise on a milling machine is with a(n) _____.
4. List two methods for machining chamfers, bevels, and tapered sections on a vertical milling machine.
5. Blind keyseats or slots are made with a(n) _____ end mill.
6. Explain how to center an end mill on round stock for the purpose of machining a keyseat using the paper strip technique.
7. A(n) _____ end mill is used when the cutter must be fed into the work like a drill.
8. List four devices that can be used to locate the first hole of a series to be drilled on a vertical milling machine.
9. What cutters are commonly used to mill flat surfaces?
10. Why should a milling cutter be keyed to the arbor?
11. Describe how to safely mount a milling cutter on an arbor.
12. Face milling cutters over 6″ (150 mm) in diameter are usually of the _____ type.
13. What is gang milling?
14. When slitting thin stock, the _____ diameter saw that provides adequate clearance should be used.
15. What should you do if a milling cutter does not fit properly into a collet or holder?
16. What is a spur gear?
17. What is the difference between diametral pitch and circular pitch of a gear?
18. How would a dividing head be set up to cut a 100-tooth gear? The dividing head has a 40:1 ratio and the index plate has the following series of holes: 33, 37, 41, 45, 49, 53, 57.
    Number of full turns. _____
    Hole series used. _____
    Number of holes in sector arm spacing. _____
19. What is the purpose of a bevel gear?
20. The parts of the teeth at the small end of a bevel gear are in exact _____ to those at the large end of the gear.
21. What method is most commonly used today to mill threads?
22. List five precautions to be observed when operating a milling machine.
23. How do milling machines in industry compare with those in training programs?
24. What are the advantages of using high-velocity machining to carve aerospace parts from large sections of metal instead of fabricating the parts from smaller pieces of sheet metal?

CHAPTER 19

Precision Grinding

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Explain how precision grinders operate.
- Identify the various types of precision grinding machines.
- Describe the work-holding devices used with surface grinders.
- Select, dress, and true grinding wheels.
- Identify types of cutting fluids used in grinding operations.
- Safely operate a surface grinder using various work-holding devices.
- Solve common surface grinding problems.
- List safety rules related to precision grinding.
- Identify other types of precision grinding operations, including cutter sharpening, cylindrical grinding, internal grinding, centerless grinding, form grinding, and other grinding techniques.

Technical Terms

centerless grinding
creep grinding
dwell
electrolytic grinding
form grinding
internal grinding
platens
plunge grinding
precision grinding
tooth rest
traverse grinding
universal tool and cutter grinder

Grinding, like milling, drilling, sawing, and turning, is a cutting operation. However, instead of using one, two, or several cutting edges, grinding makes use of an abrasive tool composed of thousands of cutting edges. See **Figure 19-1**. Since each of the abrasive particles is actually a separate cutting edge, the grinding wheel might be compared to a many-toothed milling cutter.

In ***precision grinding***, each abrasive grain removes a minute amount of material. It is one of the few machining operations that can produce a smooth, accurate surface on material regardless of its hardness. Grinding is frequently used as a finishing operation.

19.1 Types of Surface Grinders

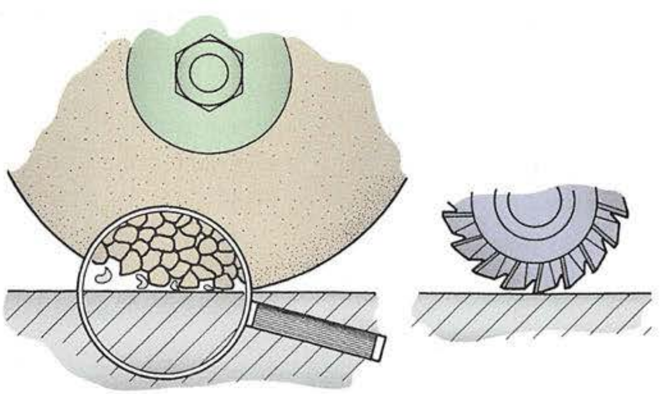
Although all grinding operations might be called *surface grinding* because all grinding is done on the surface of the material, industry classifies surface grinding as the grinding of flat surfaces. There are two basic types of surface grinding machines:

- Planer surface grinders use a reciprocating motion to move the worktable back and forth under the grinding wheel. Three variations of planer surface grinding are illustrated in **Figure 19-2**.
- Rotary surface grinders have circular worktables that revolve under the rotating grinding wheel. Two variations of the technique are shown in **Figure 19-3**.

The planer surface grinder is frequently found in training situations. It slides the work back and forth under the edge of the grinding wheel. Table movement can be controlled manually or by means of a mechanical or hydraulic drive mechanism.

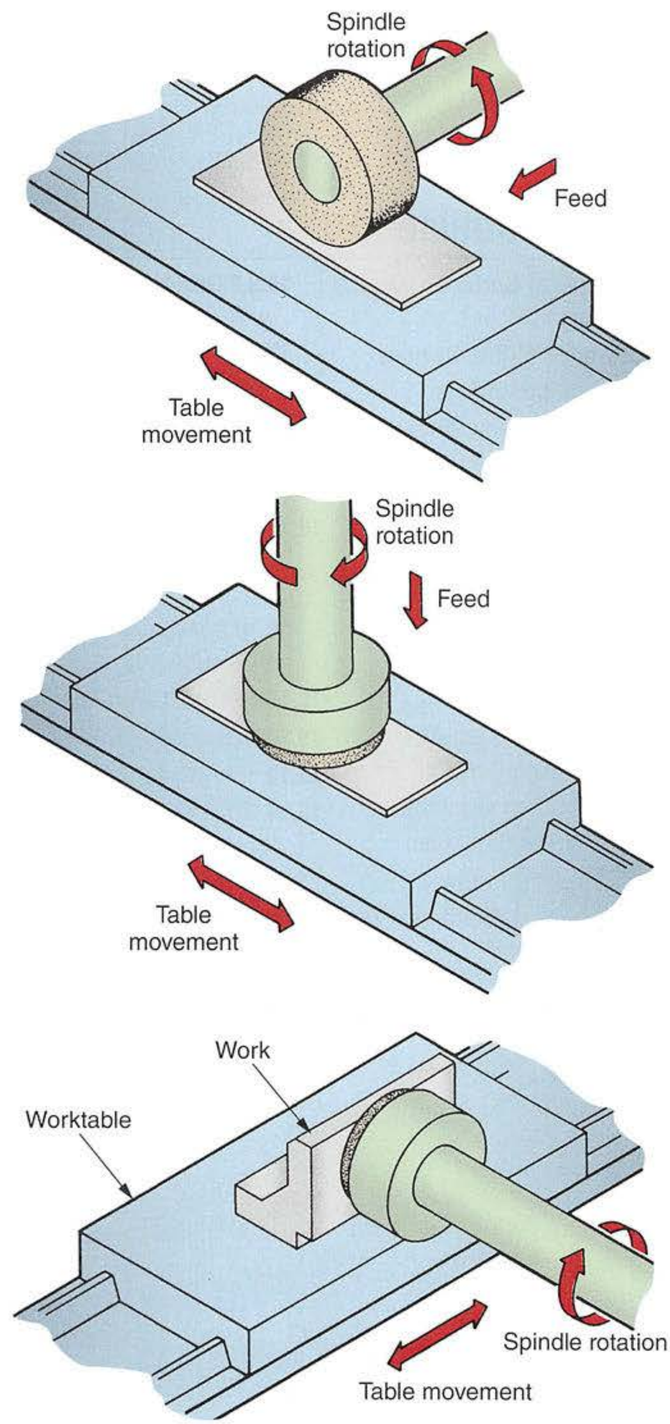
On a manually operated machine, all work and grinding wheel movements are made by hand. The large traverse handwheel moves the table to the left and right. The smaller cross-feed handwheel moves the table in and out. The down-feed handwheel moves the grinding wheel up and down. This handwheel is located on the top of the vertical column.

Variations of this type of surface grinder can be run manually or automatically. To prepare the machine for automatic operation, the operator simply fills in the blanks when requested by the menu prompts. No computer or CNC experience is needed. As the operator creates the part manually,



Goodheart-Willcox Publisher

Figure 19-1. A grinding wheel removes material in the same manner as a milling cutter, but the chips of metal removed are much smaller.



Goodheart-Willcox Publisher

Figure 19-2. Three variations of the planer surface grinder.

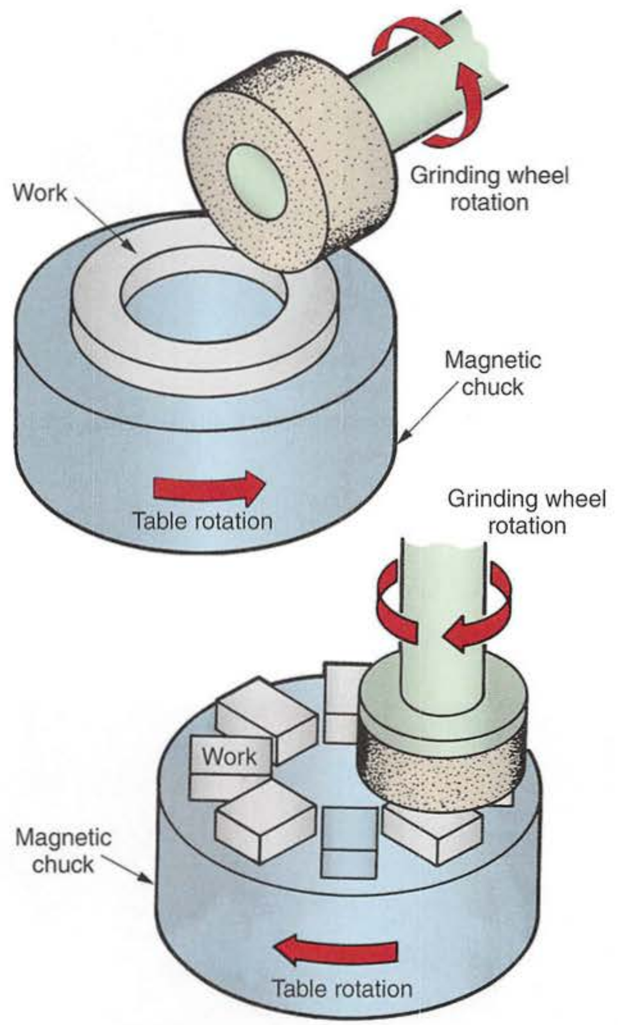


Figure 19-3. Two variations of the rotary-type surface grinder.

pressing a button after each move, the machine "memorizes" how the part is made. The machine can run subsequent parts automatically.

Manual and automatic machines operate in much the same manner. However, the person using a manually operated machine must develop a rhythm to get a smooth, even cutting stroke. Spring stops act as cushions at the end of maximum table travel, **Figure 19-4**.

Adjustable table stops on the hydraulically activated traverse feed permit the operator to position the table precisely, **Figure 19-5**. At the end of the stroke, table direction reverses automatically. The automatic cross-feed moves the work in or out a predetermined distance at the completion of each cutting cycle.

A control console is located on the front of the machine. From this console, the operator can start and stop table travel and control table speed. Some grinding machines have a control for ***dwell***—a hydraulic cushion at the end of each stroke.

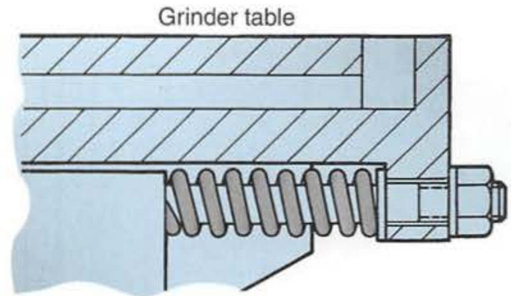


Figure 19-4. Springs are often used on the worktable guide to cushion the end of the stroke on manually operated surface grinders.

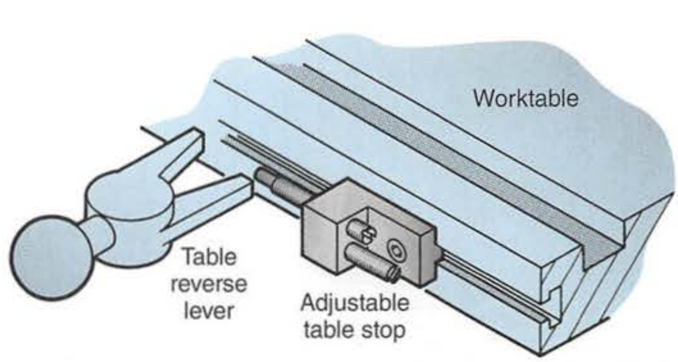


Figure 19-5. An adjustable table stop is used to regulate the length of the worktable stroke.

19.2 Work-Holding Devices

Much of the work machined on a surface grinder is held in position by a magnetic chuck. The chuck holds the work using magnetic force. Nonmagnetic materials (such as aluminum and brass) can be ground by bracing them with steel blocks or parallels to prevent movement.

An electromagnetic chuck uses an electric current to create a strong magnetic field. Another type of magnetic chuck uses a permanent magnet. This eliminates the cords needed for electromagnets and the danger of the work flying off the chuck if the electrical connection is accidentally broken. See **Figure 19-6**.

Frequently, work mounted on a magnetic chuck becomes magnetized and must be demagnetized before it can be used. A demagnetizer is used to neutralize the magnetic field.

Other ways to mount work on a surface grinder include:

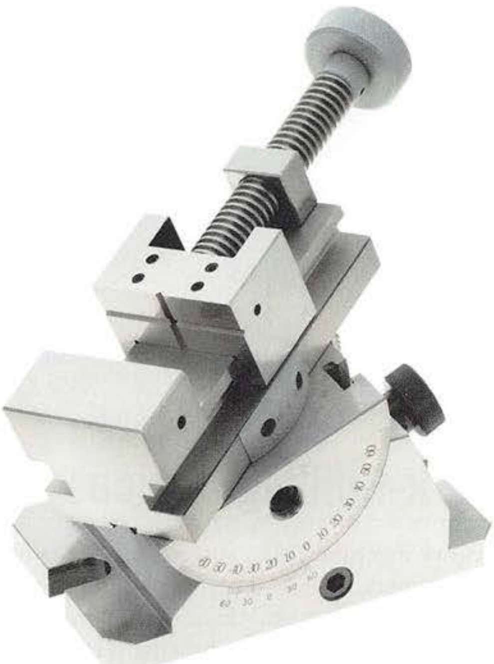
- A universal vise, **Figure 19-7A**.
- An indexing head with centers, **Figure 19-7B**.
- Clamps to hold the work directly on worktable.
- A precision vise.
- Double-faced masking tape (used to hold thin sections of nonmagnetic materials). Refer to **Figure 19-8**.

A

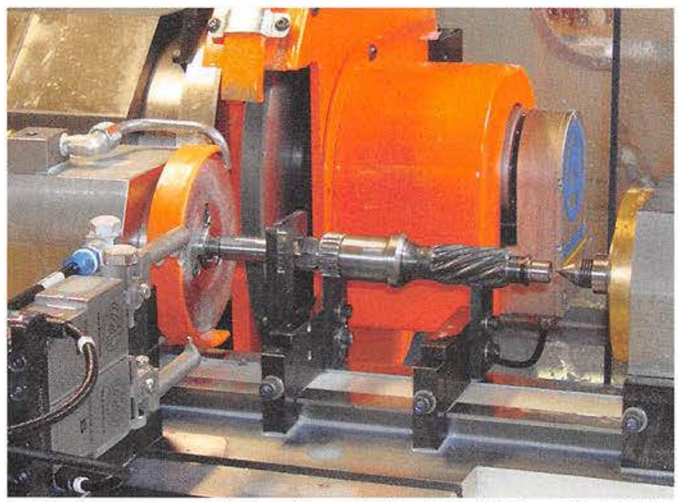
B

O.S. Walker Co.

Figure 19-6. Magnetic chucks. A—An electromagnetic chuck uses an electric current to create a magnetic field. B—A magnetic chuck with a permanent magnet.



A Photo Courtesy of Grizzly Industrial, Inc. www.grizzly.com



B Courtesy of Vermont Machine Tool and the Bryant Grinder Division

Figure 19-7. Work-holding devices. A—A universal vise can be used for grinding operations. B—Centers and an indexing head are used for grinding tasks when the shape of the work permits. An indexing head is used in much the same manner as the dividing head in milling.

19.3 Grinding Wheels

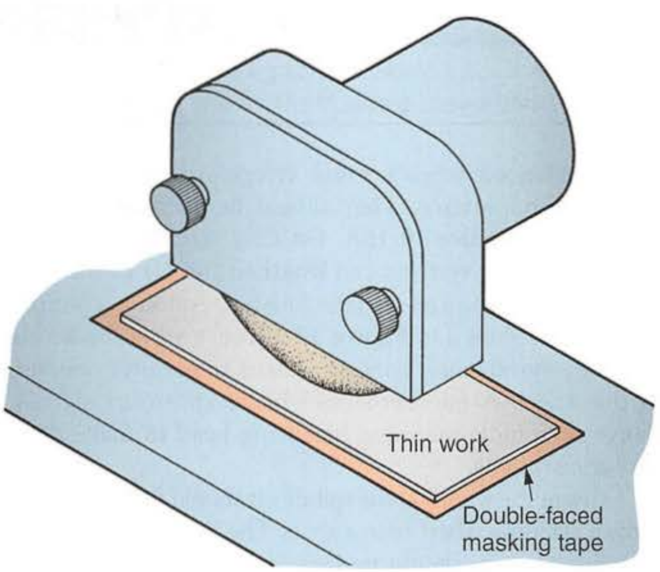
As mentioned at the beginning of this chapter, each abrasive particle in a grinding wheel is a cutting tooth. As the wheel cuts, metal chips dull the abrasive grains and wear away the bonding material (medium that holds the abrasive particles together). In the ideal grinding wheel, the bonding material would wear away slowly enough to get maximum use from the individual abrasive grains, but rapidly enough to permit dulled abrasive particles to drop off and expose new particles.

Because so many factors affect grinding wheel efficiency, the wheel eventually dulls and must be dressed with a diamond dressing tool. See **Figure 19-9**. Failure to dress the wheel of a precision grinding machine will, in time, result in the wheel face becoming loaded or glazed and unable to cut freely.

Only manufactured abrasives are suitable for high-speed grinding wheels. Their properties, including the spacing of abrasive particles and composition of the bonding medium, can be controlled to get the desired grinding performance, **Figure 19-10**.

19.3.1 Grinding Wheel Marking System

To help ensure grinding performance, a standard system of marking grinding wheels has been defined by the American



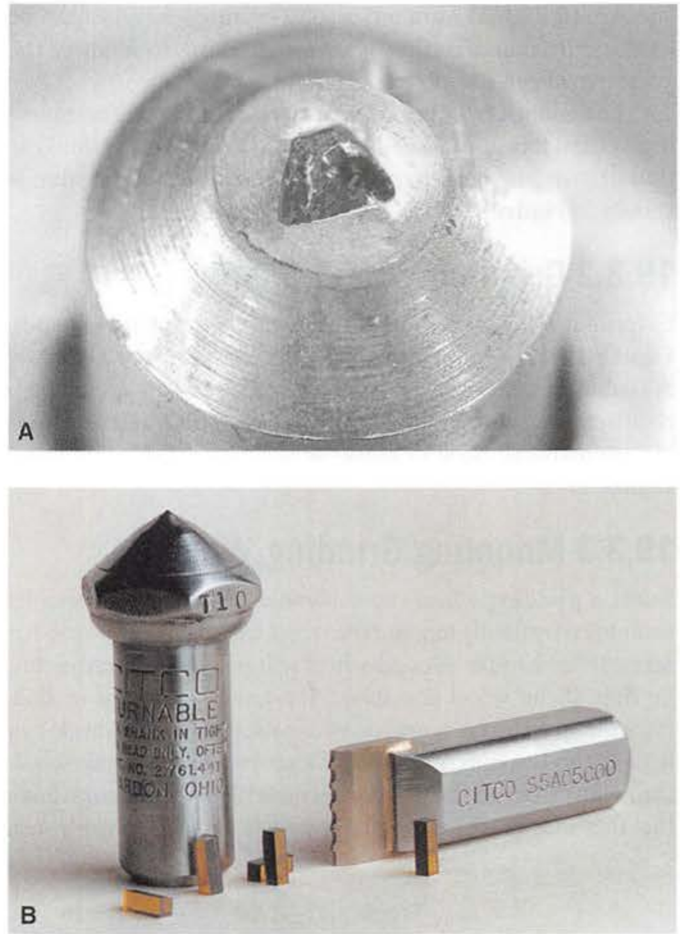
Goodheart-Willcox Publisher

Figure 19-8. Double-faced masking tape is used to mount thin, nonferrous material for grinding.

National Standards Institute (ANSI). Called ANSI Standard B74.13, it is used by all grinding wheel manufacturers. Five factors were considered:

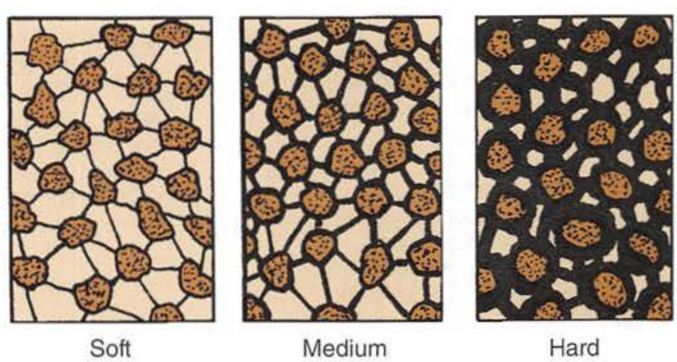
- **Abrasive type.** Classifies the abrasive material in the grinding wheel. Manufactured abrasives fall into two main groups, identified by letter symbols:
  - A = aluminum oxide
  - C = silicon carbide

  An optional prefix number may be used to designate a particular type of aluminum oxide or silicon carbide abrasive.
- **Grain size.** Indicated by a number, usually from 8 (coarse) to 600 (very fine).
- **Grade.** The strength of the bond holding the wheel together, ranging from A (soft) to Z (hard).
- **Structure.** Grain spacing or the manner in which the abrasive grains are distributed throughout the wheel. It is numbered 1 to 16. The higher the number, the more "open" the structure (wider the grain spacing). The use of this number is optional.
- **Bond type.** The type of material that holds the abrasive grains (wheel) together. Eight types are used:
  - B = Resinoid
  - BF = Resinoid reinforced
  - E = Shellac
  - O = Oxychloride
  - R = Rubber
  - RF = Rubber reinforced
  - S = Silicate
  - V = Vitrified



CITCO Div., Western Atlas, Inc.

Figure 19-9. Diamond wheel dressing tools. A—Close-up of a natural diamond chip on a grinding wheel dresser. The diamond should be rotated a partial turn each time it is used to put a new edge of the diamond into position. B—Dressing tools manufactured from manmade diamonds. These diamonds do not have the irregularities of natural diamonds, resulting in consistent diamond exposure and longer wear.



Goodheart-Willcox Publisher

Figure 19-10. Wheel hardness is determined by the type and percentage of bond and grain spacing.

An additional number or one or more letters may be used as the manufacturer's private marking to identify the grinding wheel. Its application is optional.

The adoption of this standardized grinding wheel marking system has guaranteed, to a reasonable degree, duplication of grinding performance. The wheel marking system is shown in **Figure 19-11**.

19.3.2 Grinding Wheel Shapes

Grinding wheels are made in many standard shapes, **Figure 19-12**. Only twelve basic face shapes are generally available, but these faces may be changed to suit specific job requirements, **Figure 19-13**. Wheels used for internal grinding are manufactured in a large selection of shapes and sizes, **Figure 19-14**.

19.3.3 Mounting Grinding Wheels

Select a grinding wheel recommended for the job. Check its soundness by lightly tapping the wheel with a plastic or wooden screwdriver handle. A good wheel will produce a clear metallic ring. If the wheel is cracked, the tone will be flat or dull, rather than a clear ringing sound. Cracked grinding wheels can fly apart as the rpm of the wheel increases to operating speed, throwing pieces off the wheel in all directions with enough force that they may cause injury even to people outside the work area.

SAFETY NOTE

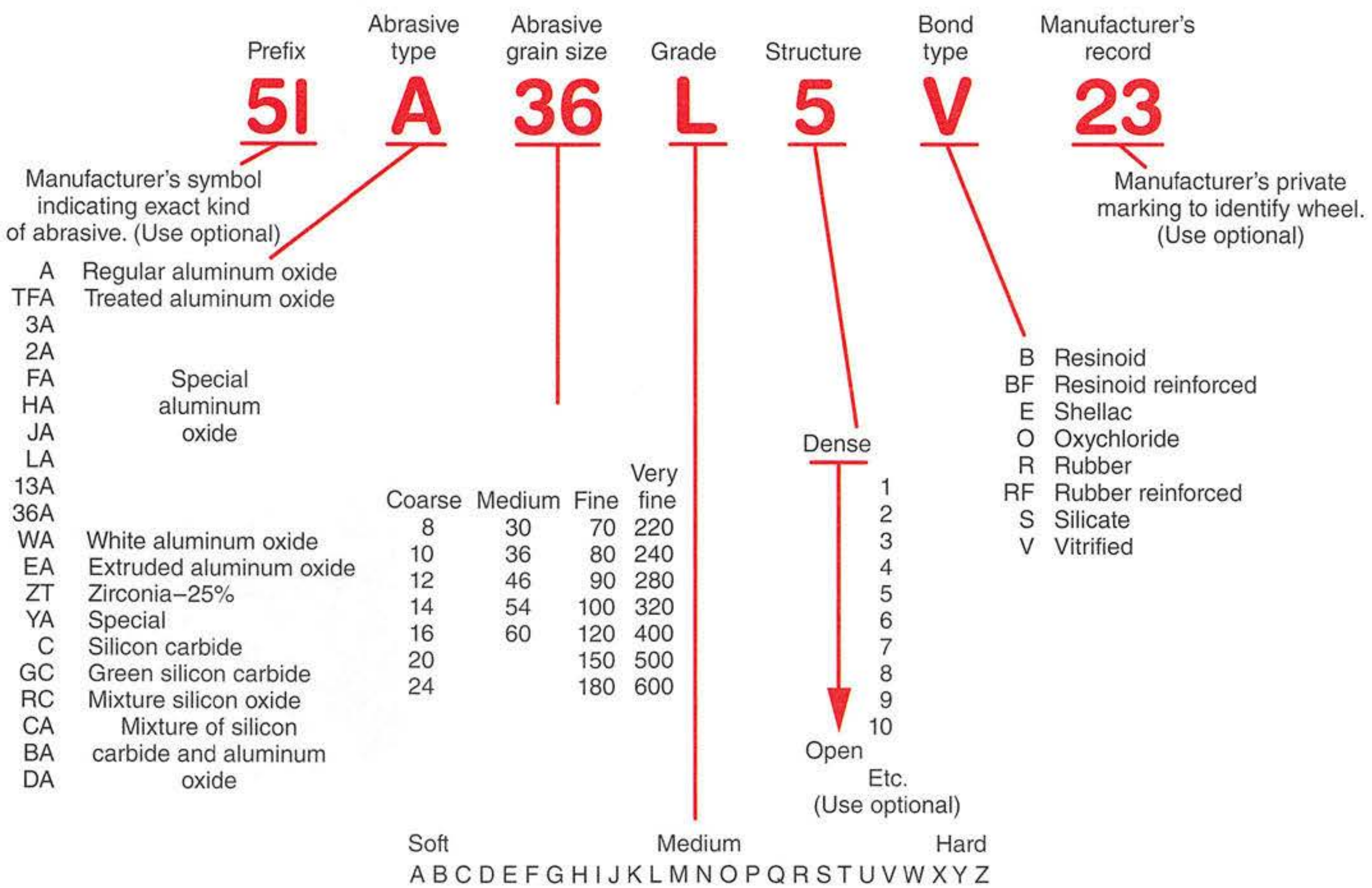
Always discard cracked grinding wheels. If possible, break them into several pieces to ensure that they are not used.

Unbalanced wheels cause irregularities on the finished ground surface. They should be statically balanced as shown in **Figure 19-15A**. On CNC grinders, automatic wheel balancing systems can lengthen the life of the wheel and provide improved surface finishes. Automatic systems like the one shown in **Figure 19-15B** use vibration sensors and ultrasound wheel contact sensors to monitor operation of the wheel. A microprocessor-based controller signals a flange or spindle-mounted balancing head to make necessary adjustments.

Mount the wheel on the spindle. It should fit snugly. Never force a grinding wheel onto a shaft. The blotter rings or compressible washers should be large enough to extend beyond the wheel flanges, **Figure 19-16**. It is essential that the wheel be mounted properly. If it is not, excessive strains will develop during the grinding operation and the wheel may shatter.

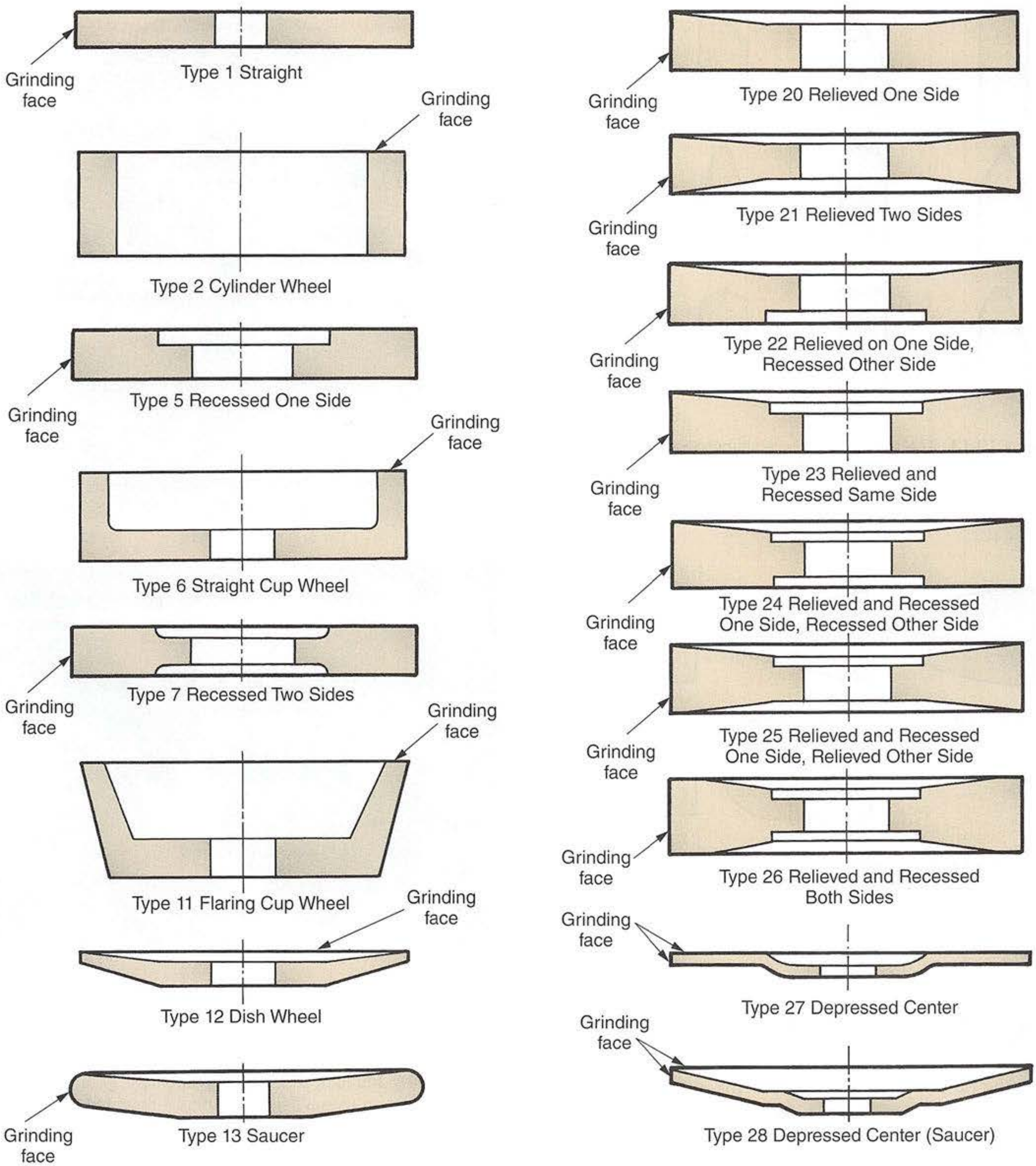
SAFETY NOTE

Avoid standing in line with the grinding wheel, especially during the first few passes across the work.



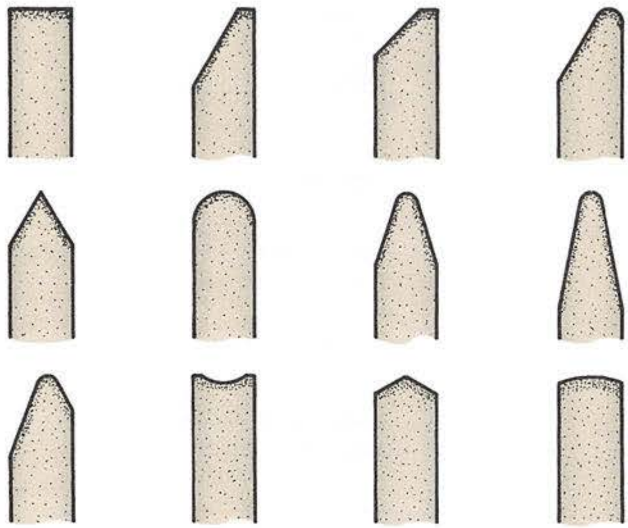
Goodheart-Willcox Publisher

Figure 19-11. Standard system for marking grinding wheels.



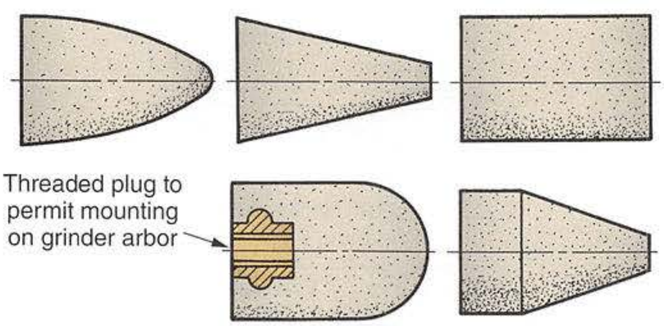
Goodheart-Willcox Publisher

Figure 19-12. Standard grinding wheel shapes.



Goodheart-Willcox Publisher

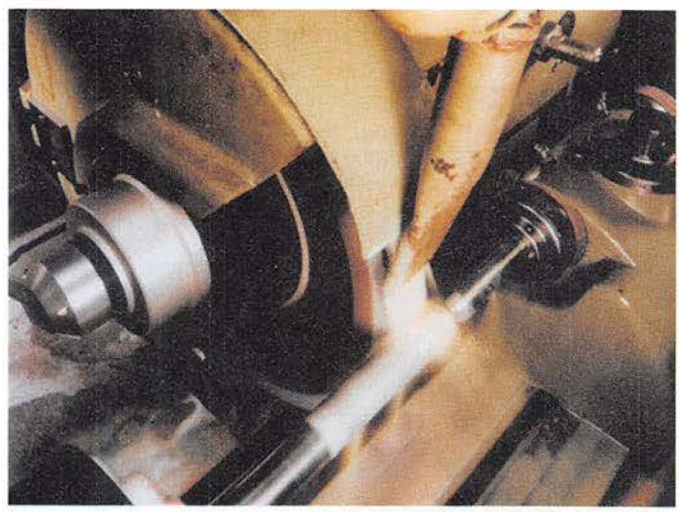
Figure 19-13. The twelve basic face shapes that are generally available.



Goodheart-Willcox Publisher

Figure 19-14. A few of the many grinding wheel shapes available for internal grinding.

A Revolution Tool Company



B Marposs Corp.

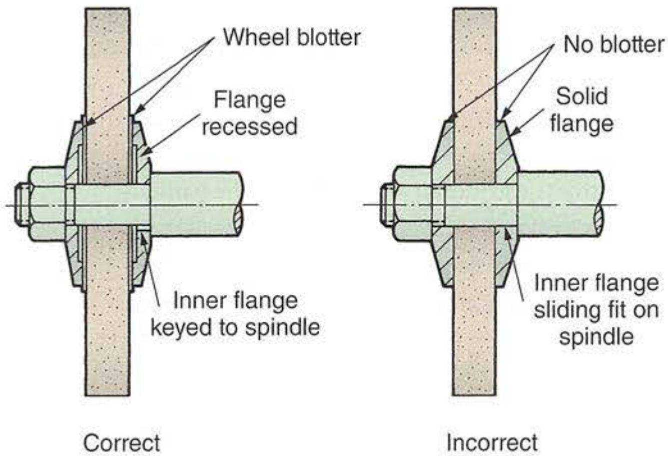
Figure 19-15. Grinding wheel balancing. A—Static balance method. The wheel nut shown features a series of threaded holes on a bolt circle. The wheel is statically balanced by adding or removing setscrews of different lengths opposite the wheel's heavy and light sections. B—Automatic grinding wheel balancing system uses a flange- or spindle-mounted balancing head, vibration and ultrasound sensors, and a microprocessor-based controller to keep the wheel in balance.

19.4 Cutting Fluids

Cutting fluids are an important factor in reducing wear on the grinding wheel. They help maintain accurate dimensions and are important to the quality of the surface finish produced. As a coolant, the cutting fluid must remove the heat generated during the grinding operation. Heat must be removed as fast as it is generated.

Several types of cutting fluids are used in grinding operations:

- Water-soluble chemical fluids take advantage of the excellent cooling ability of water. They are usually transparent and include a rust inhibitor, water softeners, detergents to improve the cleaning ability of the water, and bacteriostats (substances that regulate and control the growth of bacteria).
- Polymers are added to water to improve lubricating qualities.



Goodheart-Willcox Publisher

Figure 19-16. Do not operate a grinder unless the wheel is properly mounted.

- Water-soluble oil fluids are usually a milky white because they consist of a mixture of oil and water. They are less expensive than most chemical fluids. Bacteriostats are added to control bacteria growth.

Coolant can be applied by flooding the grinding area, **Figure 19-17**. The fluid recirculates by means of a pump and holding tank built into the machine. A mist system forces the coolant over the wheel or applies it to the work surface under air pressure. It cools by evaporation. A coolant can also be applied manually by pumping the fluid from a pressure-type oil pump can.

SAFETY NOTE

If coolant is applied manually, keep the tip of the oil pump can a safe distance from the wheel.

For safety, long equipment life, and quality control, a coolant system should be cleaned at regular intervals. To clean the coolant system, remove all dirt and sludge from the holding tank. Chips and grinding wheel residue in the coolant can mar the ground surface of the workpiece. Discard the fluid when it becomes contaminated.

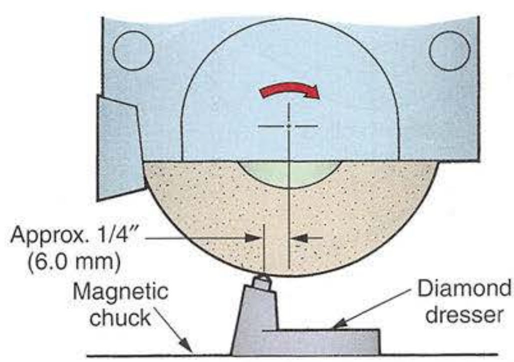
Stana/Shutterstock.com

Figure 19-17. Coolant must flood the area being ground.

19.5 Grinding Applications

The following procedure is recommended to produce a surface that is flat (free of waviness):

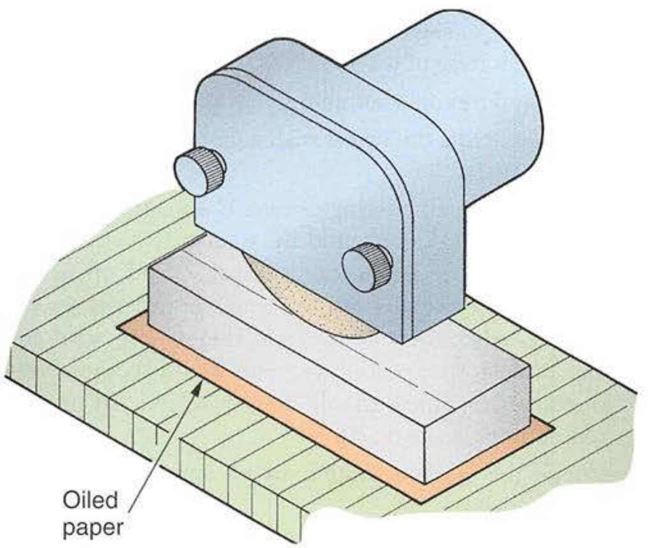
1. Select and mount a suitable grinding wheel.
2. True and dress the wheel with a diamond dressing tool, **Figure 19-18**.
3. Mount the work-holding device. If a magnetic chuck is used, it should be "ground in" to ensure a surface that is true and parallel to table travel. Grind off as little material as possible from the chuck in order to true the surface. Also, be sure to flood the surface with coolant during the procedure. For high-precision work, the magnetic chuck should be ground in each time it is remounted on the machine.
4. Check the coolant system to be sure it is operating satisfactorily.
5. Position the work and energize the chuck. If the work is already ground on one surface, protect it and the chuck surface by fitting a piece of oiled paper between the machined surface and the chuck before energizing the chuck, **Figure 19-19**.
6. Adjust the table stops, **Figure 19-20**.
7. Check the holding power of the magnetic chuck by trying to move the work.
8. Down-feed the grinding wheel until it just touches the highest point on the work surface. The grinding wheel can be set to the approximate position by down-feeding until it just touches a sheet of paper placed between the wheel and the work surface.
9. Turn on the coolant, spindle, and hydraulic pump motors.



Goodheart-Willcox Publisher

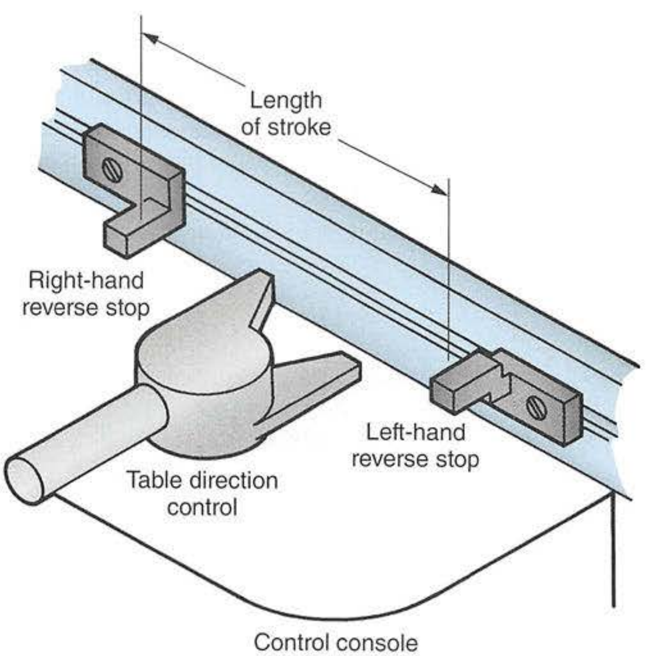
Figure 19-18. For best results when cleaning or truing a grinding wheel, position the diamond wheel dresser as shown.

10. Set the cross-feed to move the table in or out about 0.020″ (0.5 mm) at the end of each cycle.
11. With the wheel clear of the work, down-feed about 0.001″ to 0.003″ (0.025 mm to 0.076 mm) per pass for average roughing cuts.



Goodheart-Willcox Publisher

Figure 19-19. A piece of oiled paper placed between the magnetic chuck and a newly ground surface will protect the finish of the work and the surface of the chuck.



Goodheart-Willcox Publisher

Figure 19-20. Adjustable stops regulate the length of the table stroke. Take care to ensure that the stop adjustment permits the entire work surface to be ground.

12. Use light cuts of 0.0001″ (0.0025 mm) for finishing the surface. It is wise to redress the wheel for finishing cuts.

When the work surface has been ground to the required dimension and finish, use the following procedure to turn off the machine:

1. Move the grinding wheel clear of the work.
2. Turn off table travel.
3. Turn off the coolant.
4. Let the grinding wheel run for a few moments after the coolant has been turned off to remove all traces of fluid. Otherwise, the wheel may absorb some of the coolant and become out of balance.
5. Use a squeegee to remove excess coolant from the work. De-energize the chuck and remove the work.

SAFETY NOTE

Be careful of the sharp edges on newly ground work when removing it from the machine.

6. Clean the machine. Apply a light coating of oil on the chuck's work surface to prevent possible rusting.
7. Place all tools in proper storage.

19.5.1 Grinding Edges Square and Parallel with Face Sides

Most rectangular work requires the edges to be parallel to each other and square with the finished face sides. In one commonly used technique, **Figure 19-21**, edge 1 is ground while being held in a precision vise. After burrs are removed, the adjacent edge 2 is ground. Its squareness is checked vertically with a dial indicator.

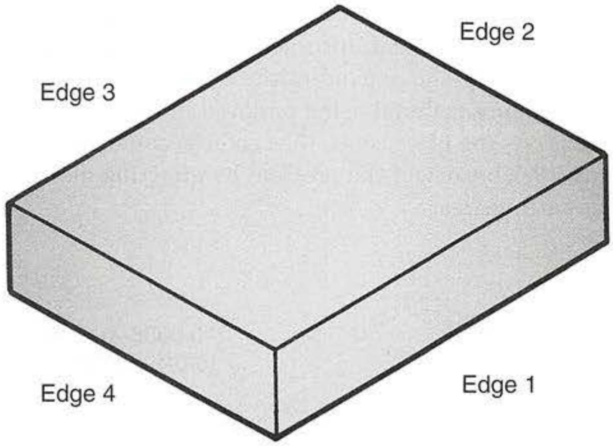
After edges 1 and 2 are ground square with each other, they serve as reference planes to grind edges 3 and 4 to the required dimensions. Use oiled paper in the vise to protect the ground faces and edges from stray metal and abrasive particles. Carefully wipe the vise clean and remove all burrs after grinding each edge.

An angle plate can also be used to grind edges square and parallel with the finished faces. See **Figure 19-22**. A parallel may be used to set the work in approximate position. The same positioning and grinding sequences are followed as previously described.

19.5.2 Creep Grinding

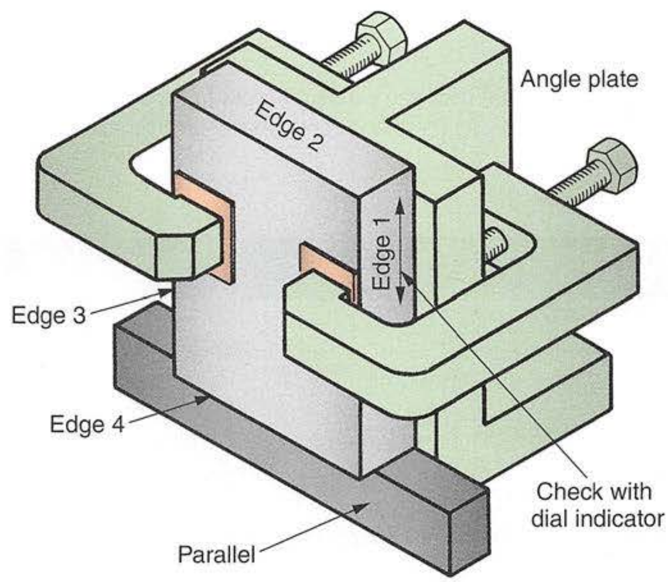
Creep grinding is a production-type surface grinding operation that makes a deep cut into the work, often performed in a single pass, **Figure 19-23**. It is also sometimes known as *deep grinding*. Special grinding machines are required for this type of work.

In comparison to conventional surface grinding, creep grinding increases the depth of cut 1000 to 10,000 times and reduces the work speed in the same proportion. Machining



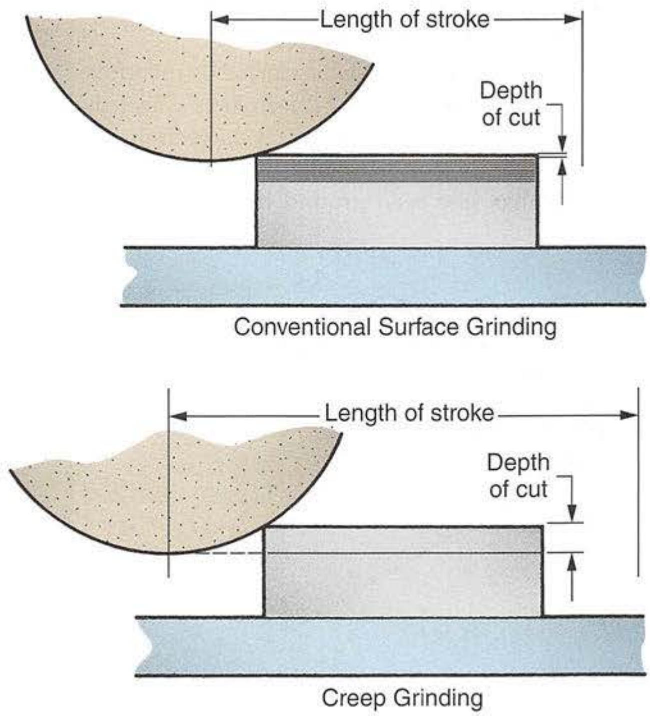
Goodheart-Willcox Publisher

Figure 19-21. The sequence for squaring edges of a rectangular workpiece after the faces have been ground flat and parallel.



Goodheart-Willcox Publisher

Figure 19-22. An angle plate can also be used to hold work for grinding the edges square and parallel.



Goodheart-Willcox Publisher

Figure 19-23. The difference between conventional and creep grinding. Creep grinding equipment must be specially designed for this heavy-duty work.

time can be reduced by 50% to 80%. The tools (such as grinding wheels and work-holding devices) must be designed for this heavy-duty work.

19.6 Grinding Problems

You may encounter several problems specific to precision surface grinding. The following paragraphs address a few of the more common difficulties and provide suggestions for their solution.

Irregular table movement or no table movement on hydraulic machines can be caused by clogged hydraulic lines, insufficient hydraulic fluid, a hydraulic pump that is not functioning properly, or inadequate table lubrication. A cold hydraulic system can also cause these symptoms. Let the machine warm up for at least 15 minutes before use. Air in the hydraulic lines can cause erratic table movement. Make corrections as recommended by the manufacturer of the machine.

Irregular scratches, of no identifiable pattern, are frequently caused by a dirty coolant system, or by particles becoming loosened in the wheel guard. Scratches may also occur if the grinding wheel is too soft, and the abrasive particles are carried to the wheel by the coolant system. Deep, irregular marks on the work surface may be caused by a loose grinding wheel.

Work surface waviness can be caused by a wheel being out of round. This can be corrected by truing the wheel.

Wheel glazing or loading often indicates that the wrong coolant is being used. A dull diamond on the wheel dresser can also cause this problem.

Chatter or vibration marks can be caused by a glazed or loaded grinding wheel. There is a slipping action between the wheel and the work. The wheel cuts until the glazed section comes into position and then slides over the work instead of cutting. Correct the problem by redressing the grinding wheel. Vibration marks can also be caused by a grinding machine that is not mounted solidly or by a wheel that is loose on the spindle. Check for these conditions and make corrections if necessary.

Burning or work surface checking may be the result of too little coolant reaching the work surface, a wheel that is

too hard, or a wheel with grain that is too fine. Make needed corrections as indicated by inspection.

Work that is not flat may be caused by insufficient coolant, a nicked or dirty chuck surface, or a wheel that is too hard. Check and make any necessary corrections.

Work that is not parallel is frequently caused by a chuck that has not been ground in since the last time it was mounted on the machine. A nicked or dirty chuck can cause the same problem. Insufficient coolant can allow the work to heat up and expand in the center of the cut. This results in more material being removed in that area than at each end. As the piece cools, the center becomes depressed, **Figure 19-24**. Correct the problem by directing more fluid to the cutting area.

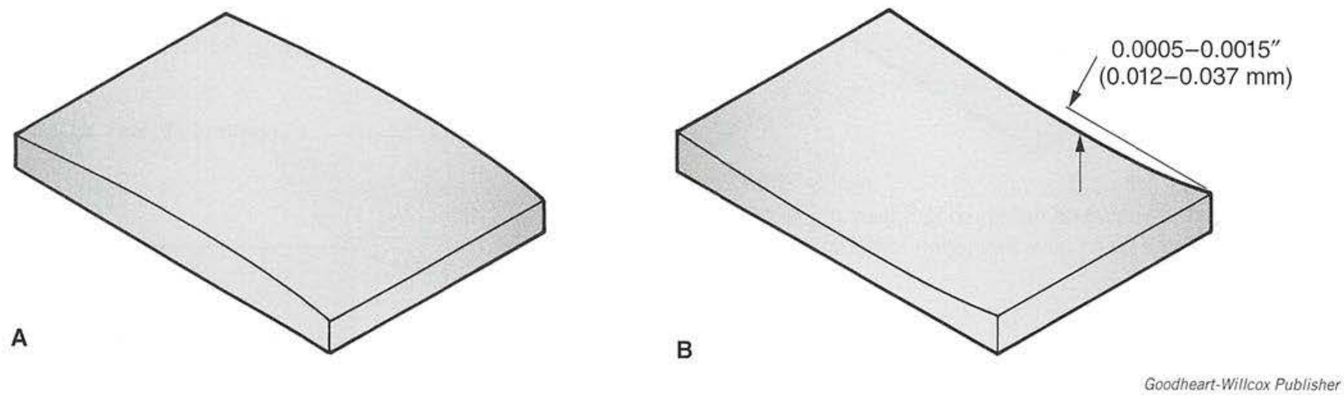


Figure 19-24. Effect of insufficient coolant. A—If not enough coolant is used, frictional heat can cause the work to expand in the center of cut, causing more material to be removed in the center than at each end. B—As the piece cools, its center becomes depressed.

CAREER CONNECTION

Aerospace Engineer

What does an aerospace engineer do?

Aerospace engineers design aircraft, spacecraft, satellites, and other aerospace products. Depending on their organization, they may also coordinate the manufacture and testing of their designs. Aerospace engineers specialize in one of two types of engineering:

- Aeronautical engineering, which focuses on the theory and technology of aircraft and their performance inside the earth's atmosphere.
- Astronautical engineering, which focuses on the theory and technology of spacecraft and their performance inside and outside the earth's atmosphere.

What education and skills are needed to be an aerospace engineer?

Aerospace engineers hold bachelor's degrees in aerospace engineering, engineering, or another science related to aerospace systems. They often specialize in more than one field and use aerospace engineering principles alongside other sciences, such as aerodynamics, celestial mechanics, or thermodynamics. Students interested in aerospace engineering should study chemistry, physics, and mathematics.

What is it like to be an aerospace engineer?

More than a third of aerospace engineers work in the manufacturing industry for aerospace products and parts. They often work in offices but may go elsewhere to test and refine designs. Aerospace engineers earn median wages of $107,000 annually, according to the *Occupational Outlook Handbook*. Those with the highest wages often work for the federal government and companies that manufacture navigational control equipment.

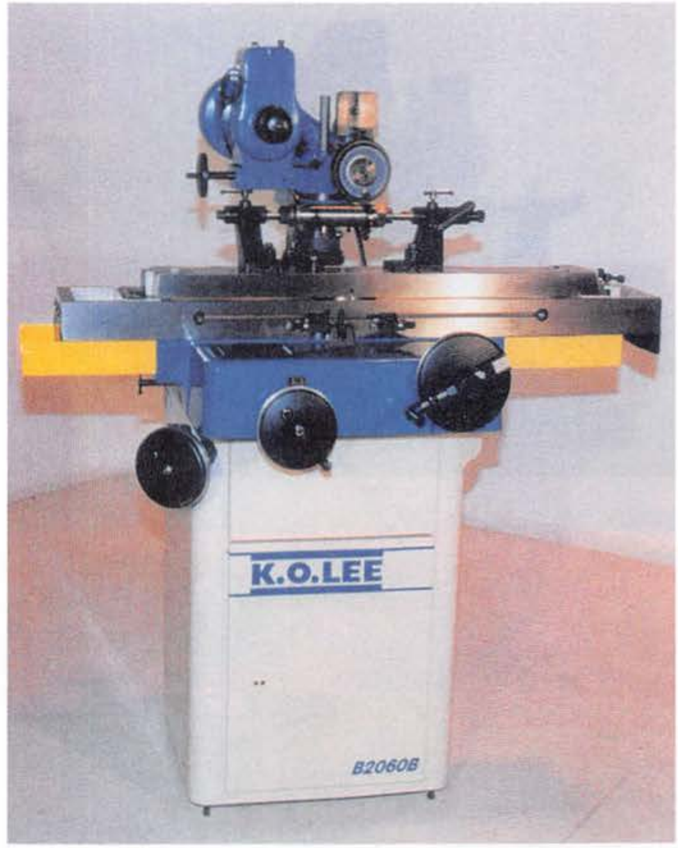
19.7 Grinding Safety

Follow these guidelines to work safely with a surface grinder:

- Never attempt to operate a grinder until you have been instructed in its proper and safe operation. When in doubt, consult your instructor.
- Do not use a grinder unless all guards and safety devices are in place and securely attached.
- Never try to operate grinding machines while your senses are impaired by medication or other substances.
- Always wear proper eye protection when performing any grinding operation.
- Never place a wheel on a grinder before checking it for soundness. Destroy faulty wheels so they cannot be used.
- Check the wheel often and dress it when required to prevent it from becoming glazed or loaded.
- Make sure the grinding wheel is clear of the work before starting the machine.
- Never operate a grinding wheel at a speed higher than specified by the manufacturer.
- Change coolant fluid before it becomes contaminated. It is good practice to set up a schedule for replacing the fluid at regular intervals or adding chemicals to control bacterial growth.
- Have any cuts, burns, or abrasions treated promptly—major infections can result from untended minor injuries.
- Immediately wipe up any spilled coolants from the floor around the machine.
- Stop the machine before making measurements or performing machine and work adjustments.
- If you are using a magnetic chuck, make sure it is holding the work solidly before starting to grind.
- Remove any jewelry, including watch and rings, before using a magnetic chuck to prevent them from becoming magnetized or being pulled toward the machine.
- If you are using automatic feed, run the work through one cycle by hand to be sure there is adequate clearance and that the dogs are adjusted properly.
- Keep all tools clear of the worktable.

19.8 Universal Tool and Cutter Grinder

The ***universal tool and cutter grinder*** is a grinding machine designed to sharpen cutters (primarily milling cutters) to specified tolerances. See **Figure 19-25**. Attachments permit straight, spiral, and helical cutters to be sharpened



K.O. Lee Co.

Figure 19-25. Universal tool and cutter grinder.

accurately. Other attachments are available to adapt the machine for all types of internal and external cylindrical grinding. See **Figure 19-26**.

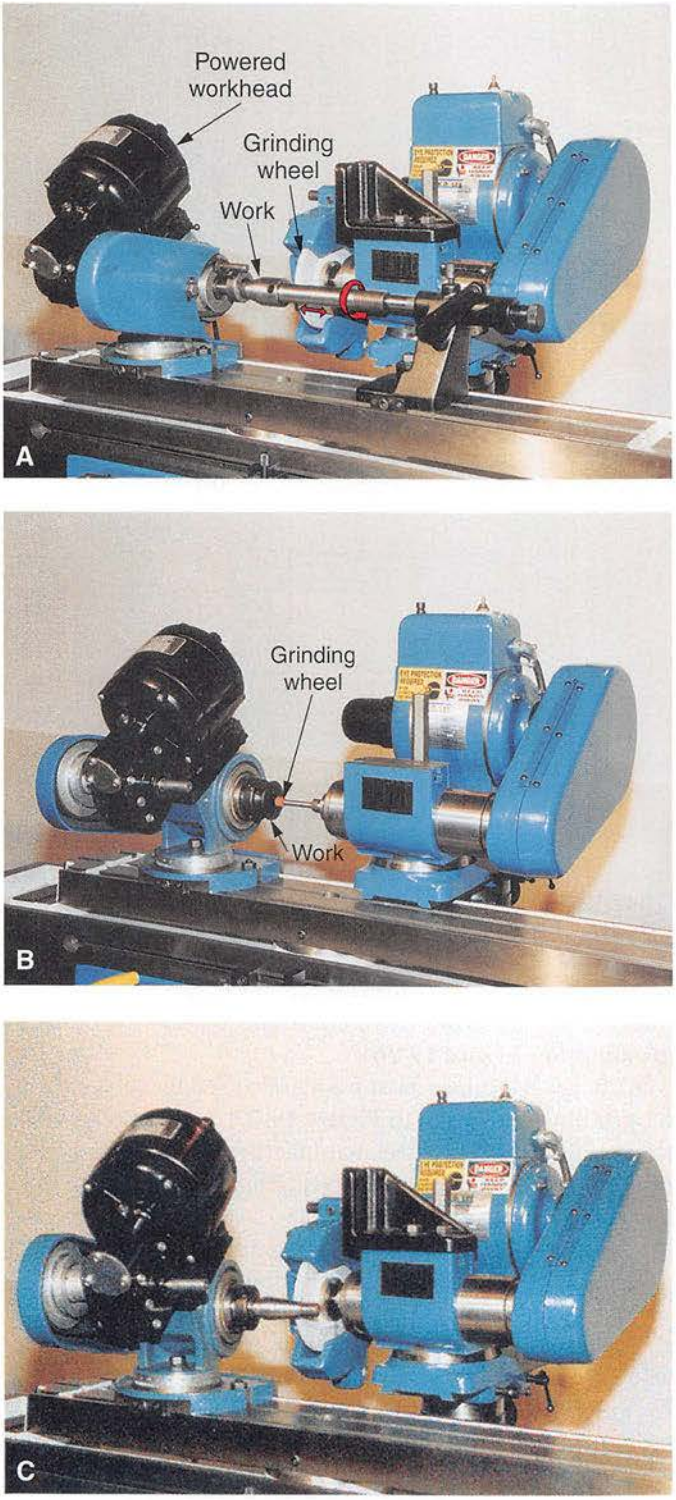
The wheel shapes most frequently used for tool and cutter grinding are shown in **Figure 19-27**. Charts prepared by the various grinding wheel manufacturers are used to determine the correct wheel composition (abrasive, grain size, and bond) for the job at hand.

Keep the grinding wheel clean and sharp by frequent dressing with a diamond tool. Use light cuts to avoid drawing the temper out of the tooth cutting edge.

Crowding the wheel into the cutter is a common mistake when grinding cutters. The cutters are made from materials such as HSS and cemented carbides, which do not give off a brilliant shower of sparks when in contact with a grinding wheel. This creates the illusion that the cut being made is too light.

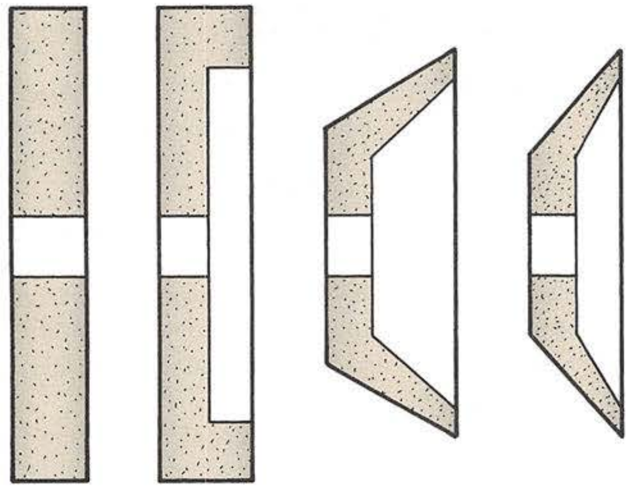
19.9 Sharpening Cutters

A ***tooth rest*** places each tooth of a cutter quickly and accurately into position for sharpening. Several types are used to permit different cutter types to be sharpened. The bracket



K.O. Lee Co.

Figure 19-26. Tool and cutter grinder applications. A—Limited cylindrical grinding can be done on a universal tool and cutter grinder. B—A universal tool and cutter grinder can also be used for internal grinding. C—The center of a spindle is being trued using a universal tool and cutter grinder. The work is rotated by a powered workhead.



Goodheart-Willcox Publisher

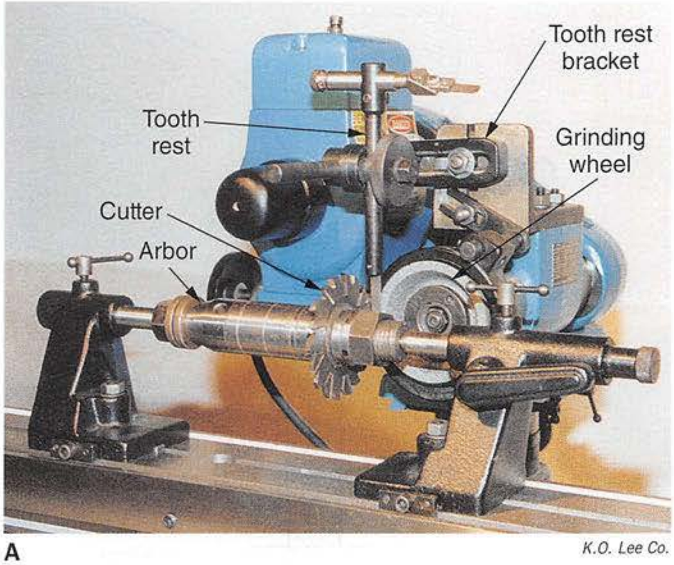
Figure 19-27. Wheel shapes typically used for grinding cutters.

that supports the tooth rest can be mounted to the worktable or on the grinding wheel housing. See **Figure 19-28**.

19.9.1 Grinding Plain Milling Cutters

For this procedure, proper eye protection is a must. If the machine is not equipped with a vacuuming system, you should also wear a respirator mask.

1. Select the correct wheel for the job. True it with a diamond tool.
2. Mount the cutter on a suitable arbor and place the unit between centers.
3. Mount the tooth rest to the wheel head. Position the edge about 1/4″ (6.0 mm) above the centerline of the grinding wheel, **Figure 19-29**. This creates a 5° to 6° clearance angle on the tooth cutting edge of a 6″ (150 mm) diameter cutter. Adjust to suit the cutter being ground.
4. Set up the cutter so that the wheel grinds away from the tooth cutting edge. While requiring more machining care than grinding into the cutting edge of the tooth, this method has less chance of drawing the temper. Also, it prevents the formation of a burr, which would need to be taken off with an oilstone in order to secure a sharp edge. See **Figure 19-30**.
5. If you are using a flare cup wheel, set up the wheel as shown in **Figure 19-31**. Since there is a greater area of contact when using a flare cup wheel, take lighter cuts than with straight grinding wheels.
6. Apply a bit of thinned layout bluing to the back of the tooth. Start the machine and feed the cutter into the wheel. Take a light cut. The bluing will allow a visual check of how the grinding operation is progressing and whether the setup is producing the proper clearance angle.



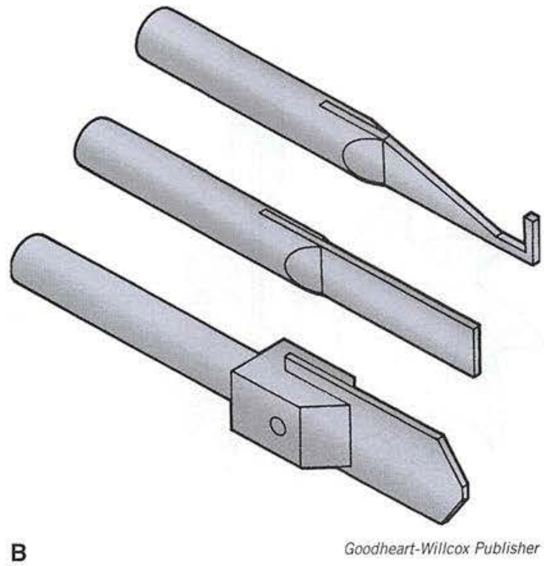


Figure 19-28. Tooth rest. A—A tooth rest is used to position the teeth of a cutter. Note the support bracket and the cup-shaped wheel that does the grinding. B—Several types of tooth rests.

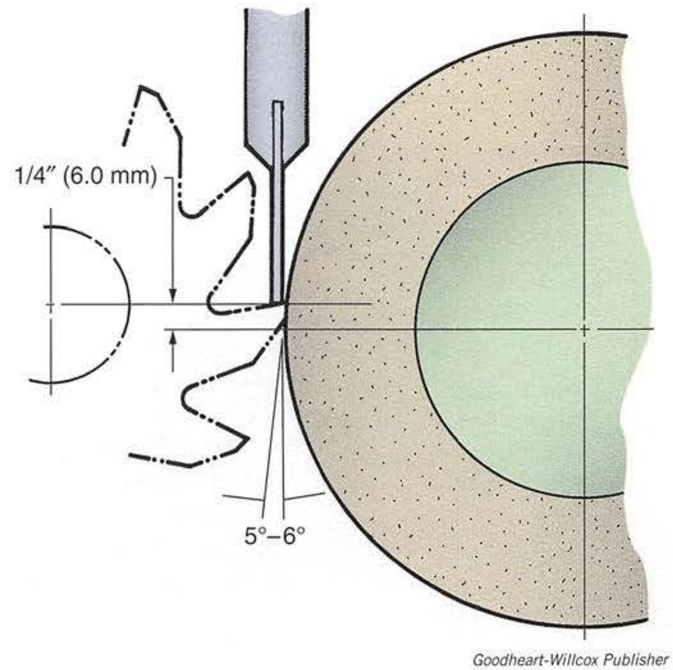


Figure 19-29. This setup is for grinding a 6" (150 mm) diameter cutter.

7. When you are satisfied with the setup, bring the next tooth into position on the tooth rest and grind that tooth.
8. Repeat the operation until all of the teeth are sharpened. Make necessary adjustments to ensure tooth concentricity. The cutting surfaces of all teeth must be the same distance from arbor hole centerline.

After a cutter has been sharpened several times, the clearance angle flat (land) will become too wide. Then it becomes necessary to grind in a secondary clearance angle.

If it becomes apparent that more material is being removed from some teeth than others, make a quick check to determine the cause:

- The grinding wheel may be too soft and wearing down too rapidly. As the wheel wears, less material is removed from the cutter tooth.
- The tooth rest may not be mounted solidly, allowing it to move during the grinding operation.
- The arbor may not be running true on the centers. Check the arbor runout with a dial indicator as the arbor is rotated.

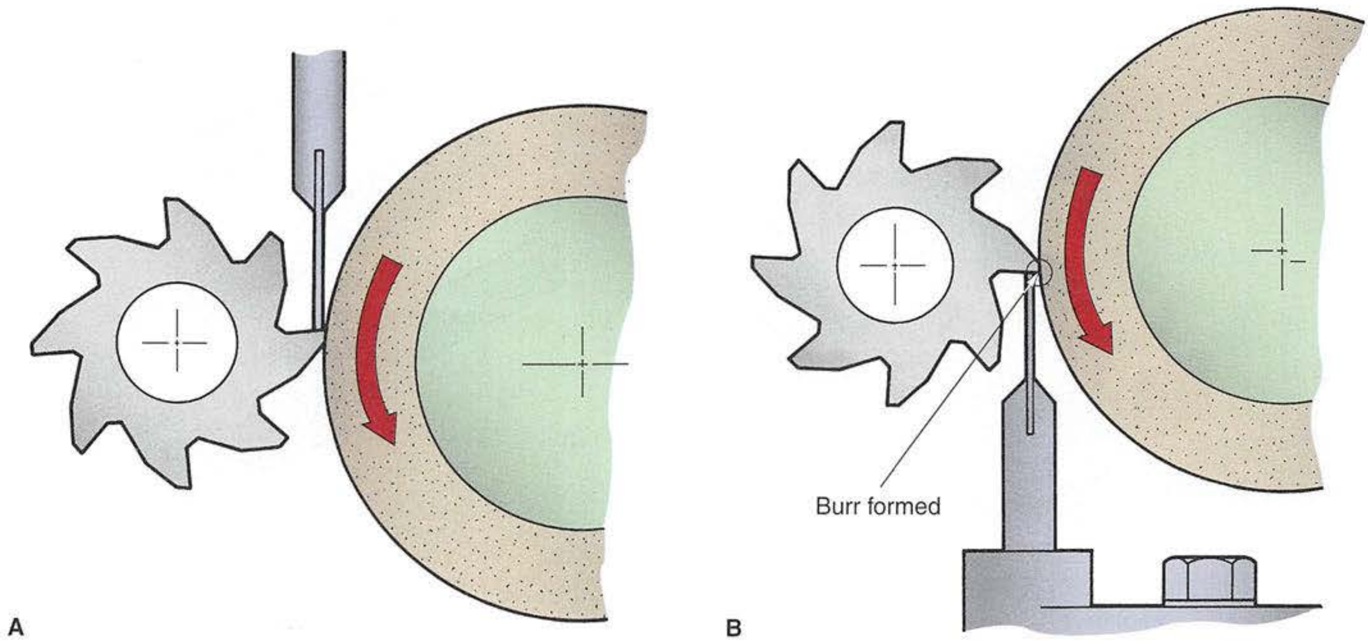
When you have identified the cause of the problem, make the necessary corrections and continue the operation.

An indexing disc may also be used to position each tooth for sharpening, **Figure 19-32**. It is mounted on the arbor. The divisions are normal to each other, plus or minus 4 minutes (1/15°). They are available in a range of graduations.

Teeth on a side-milling cutter must also be sharpened. This is done by mounting the cutter on a stub arbor and fitting the unit into a workhead, rather than positioning it between centers. Facing mills are sharpened in the same manner.

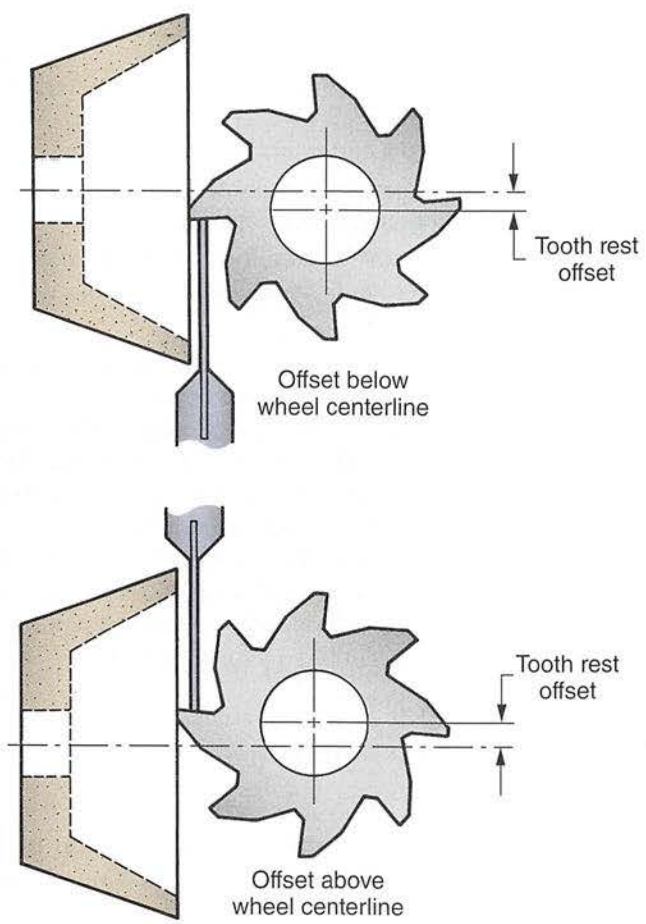
19.9.2 Grinding Cutters with Helical Teeth

Slabbing cutters and other cutters that have helical teeth are sharpened in much the same manner as plain milling



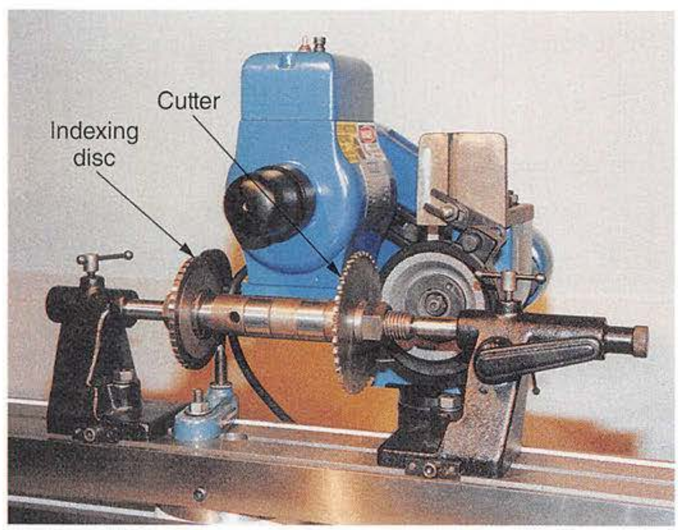
Goodheart-Willcox Publisher

Figure 19-30. Tooth grinding setup. A—The wheel grinds away from the cutting edge of the tooth. With this technique, there is less chance of drawing the temper out of the tooth, and no burr is formed. B—If the wheel grinds into the cutting edge of the tooth, there is a greater chance that a burr will form or the temper will be drawn.



Goodheart-Willcox Publisher

Figure 19-31. Milling cutter being ground with a flare cup wheel.



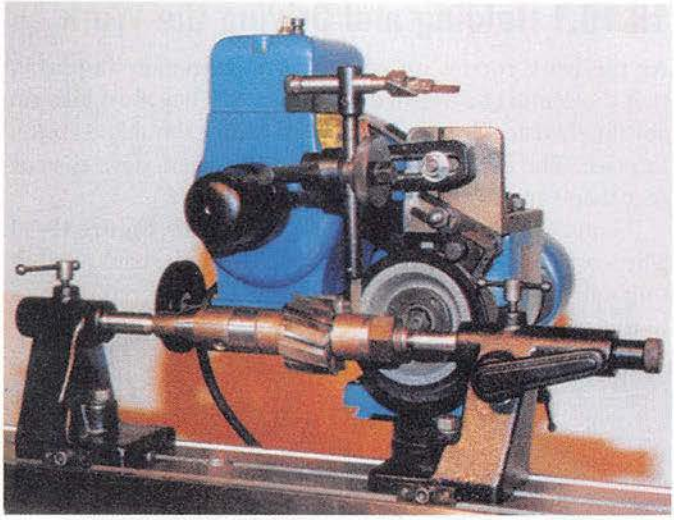
K.O. Lee Co.

Figure 19-32. An indexing disc is often used to position each tooth for sharpening.

cutters, **Figure 19-33**. However, these cutters must be held against the tooth rest as the table is traversed. This will create a twisting motion to keep the tooth correctly located against the grinding wheel.

19.9.3 Grinding End Mills

End mills are sharpened in basically the same way as helical teeth cutters, with the end mill mounted in a workhead rather than between centers. The end teeth are sharpened



K.O. Lee Co.

Figure 19-33. Setup for sharpening a cutter with helical teeth.

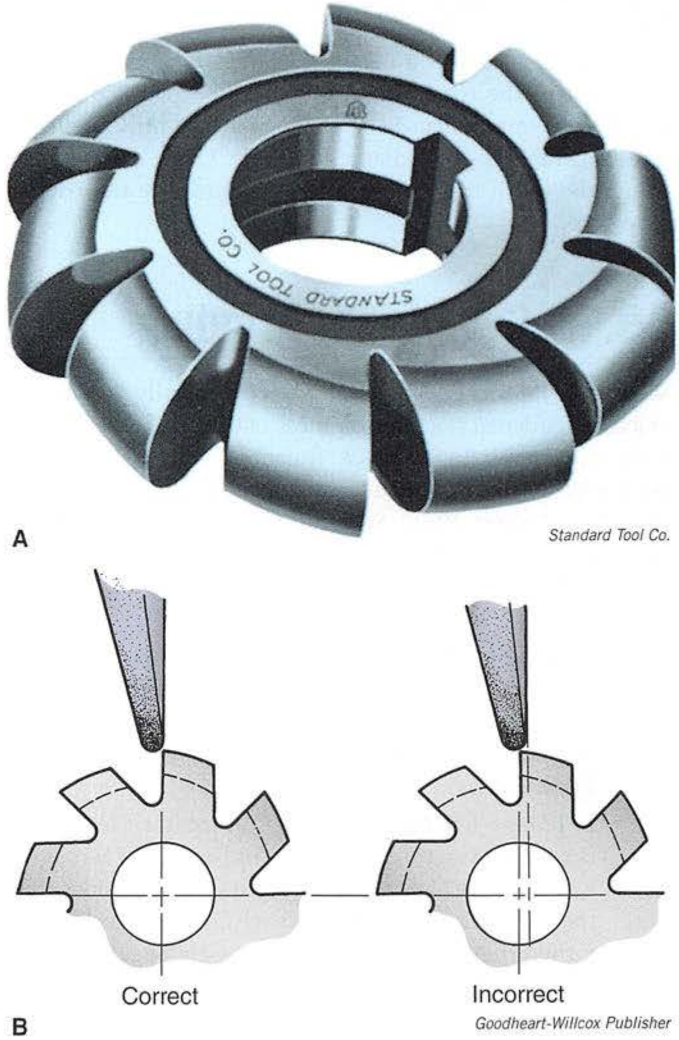
using the same technique used to sharpen the side teeth on a side milling cutter.

19.9.4 Grinding Form Cutters

Form tooth cutters must be ground radially to preserve the tooth shape, **Figure 19-34**. An index disc or a form cutter grinder can be used.

19.9.5 Grinding Taps

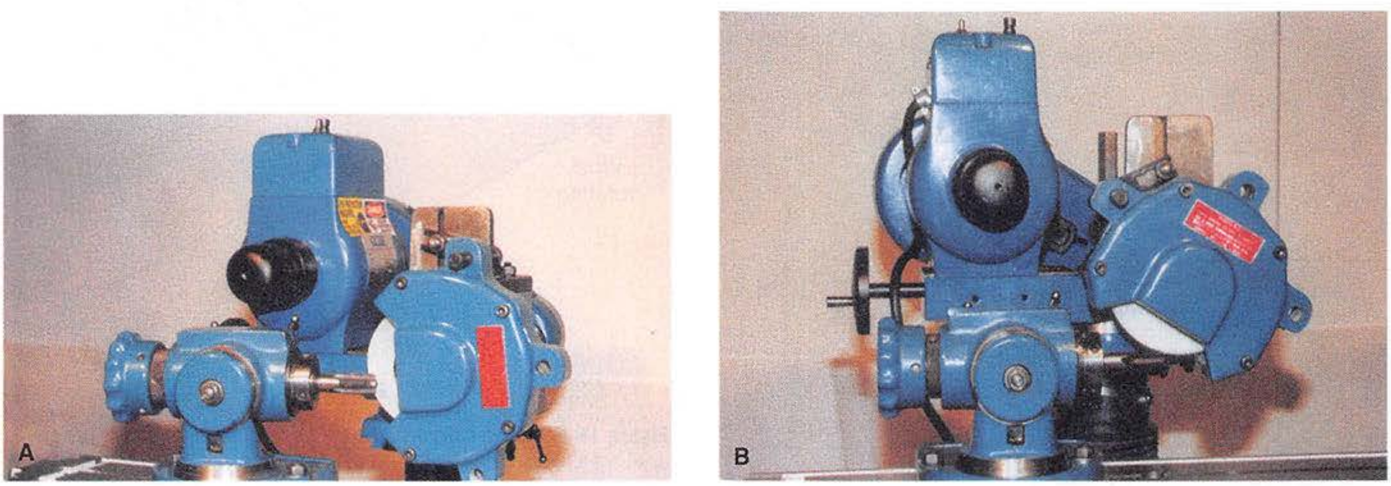
A universal tool and cutter grinder can also be used to resharpen taps. A tap becomes dull when the leading edges of the starting chamfer become worn. The chamfer can be reground by mounting the tap in a workhead, **Figure 19-35**. Flutes are reground using a straight wheel with an edge that has been shaped to fit the flutes.



Standard Tool Co.

Goodheart-Willcox Publisher

Figure 19-34. Form tooth cutters. A—This convex cutter is typical of form tooth cutters. B—Form tooth cutters must be ground radially. Otherwise, the form or shape that the cutter was designed to machine will be altered.



K.O. Lee Co.

Figure 19-35. Grinding taps. A—Setup for regrinding a chamfer on a tap. B—Flutes are being reground on a tap to renew the cutting edges of the teeth.

19.9.6 Grinding Reamers

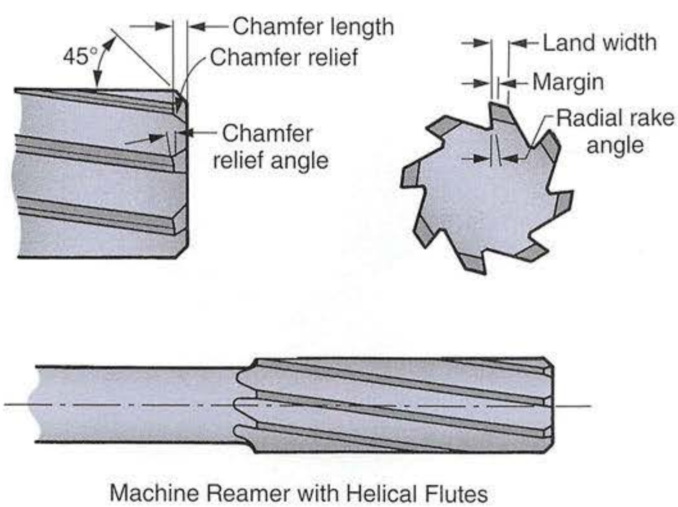
The cutting action of a machine reamer takes place at the front end of the teeth, **Figure 19-36**. Sharpen the reamer in the same manner used to sharpen a face milling cutter. The worktable is pivoted at a 45° angle. Using a cup wheel, adjust the tooth rest and grinding head to give the correct clearance.

19.10 Cylindrical Grinding

With a cylindrical grinder, it is economically feasible to machine hardened steel to tolerances of 0.00001″ (0.0002 mm) with extremely fine surface finishes. Work is mounted between centers and rotates while in contact with the grinding wheel, **Figure 19-37**. Straight, taper, and form grinding operations are possible with this technique. Two variations of cylindrical grinding are traverse grinding and plunge grinding.

- In ***traverse grinding***, a fixed amount of material is removed from the rotating workpiece as it moves past the revolving grinding wheel. Work wider than the face of the grinding wheel can be ground. See **Figure 19-38**.
- In ***plunge grinding***, the work still rotates. However, it is not necessary to move the grinding wheel across the work surface. The area being ground is no wider than the wheel face. Grinding wheel infeed is continuous rather than incremental (minute changes at end of each cut), **Figure 19-39**.

Figure 19-40 shows both techniques being used to grind to a shoulder.



Goodheart-Willcox Publisher

Figure 19-36. Cutting edges of a machine reamer.

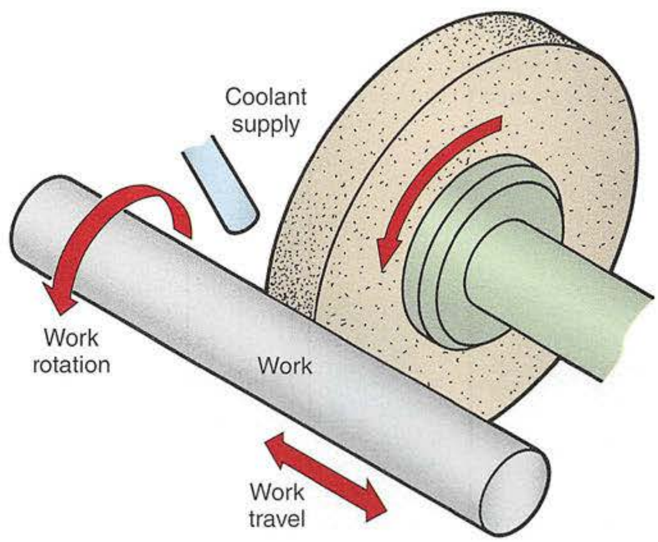
19.10.1 Holding and Driving the Work

As the work rotates on centers, it is extremely important that the centers be free of dirt and nicks. They must also run absolutely true. If possible, the head center should be ground in place. The center holes must also be clean, have the correct shape and depth, and be well-lubricated.

Long work is best supported by work rests, **Figure 19-41**. The work rests support the workpiece from the back and bottom and are adjustable to compensate for material removed in the grinding operation.

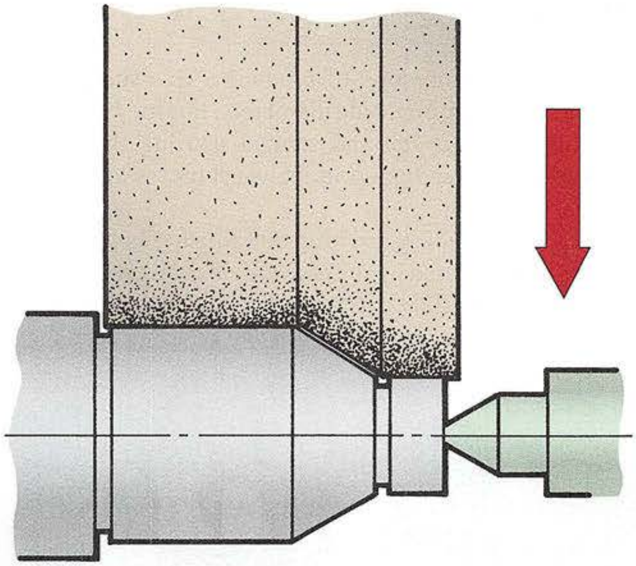
Landis Div. of Western Atlas

Figure 19-37. Close-up of a cylindrical grinding operation.



Goodheart-Willcox Publisher

Figure 19-38. The principle of traverse grinding. The rotating work moves past the rotating grinding wheel.



Goodheart-Willcox Publisher

Figure 19-39. In plunge grinding, the grinding wheel is fed into the rotating work. Because the work is no wider than the grinding wheel, reciprocating motion is not needed.

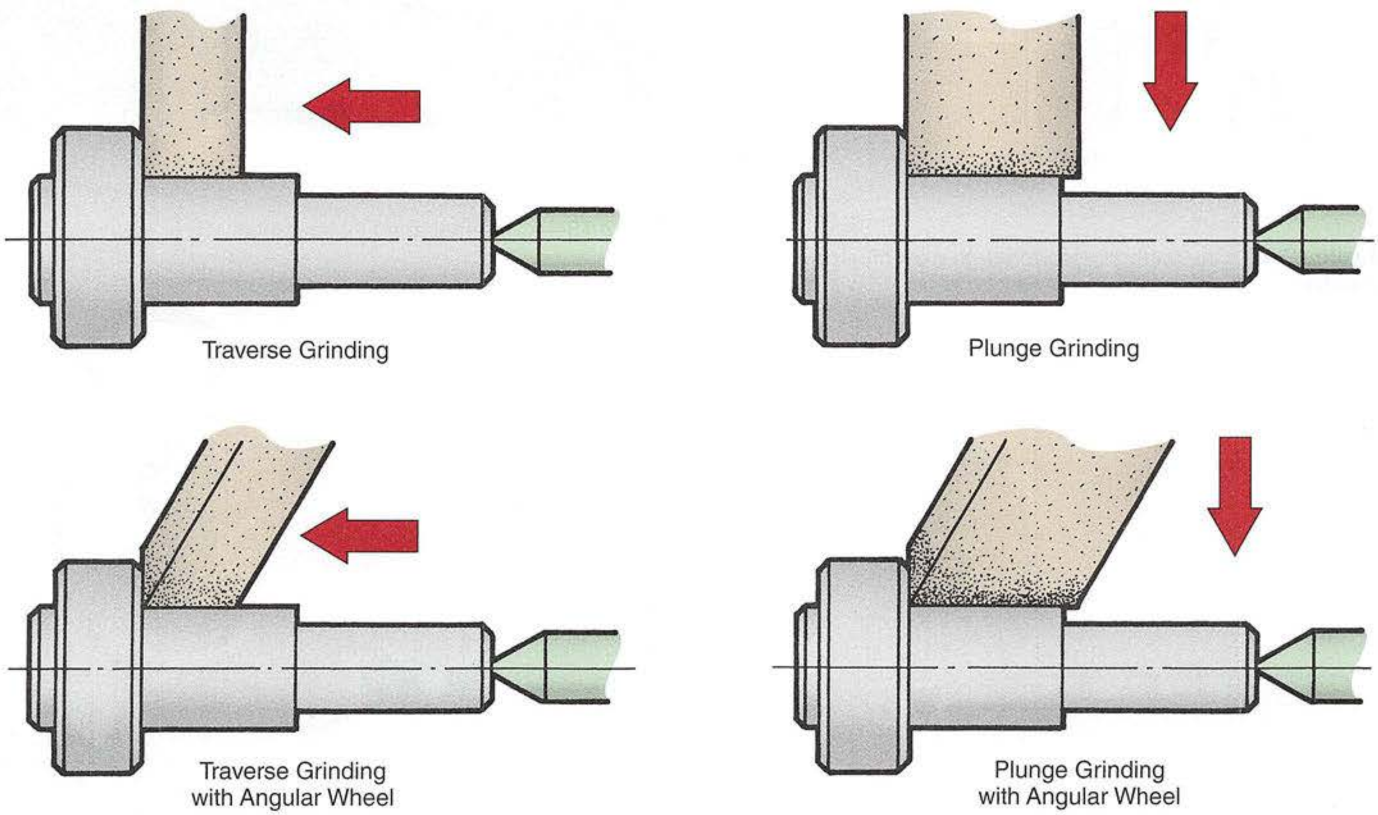
The work can be rotated using a drive plate that revolves around the headstock center and an adjustable drive pin and dog, **Figure 19-42**. Work can also be mounted in a chuck.

19.10.2 Machine Operation

To ensure a good finish and accurate size, it is vital that work rotation and traverse table movement (back and forth in front of the grinding wheel) be smooth and steady. Adjust table movement so that the wheel overruns the work end by about one-third the width of the wheel face, **Figure 19-43**. This permits the grinding wheel to do a more accurate grinding job. Insufficient runoff results in work that is oversize. Complete runoff of the grinding wheel causes the piece to be undersize.

The grinding wheel must be trued and balanced. Otherwise, vibration will cause chatter marks on the work and may cause it to be out-of-round.

Cutting speeds and feeds can be determined from information available on charts furnished by the grinding machine and grinding wheel manufacturers. The charts also specify which coolant will give the best results.



Goodheart-Willcox Publisher

Figure 19-40. Grinding to a shoulder using traverse and plunge grinding techniques.

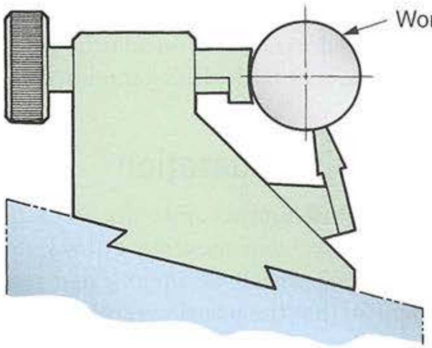


Figure 19-41. Work rests should be placed every four or five diameters along the work for support. They must be adjusted after each grinding pass.

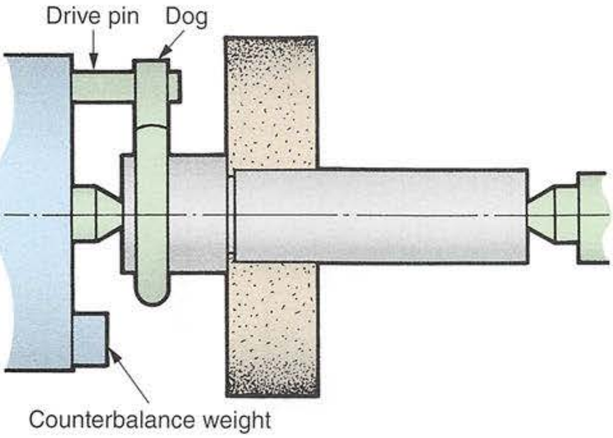


Figure 19-42. One method used to rotate work on a cylindrical grinder.

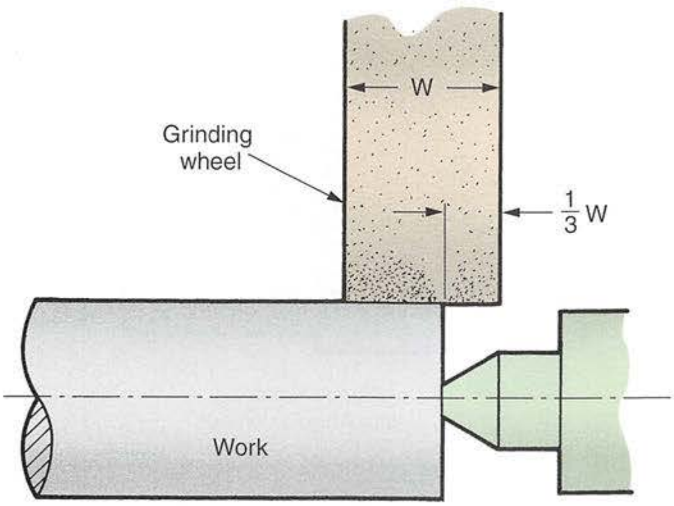


Figure 19-43. Table movement should permit the wheel to extend about one-third of its width beyond the end of the work. This ensures that the wheel removes the proper amount of metal, improving the accuracy of the job.

19.11 Internal Grinding

Internal grinding is done to produce a fine surface finish with high accuracy on inside diameters, **Figure 19-44**. Work is mounted in a chuck and rotated. During the grinding operation, the revolving grinding wheel moves in and out of the hole.

A type of grinding machine that finishes holes in pieces too large to be rotated by the conventional machine is shown in **Figure 19-45**. Hole diameter is controlled by regulating the diameter of the circle in which the grinding head moves.

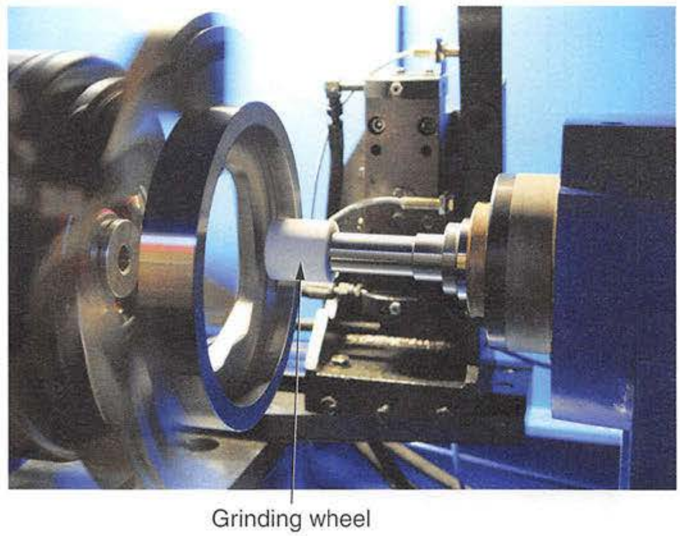


Figure 19-44. Internal grinding operation.

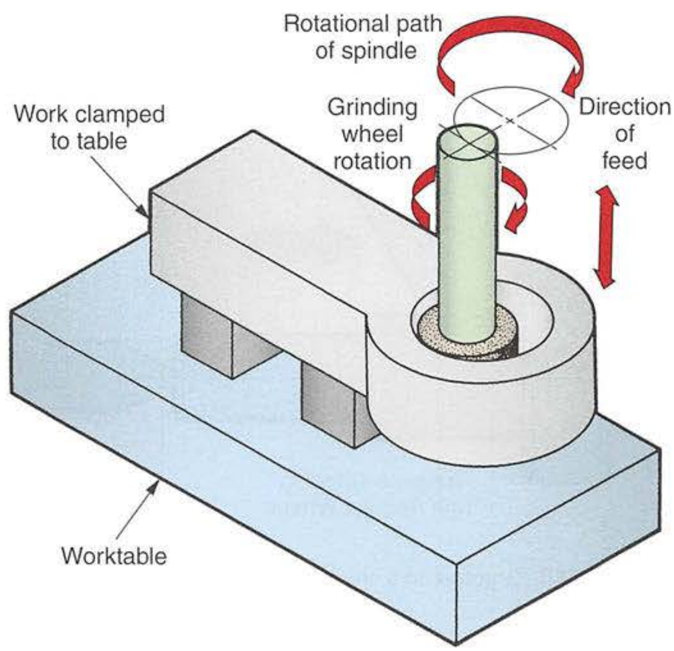


Figure 19-45. Internal grinding technique used when work is too large or odd-shaped to be rotated.

19.12 Centerless Grinding

In *centerless grinding*, the work does not have to be supported between centers because it is rotated against the grinding wheel. Instead, the piece is positioned on a work support blade and is fed automatically between a regulating or feed wheel and a grinding wheel. See **Figure 19-46**. The regulating wheel causes the piece to rotate, and the grinding wheel does the cutting. Feed through the wheels is obtained by setting the regulating wheel at a slight angle.

There are four variations of centerless grinding. See **Figure 19-47**.

- **Through-feed grinding.** This method can be used only to produce simple cylindrical shapes. The work is fed continuously by hand, or from a feed hopper, into the gap between the grinding wheel and the regulating wheel. The finished pieces drop off the work support blade.

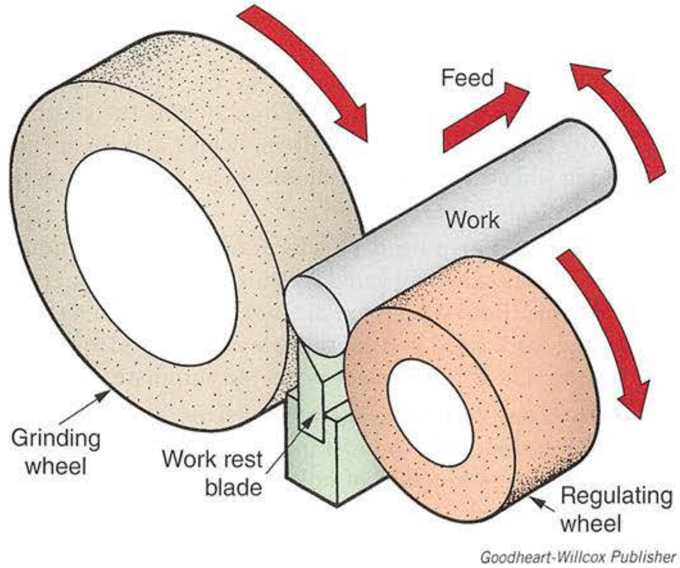


Figure 19-46. Basic arrangement for centerless grinding.

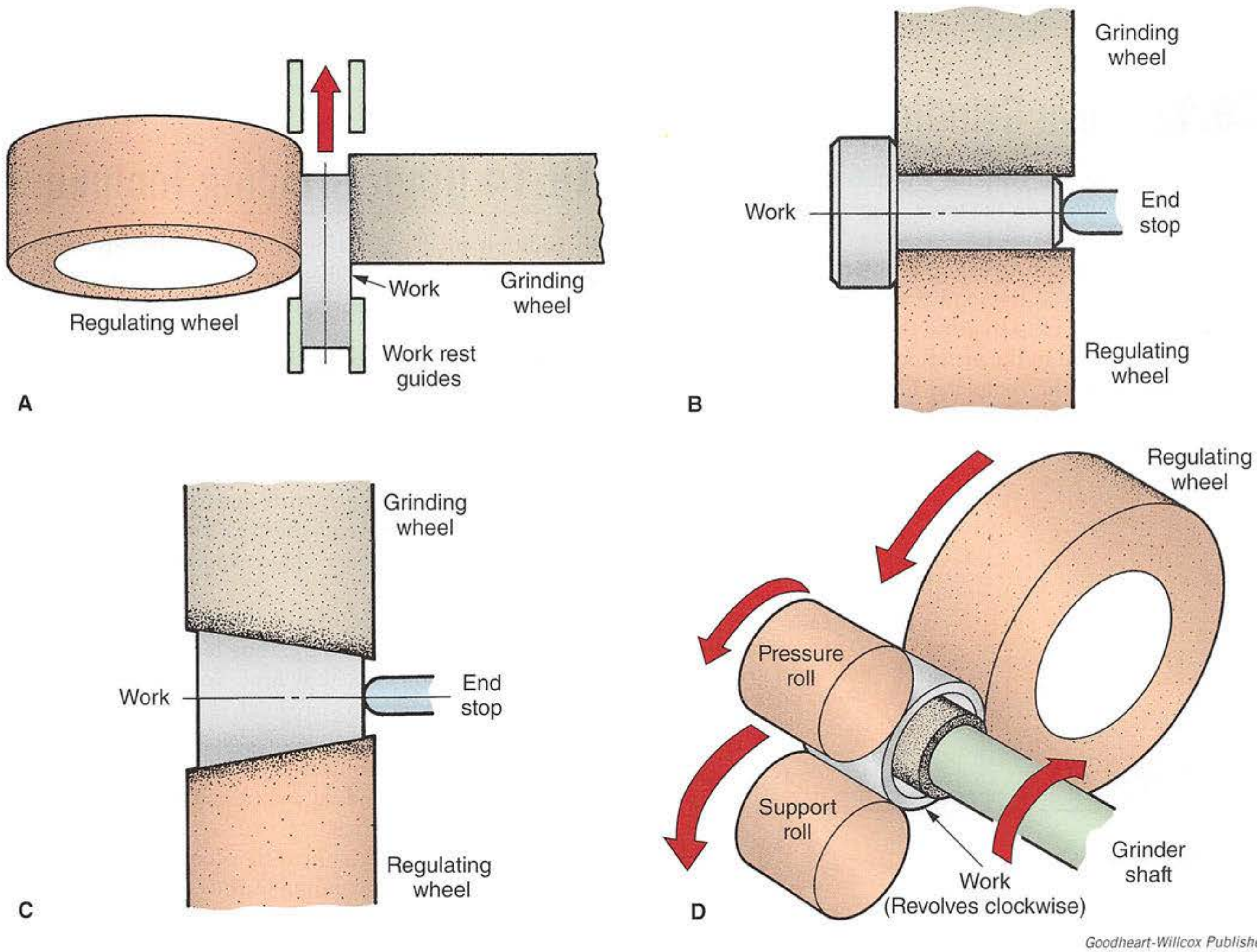


Figure 19-47. Centerless grinding variations. A—Through-feed centerless grinding. The angle of the regulating wheel pulls the work over the grinding wheel. B—Infeed centerless grinding. The work is fed into the wheel gap until it reaches a stop. The piece is ejected when the grinding operation is complete. C—End-feed centerless grinding is best suited for grinding short tapers and spherical shapes. D—Setup for internal centerless grinding.

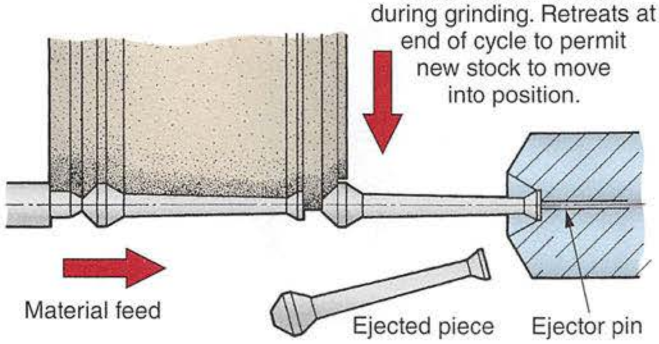
- **Infeed grinding.** This technique feeds the work into the wheel gap until it reaches a stop. The piece is ejected at the completion of the grinding operation. Work diameter is controlled by adjusting the width of the gap between the regulating wheel and the grinding wheel. Work with a shoulder can be ground using this technique.
- **End-feed grinding.** This form of centerless grinding is ideally suited for grinding short tapers and spherical shapes. Both wheels are dressed to the required shape and work is fed in from the side of the wheel to an end stop. The finished piece is ejected automatically.
- **Internal centerless grinding.** This method minimizes distortion in finishing thin-wall work and eliminates reproduction of hole-size errors and waviness in the finish.

Centerless grinding is used when large quantities of the same part are required. Production is high and costs are relatively low, because there is no need to drill center holes or to mount the work in a holding device. Almost any material can be ground by this technique.

19.13 Form Grinding

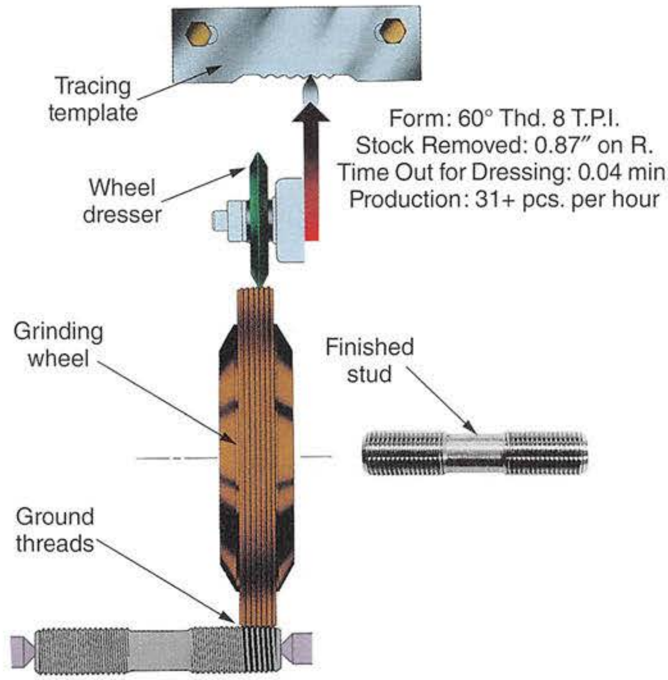
In *form grinding*, the grinding wheel is shaped to produce the required contour on the work. **Figure 19-48** shows this principle.

Thread grinding is an example of form grinding. A form or template guides a diamond dressing wheel that shapes the grinding wheel used to grind the required thread shape. The pattern formed in the grinding wheel is a mirror image of the pattern in the template, **Figure 19-49**. The grinding machine automatically compensates for the material removed from the grinding wheel when it is dressed.



Goodheart-Willcox Publisher

Figure 19-48. Form grinding of this engine part is done at rate of 200 pieces an hour. The material was heat-treated before grinding.



Jones & Lamson Machine Co.

Figure 19-49. Precision threads are being form-ground on a special stud.

19.14 Other Grinding Techniques

In addition to the grinding techniques already described, industry makes considerable use of other abrasive processes. These processes are described in the following paragraphs.

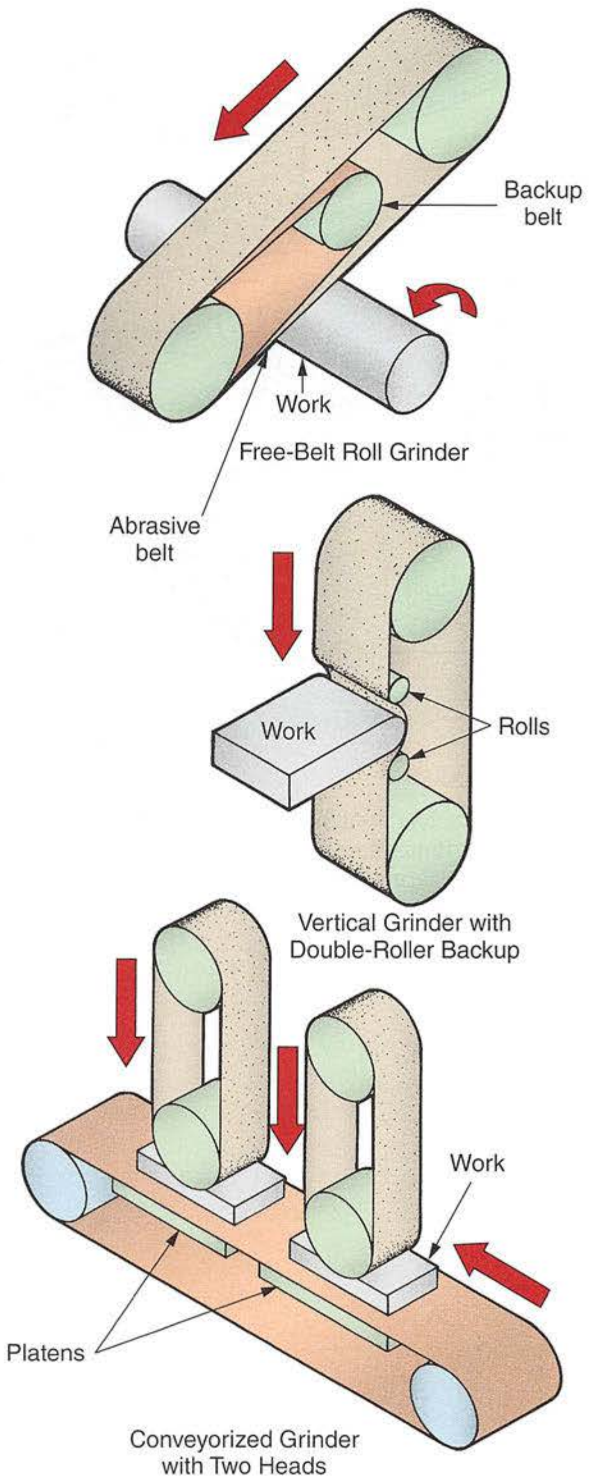
19.14.1 Abrasive Belt Machining

Abrasive belt grinding was first used for light stock removal and polishing operations. However, the technique has advanced so that high-rate metal removal to close tolerances is now possible. This is primarily due to tougher and sharper abrasive grains, improved adhesives, and stronger backings. Several abrasive grinding machine applications are shown in **Figure 19-50**.

Abrasive belts, because of their length, run cool and require light contact pressure, thus reducing the possibility of metal distortion caused by heat. Soft contact wheels and flexible belts conform to irregular shapes. A major advantage of abrasive belt grinding is its versatility. A machine can be converted quickly from heavy stock removal to finishing operations, or for grinding a different material, by simply changing the abrasive belt.

Belts may be used dry or with a coolant. The most satisfactory belt speed for grinding ferrous and nonferrous metals is between 5000 and 9000 surface feet per minute (sfm). Slower speeds of 1500 to 3000 sfm are required for tougher materials such as titanium.

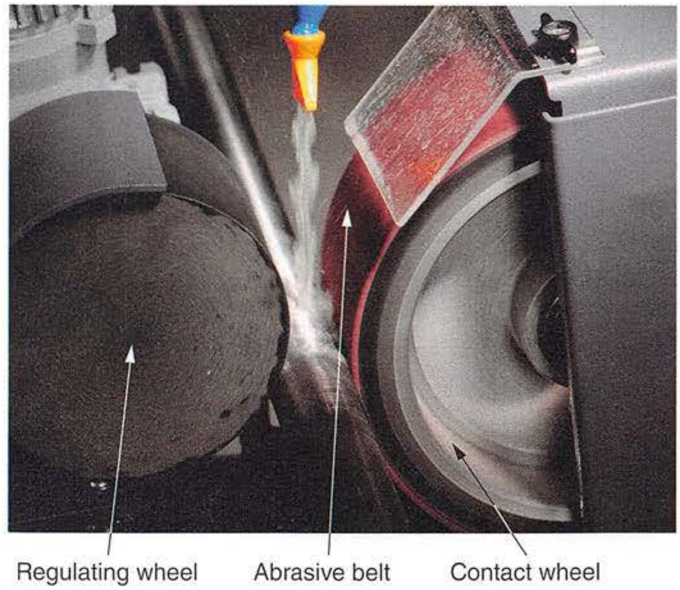
Abrasive belt grinding usually requires support behind the belt, **Figure 19-51**. This may be in the form of contact



Goodheart-Willcox Publisher

Figure 19-50. A few of the many abrasive belt grinding techniques.

wheels or platens. Contact wheels are usually made of cloth or rubber. The hardness and density of the contact wheel affects stock removal and finish. Serrated or slotted wheels improve cutting action and prolong abrasive belt life.



C. & E. FEIN GmbH

Figure 19-51. The contact wheel supports the abrasive belt to prevent it from deflecting when it contacts the work.

Platens are made of metal (some have cemented carbide inserts) and are usually not as effective as contact wheels. They are flat, but can be shaped to conform to the contour required on the work. Jets of air or water may be applied between the belt and platen to reduce friction.

19.14.2 Electrolytic Grinding

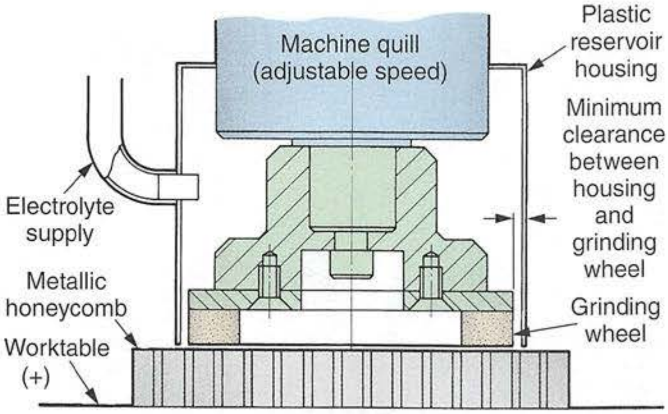
Electrolytic grinding is actually a form of electrochemical machining. Applications of this technique include rapid removal of stock from alloy steel parts, sharpening carbide tools, and machining heat-sensitive work.

An electric current is passed between a metal-bonded grinding wheel (cathode) and the work (anode) through a conductive electrolyte, **Figure 19-52**. The surface of the work is attacked electrochemically and is dissolved in a process similar to electroplating, but in reverse.

The dissolved material is removed by the wheel. No burr is developed, making it possible to machine materials such as stainless steel and exotic metal honeycomb sections. No heat is generated, and there is no metallurgical change in the metal.

19.14.3 Computer-Controlled (CNC) Grinders

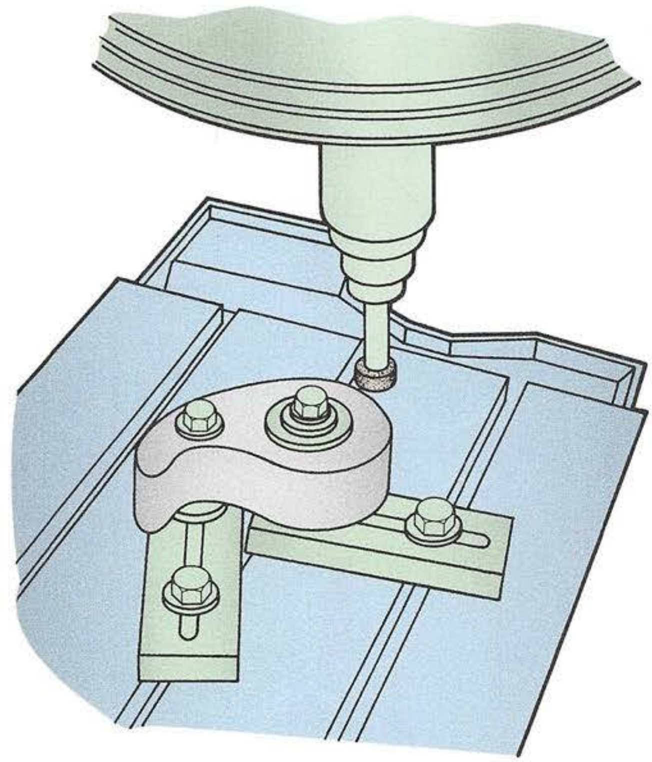
Many types of computer numerical control (CNC) grinders are available. These machines are designed to operate automatically. Functions such as positioning, spindle start and stop, vertical feed motion, and linear feed rates are programmed into the machine's computer. This relieves the operator of the responsibility for controlling and monitoring the numerous coordinate settings and related machine functions.



Goodheart-Willcox Publisher

Figure 19-52. The electrolytic grinding process is actually an electrochemical machining process.

The grinding wheel path representing the contours of the part (at a selected distance from the part edge) is programmed directly to its dimensional specifications. The machine compensates automatically for grinding wheel wear (wheel diameter decreases each time it is dressed). The part's contour is generated in a continuous motion by the X- and Y-axis slides of the machine. See **Figure 19-53.**



Goodheart-Willcox Publisher

Figure 19-53. Contour grinding using CNC to shape a cam. In this illustration, it may appear as though the grinding wheel is moving around the cam edge. In reality, the part moves along its programmed path around the vertical machine spindle holding the rotating wheel.

Chapter Review

Summary

- Precision grinding operates by removing a small amount of material through abrasive action to generate a smooth, accurate surface.
- The two basic types of precision grinders are planer and rotary surface grinders.
- Work-holding devices for surface grinding include magnetic and electromagnetic chucks, parallels and blocks for nonferrous materials, a universal vise, an indexing head with centers, clamps, a precision vise, and double-faced masking tape.
- ANSI Standard B74.13 standardizes grinding wheel marking systems and is used by all grinding wheel manufacturers.
- Cutting fluids used for grinding operations include water-soluble chemical fluids, polymers, and water-soluble oil fluids.
- Selecting the correct abrasive type, grain size, grade, structure, and bond, as well as the correct shape grinding wheel, are all important aspects of proper grinding operations.
- Most problems with precision surface grinding can be identified and corrected by carefully checking the machine settings and the characteristics of the wheel.
- All safety guidelines must be followed to ensure the safety of the operator and any bystanders present in the area.
- Other types of precision grinding operations include cutter sharpening; cylindrical, internal, centerless, and form grinding; and abrasive belt, electrolytic, and CNC grinding.

Review Questions

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

1. Industry classifies surface grinding as the grinding of ______ surfaces.
2. Name the two basic categories of surface grinding machines.
3. Name three work-holding devices used to hold work for surface grinding.
4. The ideal grinding wheel will wear away ______.
   A. as the abrasive particles become dull
   B. at a predetermined rate
   C. slowly to save money
   D. All of the above.
   E. None of the above.
5. A(n) ______ dressing tool is usually used to true and dress wheels for precision grinding.
6. Name the two conditions that commonly prevent a grinding wheel from cutting efficiently.
7. List the five distinguishing characteristics of a grinding wheel.
8. A sound (uncracked) grinding wheel gives off a clear ______ when struck lightly with a metal rod.
9. Why are cutting fluids or coolants necessary for grinding operations?
10. What is the difference between conventional grinding and creep grinding?
11. Irregular scratches on the work are usually caused by ______.
    A. rancid coolant
    B. using the wrong coolant
    C. too little coolant
    D. too much coolant
    E. a dirty coolant system
12. How can the problem in the above question be corrected?
13. Chatter and vibration marks on the work are caused when the grinding wheel is ______.
    A. too hard
    B. too soft
    C. glazed or loaded
    D. out-of-round
    E. loose
14. The problems in the above question can be corrected by ______ the grinding wheel.
15. What should you do if you check a grinding wheel and discover a small crack in the wheel?
16. A(n) ______ is a grinding machine designed to support cutters (usually milling cutters) while they are being sharpened.
17. What is the purpose of a tooth rest?
18. List the two variations of cylindrical grinding.
19. What is the primary purpose of internal grinding?
20. In ______ grinding, it is not necessary to support work between centers or mount work in a chuck while it is being rotated against the grinding wheel.
21. Briefly describe form grinding.
22. The grinding technique that uses a belt on which abrasive particles are bonded for stock removal, finishing, and polishing operations is known as ______.
23. ______ grinding is actually an electrochemical machining process.
24. Describe the electrochemical machining process named in the above question.

CHAPTER 20
Band Machining and Broaching

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Explain the advantages of band machining.
- Select the proper blade for the job to be done.
- Weld a blade and mount it on a band machine.
- Safely operate a band machine.
- Describe power feed attachments and non-sawing applications for band machines.
- Troubleshoot band machining problems.
- Describe broaching operations.
- Explain the advantages of broaching.

Technical Terms

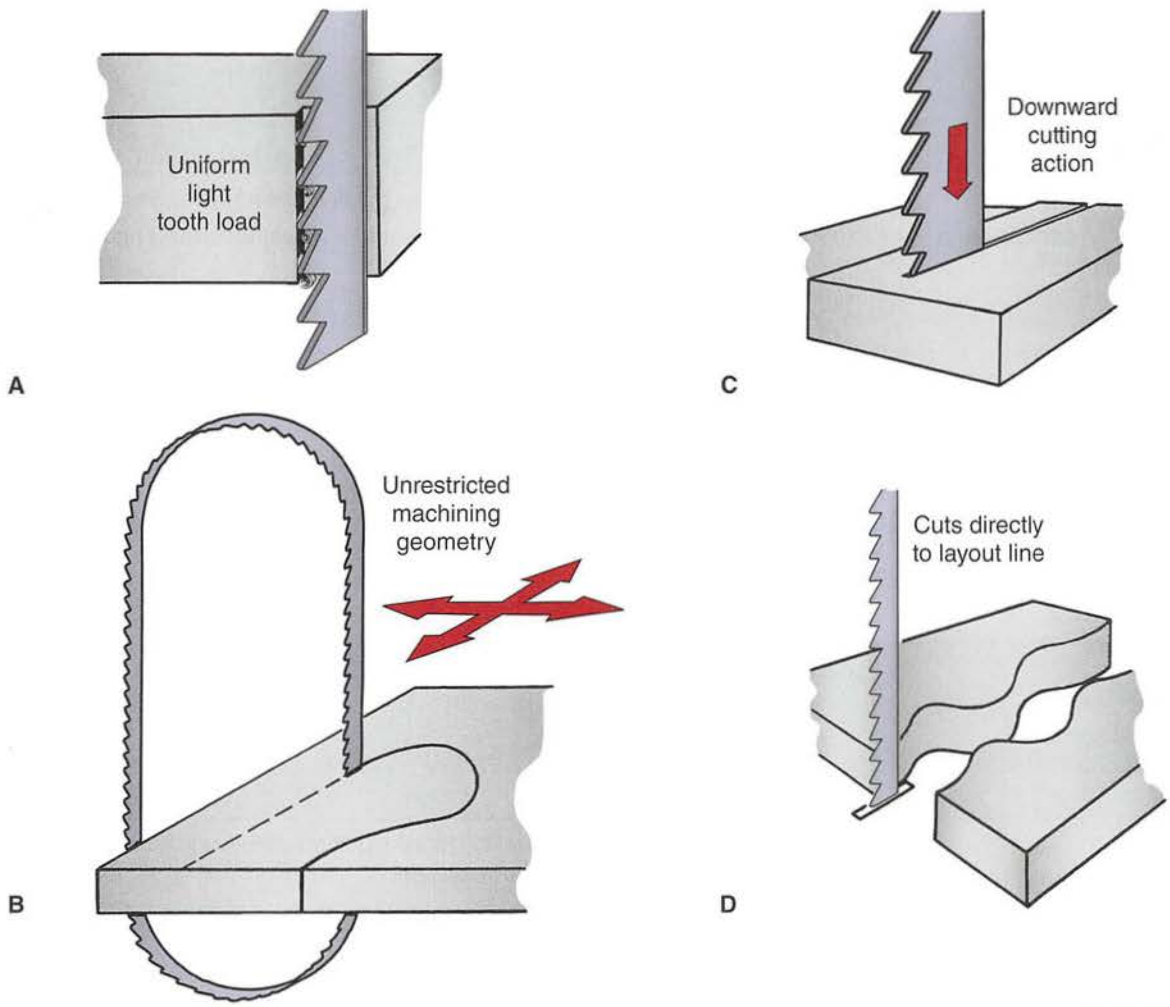
band machining	pot broaching
burnishing	raker set
diamond-edge band	straight set
file band	tooth form
knife-edge blade	wavy set
mist coolant	

Band machining is a machining technique that makes use of a continuous saw blade. Each tooth is a precision cutting tool, so accuracy can be held to close tolerances. This eliminates or minimizes many secondary machining operations. While other technologies—such as wire electrical discharge machines (EDM), waterjet cutting machines, and laser cutting machines—have replaced the band machining process in higher-volume production operations, band machining is still widely used in many manufacturing sectors, especially in areas in which most of the raw materials being machined are wood products.

20.1 Band Machining Advantages

Band machining offers several major advantages over other machining techniques, **Figure 20-1**:

- It maintains sharpness because wear is distributed over many teeth. Chip load is uniform and constant on each tooth, minimizing tool wear.
- It provides unrestricted cutting geometry. Cutting can be done at any angle and in any direction. The length of the cut is unlimited.
- It provides a built-in work-holder. Cutting action is downward, so cutting forces hold the workpiece against the table. In most situations, work need not be clamped.
- It is efficient. Excess chip production wastes power. Band machining produces the desired shape with a minimum of chips. There is little waste because band machining cuts directly to shape, and unwanted material is removed in solid sections.



Goodheart-Willcox Publisher

Figure 20-1. Band machining advantages. A—Wear is distributed over many cutting edges (teeth). B—Band machining permits machining at any angle or direction. Cut length is almost unlimited. C—The downward force of the cutting action helps hold the work on the table. D—Band machining is very efficient and produces little waste. Unwanted material is removed in solid sections.

20.2 Band Blade Selection

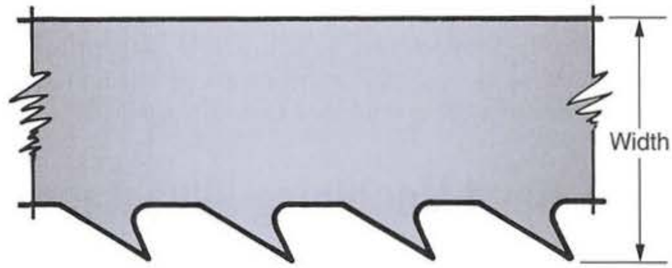
Some blade manufacturers list more than 500 different band saw blades. When selecting the correct blade for a specific job, the machinist must consider the blade type and blade characteristics.

There are six basic band machine blade types. **Figure 20-2** lists them, along with their applications:

- Tungsten carbide.
- Bimetal (high-speed steel [HSS] cutting edge with a flexible carbon steel back).
- High-speed steel.
- Shock resistant high-speed steel.
- Hard edge with spring-tempered back.
- Carbon steel with a flexible back.

Blade characteristics include width, pitch, set, gage, and tooth form.

- **Width.** The wider the blade, the greater its strength and the more accurately it will cut, **Figure 20-3**. For making straight cuts, use the widest blade the machine will accommodate. For contour cutting, use the widest blade that will cut the required radius,



Goodheart-Willcox Publisher

Figure 20-3. Blade width is measured from the tooth tip to the back of the blade.

Match the Tools to the Job		
Type of Blade	**Applications**	**Band Machine**
Tungsten Carbide (T/C) Inserted tungsten carbide teeth on fatigue-resistant blade.	Heavy production and slabbing operations in tough materials.	Horizontal cutoff machines over 5 hp with positive feed. Vertical contour machines over 5 hp with positive feed.
Imperial Bimetal HSS cutting edge with flex-resistant carbon-alloy back.	Mild to tough production and cutoff applications.	Horizontal cutoff machines over 1 1/2 hp with controlled feed, generally with variable-speed drives and with coolant system. Vertical contour machines over 1 1/2 hp with coolant system.
Demon M–2 HSS blade.	Heavy-duty toolroom and maintenance shop work. Full-time production applications.	Horizontal cutoff machines over 1 1/2 hp with controlled feed, generally with variable-speed drives and with coolant system. Vertical contour machines over 1 1/2 hp with coolant system.
Demon Shock-Resistant M–2 HSS blade specially processed for greater shock resistance.	Structurals, tubing, materials of varying cross section.	Horizontal cutoff machines over 1 1/2 hp with controlled feed, generally with variable-speed drives and with coolant system. Vertical contour machines with 1 1/2 hp with coolant system.
Dart Carbon-alloy, hard-edge, spring-tempered back blade.	Superior accuracy for light toolroom and maintenance shop applications as well as light manufacturing.	Horizontal cutoff machines under 1 1/2 hp with coolant system. Vertical contour machines under 1 1/2 hp with coolant system.
Standard Carbon All-purpose, hard-edge, flexible back blade.	Light toolroom and maintenance shop applications.	Horizontal cutoff machines under 1 1/2 hp with weight feed and without coolant, generally with step speeds. Vertical contour machines under 1 1/2 hp without coolant.

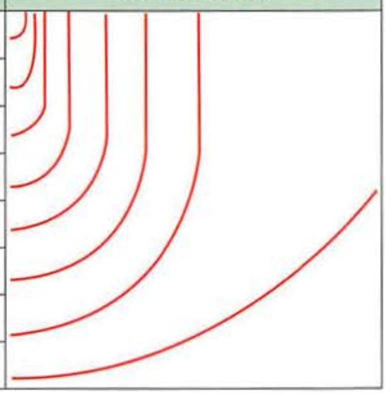
DoALL Co.

Figure 20-2. Recommendations for using basic blade types.

Figure 20-4. Widths from 1/16″ to 2″ (1.5 mm to 50 mm) are available.

- **Pitch.** Refers to the number of teeth per inch or the distance between teeth measured in millimeters. The thickness of the material to be cut determines the proper band pitch to use, **Figure 20-5**. At least three teeth should be in contact with the work at all times for best performance. Blades are available with pitches from 2 to 32 teeth per inch.
- **Set or blade set.** Provides clearance for the blade back, **Figure 20-6**. There are three basic forms, each with a different application:
  - ***Raker set*** is a three-tooth saw set in which one tooth is angled toward the left, the next one straight, and the next one angled toward the right, alternating continuously along the length of the blade. Raker set is recommended for cutting large solids or thick plate and bar stock.
  - ***Wavy set*** is a saw tooth set in which several teeth are angled to the right, followed by several to the left, alternating continuously along the blade. Wavy set should be used for work with varying thicknesses, such as pipe, tubing, and structural materials.
  - ***Straight set*** is a two-tooth saw set in which one tooth is angled to the side and the next one straight, alternating continuously along the length of the blade. Straight set is specified for free-cutting materials, such as aluminum and magnesium.
- **Gage.** Refers to blade thickness, **Figure 20-7**. Heavier gage blades are stronger than thin gage blades.
- **Tooth form.** The term ***tooth form*** refers to the shape of the tooth, **Figure 20-8**. There are three basic tooth forms. Each has a specific application:
  - Standard tooth blades, with well-rounded gullets, are best for most ferrous metals, hard bronze, and brass.
  - Skip-tooth blades provide more gullet and better chip clearance without weakening the blade body. They are recommended for most aluminum, magnesium, and brass alloys.
  - Hook-tooth blades offer two advantages over the skip-tooth blade. The blade's design helps the blade feed more easily and prevents the blade from gumming up.

Width of Blade	Smallest Radius	The width of the blade is determined by the smallest radius to be cut.
1/16	1/16	
3/32	1/8	
1/8	7/32	
3/16	3/8	
1/4	5/8	
5/16	7/8	
3/8	1 1/4	
1/2	3	



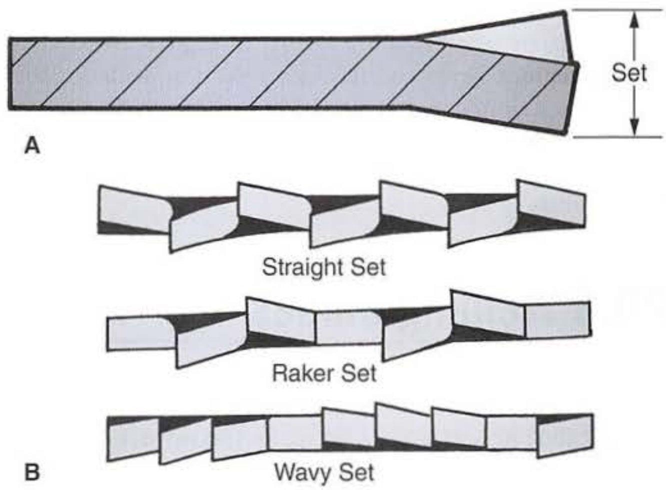
Goodheart-Willcox Publisher

Figure 20-4. Blade width dictates the smallest radius that can be cut.

Material Thickness	Band Pitch
Less than 1″ (25 mm)	10 or 14
1″ to 3″ (25 to 75 mm)	6 or 8
3″ to 6″ (75 to 150 mm)	4 to 6
6″ to 12″ (150 to 300 mm)	2 or 3

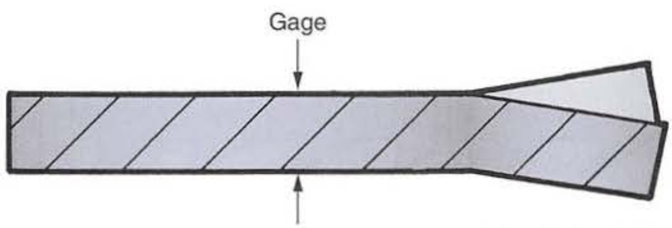
Goodheart-Willcox Publisher

Figure 20-5. Recommended band pitches to saw various material thicknesses.



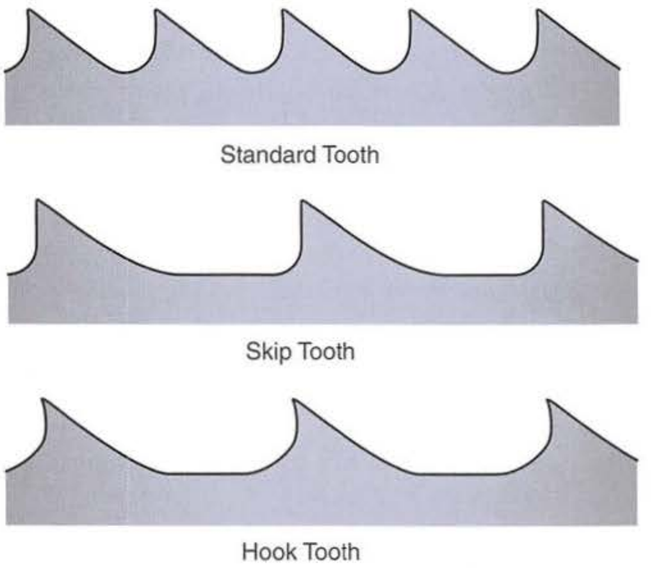
Goodheart-Willcox Publisher

Figure 20-6. Blade set. A—The term *blade set* refers to the side angle of the teeth. B—The different types of blade set.



Goodheart-Willcox Publisher

Figure 20-7. Blade thickness is referred to as *gage.*



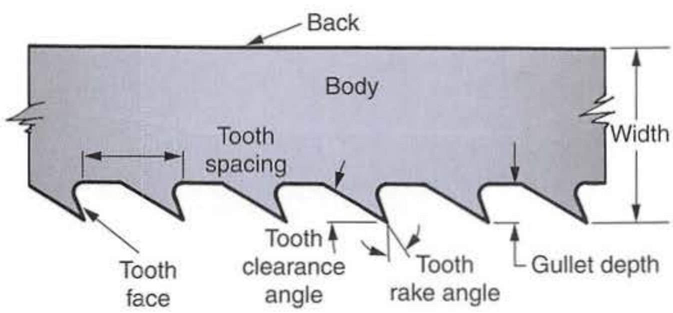
Goodheart-Willcox Publisher

Figure 20-8. Standard tooth blades, with rounded gullets, are usually best for most ferrous materials, hard bronzes, and hard brasses. Skip tooth blades provide for more chip clearance without weakening the blade body. They are recommended for aluminum, magnesium, copper, and soft brasses. Hook tooth blades offer two advantages over skip tooth blades: easier feeding and less gumming up.

The parts of a blade are shown in **Figure 20-9**. Many band machines have built-in blade selection devices. With these, it is a simple matter to dial in the information necessary to determine the best blade for the job. Following this recommendation allows the job to be done faster and with a better finish.

20.3 Welding Blades

Band machine blade stock can be purchased as ready-to-use welded bands. However, it is more economical to buy it in 100′, 300′, or 500′ (30 m, 90 m, or 150 m) strip-out containers. The desired length of blade material is withdrawn from the container, the ends squared, and the blade welded.



Goodheart-Willcox Publisher

Figure 20-9. Saw blade terminology.

Extreme care must be taken to make a good weld (one that is as strong as the blade).

SAFETY NOTE

Always wear leather gloves and approved eye protection when handling band machine blades or blade stock.

20.3.1 Preparing the Blade for Welding

Use snips or blade cutoff shears to trim the blade to length. Cuts in blade stock must be square to avoid the problems shown in **Figure 20-10**.

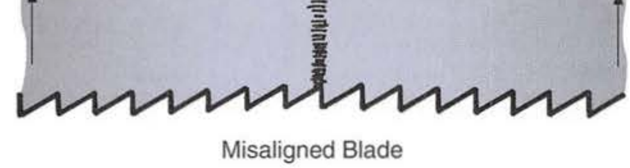
After squaring the blade ends, you will need to remove several teeth (depending on blade pitch) by grinding. Since about 1/4″ (6.5 mm) of the blade is consumed in the welding process, teeth must be ground off to ensure uniform tooth spacing after the weld has been made. Remove only the teeth. Do not grind into the back of the blade.

20.3.2 Making the Blade Weld

For convenience, most band machines have a built-in resistance butt welder. Blades up to 1/2″ (12.5 mm) wide can be welded in a light-duty welder. A heavy-duty resistance, or "flash," butt welder is required for heavier blades. Clean the welder jaws before and after making a weld.

The blade ends are butted together and clamped in the jaws of the welder. Place the saw teeth against the aligning plates. Apply pressure (determined by the width and thickness of the blade being joined) to the band ends. Check the blade to be sure the ends are touching across their entire width and are in the center of the gap between the welder jaws.

After checking, stand to one side to be clear of any flash that might result. Press the welder switch in as far as



Goodheart-Willcox Publisher

Figure 20-10. Problems commonly encountered when welding band machine blades. These problems are usually the result of poorly squared blade ends or dirty blade material.

CAREER CONNECTION

Welder

What does a welder do?

Welders are professionals in construction and manufacturing industries who cut and join metal parts with heat. They are involved in the construction of almost all complex metal structures, including aircraft, bridges, ships, and buildings. The technology and equipment used for each weld varies by industry, material, and the task at hand.

What education and skills are needed to be a welder?

While most employers require only a high school diploma, they prefer to hire workers with formal training through technical or trade schools and community colleges. Several professional organizations offer certification exams for further education.

Students interested in becoming welders should study physics, chemistry, and shop mathematics. Coursework in blueprint reading, mechanical drawing, and computer programming will prepare students for more advanced work like programming industrial welding robots.

What is it like to be a welder?

Specialty trade contractors employ the largest concentration of welding professionals. Depending on the trade, welders may work on one jobsite or many. Welding can be hazardous and physically demanding work, and welders must adapt to the conditions of the jobsite. Observing safety procedures will limit risks of injury.

In recent years, welders and other welding professionals, such as cutters, solderers, and brazers, made median annual wages of $38,100. According to *Occupational Outlook Handbook*, overall pay for these professionals ranges from $25,900 to $60,000 or more per year.

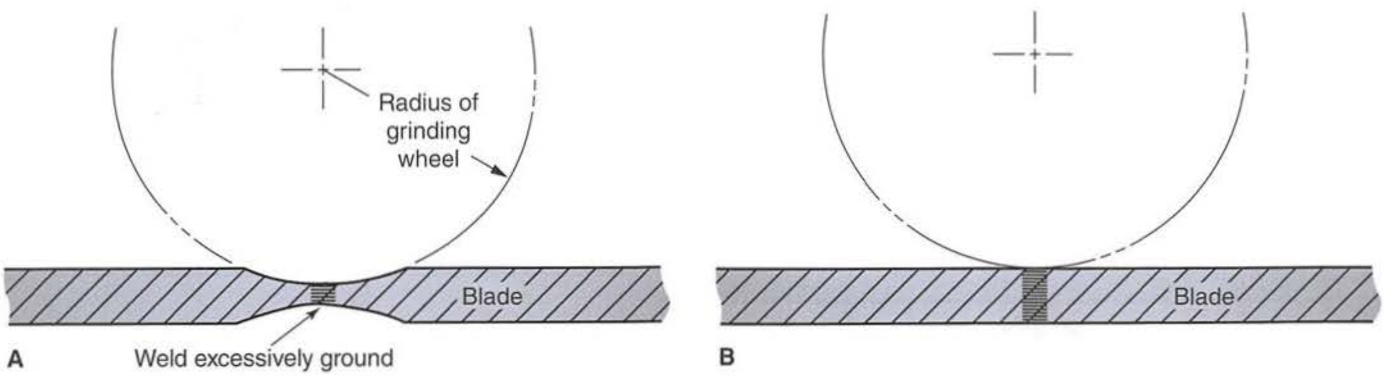
it will go, then release it immediately. The weld will be made automatically.

SAFETY NOTE

Wear approved eye protection with tinted filter lenses when welding band saw blades. The bright flash can cause eye injury. Refer to the manufacturer's instructions to determine the proper filter lens number.

At this point, the weld is brittle. It must be annealed before it can be used. Follow the recommendations for annealing furnished by the manufacturer of the machine being used. Avoid overheating the blade, or it will remain brittle. Let it cool slowly after heating.

Use the grinder built into the welder to remove the flash formed during welding. Use care when grinding to avoid dulling the teeth. The blade will also be weakened if it becomes "dished" during the grinding, **Figure 20-11**.



Goodheart-Willcox Publisher

Figure 20-11. A blade is seriously weakened if it becomes "dished" during the grinding operation that removes weld flash. A—Incorrect grinding. Too much material is being removed from the saw blade, causing it to become dished. B—Correct grinding. Only the flash buildup and surface irregularities around the weld are removed.

20.4 Band Machine Preparation

As with all other machine tools, a band machine must be prepared with care if the tool is to operate at maximum efficiency. Pay attention to lubrication, the blade guides, tracking, and blade tension, as well as cutting speed and cutting fluids.

20.4.1 Band Machine Lubrication

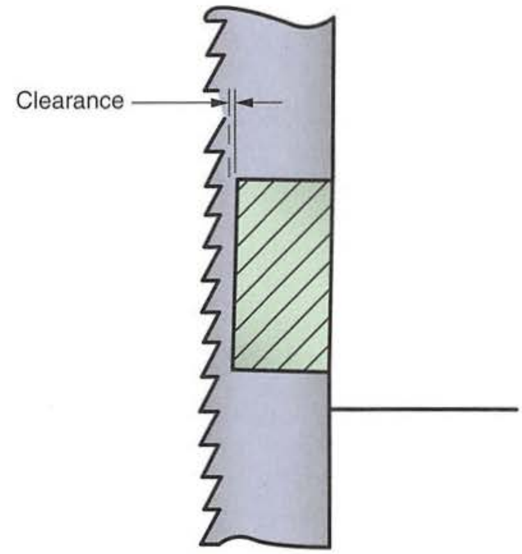
Use the grades of lubricants specified in the manufacturer's manual for the machine. Develop and use a specific lubrication sequence. Following the same sequence each time reduces the possibility of missing a vital point.

20.4.2 Blade Guides

Select and install blade guides suitable for the job at hand, **Figure 20-12**. Use blade guide inserts for light sawing. Roller guides are recommended for continuous, high-speed sawing.

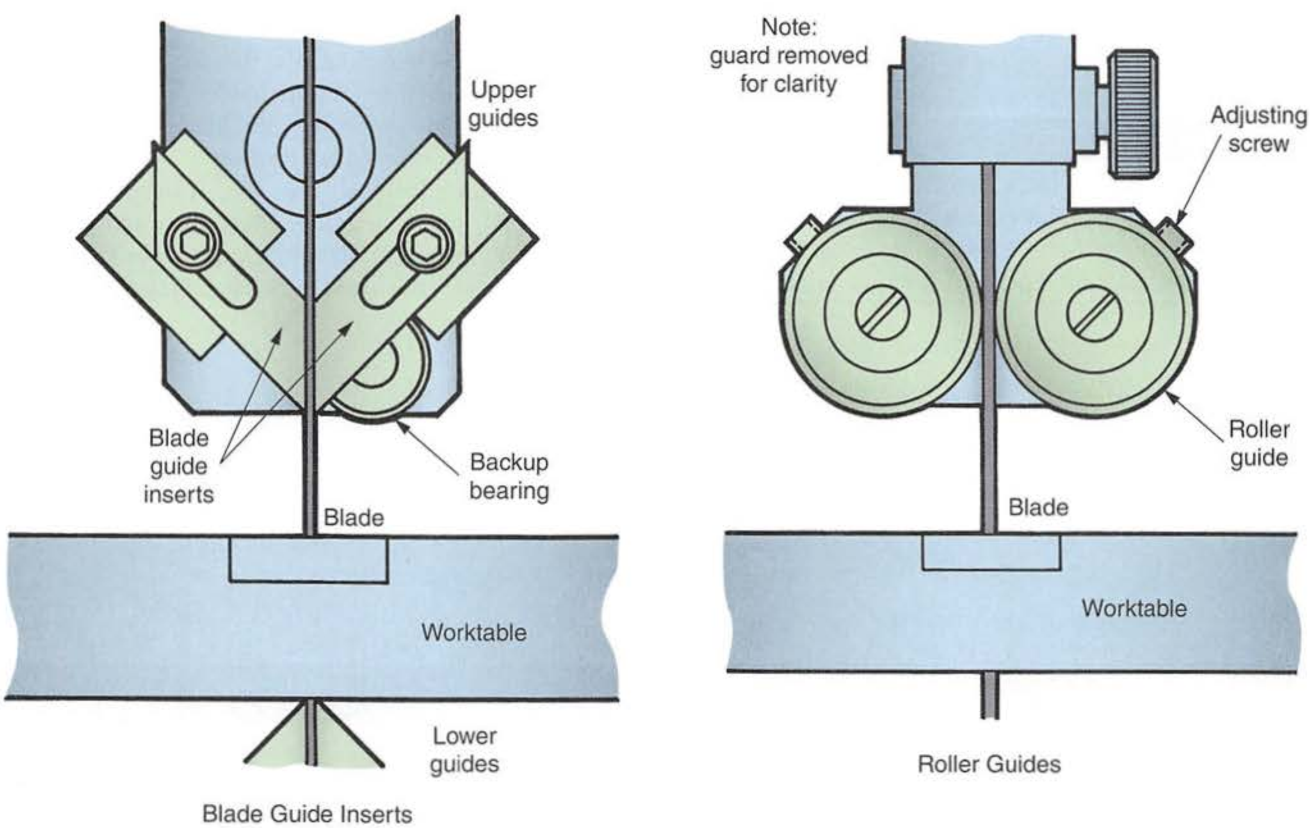
The guides must be the proper width, **Figure 20-13**. If they are too wide, the saw teeth will be damaged. If they are too narrow, the blade will tend to twist in the work, making it difficult to follow the desired cutting path.

Before sawing, inspect and clean the upper and lower blade guides. Make sure the backup bearings are not clogged with chips. Typically, there should be 0.001″ to 0.002″ (0.025 mm to 0.050 mm) clearance between the guide and blade, **Figure 20-14**. For best results, follow the manufacturer's recommendations.



Goodheart-Willcox Publisher

Figure 20-13. Blade guides must be wide enough to prevent the blade from twisting, but narrow enough that they will not damage the teeth.



Goodheart-Willcox Publisher

Figure 20-12. Blade guides.

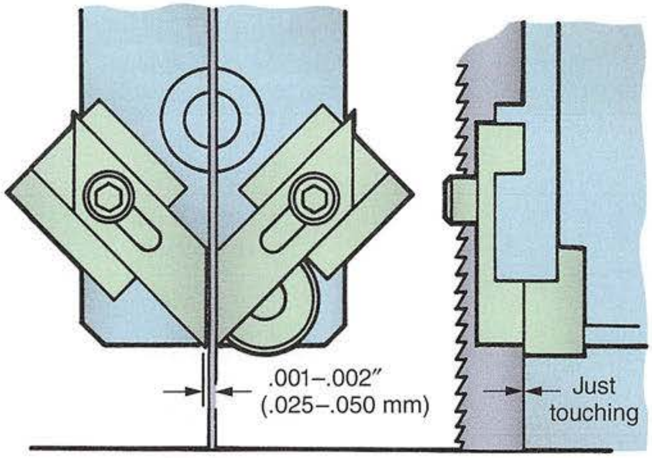


Figure 20-14. Proper blade guide adjustment.

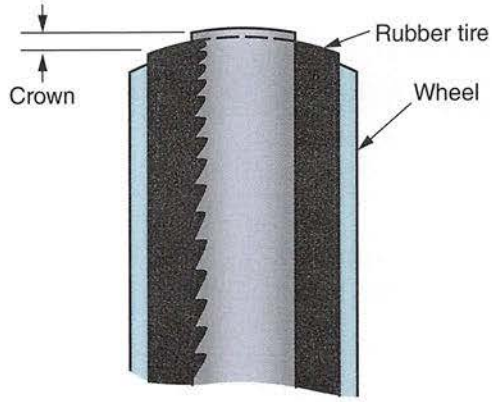


Figure 20-15. The blade should ride directly on the centerline of the wheel's crown.

20.4.3 Blade Tracking

Adjust the band carrier wheels so the blade will track correctly. Again, it is important to follow the manufacturer's recommendations carefully. If the manufacturer's information is not available, observing the following points usually permits satisfactory operation:

- The center of the band should ride directly over the center of the wheel crown on the rubber tire, **Figure 20-15.** Replace the tire if it becomes frayed or damaged.
- There should be no noticeable gap between the back of the band and the backup bearings of the saw guides.
- The blade must be installed with the teeth downward.

SAFETY NOTE

Exercise extreme caution when handling saw blades. They are sharp and can cause serious injury. Be sure that power to the band machine has been turned off at the master switch before attempting to install a blade.

20.4.4 Band Blade Tension

Blade tension refers to the pressure put on the saw band to keep it taut and tracking properly. On smaller machines, tension is usually applied by means of a hand crank, **Figure 20-16.** On large, heavy-duty machines, tension is applied hydraulically. The amount of blade tension required is determined by the width and pitch of the blade. Use the band tension chart furnished with the machine.

Many band machines have built-in tension meters that make it easy to adjust and maintain proper blade tension. This is especially important when a new blade is installed. A new blade has a tendency to stretch slightly when first used. This can create a safety problem if tension is not readjusted before it falls off too far.

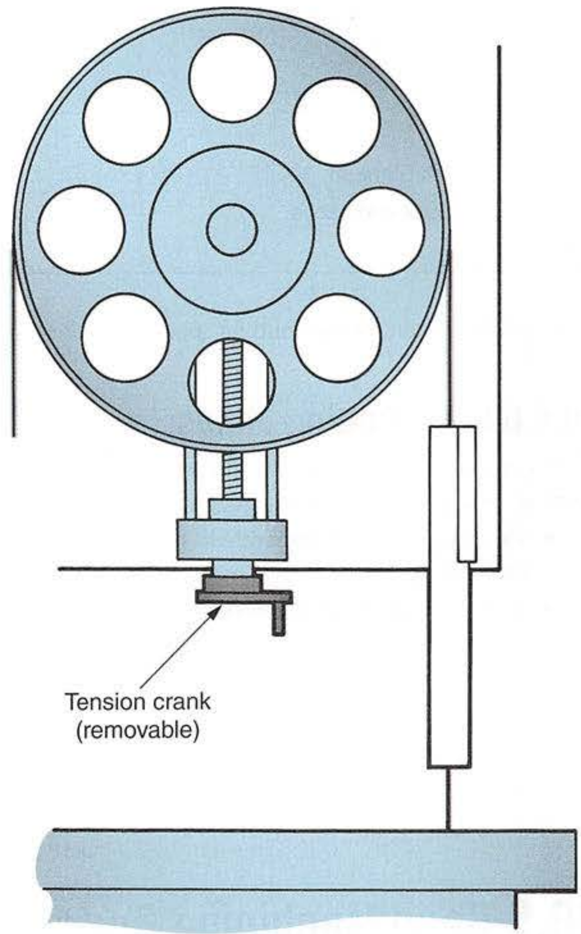


Figure 20-16. Tension-adjusting hand crank.

20.4.5 Band Cutting Speed

As with other machining operations, best results are obtained if recommended cutting speeds are maintained. See **Figure 20-17** for the recommended band speeds for a few selected materials.

Material	Thickness Inches (millimeters)	Band Speed Surface Feet per Minute (meters per minute)
Aluminum alloys	—	360+ sfm (110+) mpm
Low-to-medium carbon steels	Under 1″ (25 mm)	345–360 sfm (105–110) mpm
	1″–6″ (25 mm–150 mm)	295–345 sfm (90–105) mpm
Medium-to-high carbon steels	Under 1″ (25 mm)	225–250 sfm (70–75) mpm
	1″–6″ (25 mm–150 mm)	200–225 sfm (60–70) mpm
Free machining steels	Under 1″ (25 mm)	260–395 sfm (80–120) mpm
	1″–6″ (25 mm–150 mm)	260–345 sfm (80–105) mpm
Titanium, pure and alloys	Under 1″ (25 mm)	100–115 sfm (30–35) mpm
	1″–6″ (25 mm–150 mm)	90–110 sfm (30–35) mpm

Goodheart-Willcox Publisher

Figure 20-17. Recommended cutting speeds for selected metals and alloys.

20.4.6 Band Cutting Fluids

Cutting fluid can be applied on a band machine by flooding, misting, or applying a solid lubricant.

- Flooding is recommended for heavy-duty band machining.
- ***Mist coolant*** is used for high-speed sawing of free-machining nonferrous metals. It is also used for tough, hard-to-machine materials.
- Solid lubricants are applied when the machine does not have a built-in coolant system.

If requested, coolant manufacturers will furnish a coolant chart with recommendations for band machining operations.

20.5 Band Machining Operations

Vertical band machines, like the one shown in **Figure 20-18**, are designed to perform several sawing operations. These include straight, contour, and angle sawing, as well as internal cuts.

20.5.1 Straight Sawing

Straight, two-dimensional sawing is band machining in its simplest form, **Figure 20-19**. The operator just follows a straight layout line. Slitting, slotting, and notching can also be done rapidly on a band machine.

SAFETY NOTE

Exercise extreme care when the blade breaks through the work at the completion of a cut. If possible, a piece of backup metal, called a *push block*, should be between your hand and the point where the band will break through the work. The sharp moving blade can cause serious injury!

20.5.2 Contour Sawing

Contour sawing is possible on a vertical band machine. Machine size is the limiting factor on the work dimensions that can be cut.

20.5.3 Angular Sawing

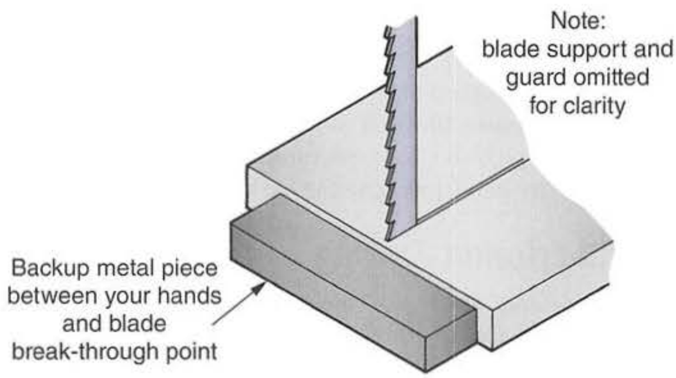
The table on a vertical band machine is usually mounted on trunnions, permitting it to be tilted. This makes it possible to machine compound angular cuts.

20.5.4 Internal Cuts

Precision internal cuts can be made on a vertical band machine. The band is threaded through a hole drilled in the

DoALL Co.

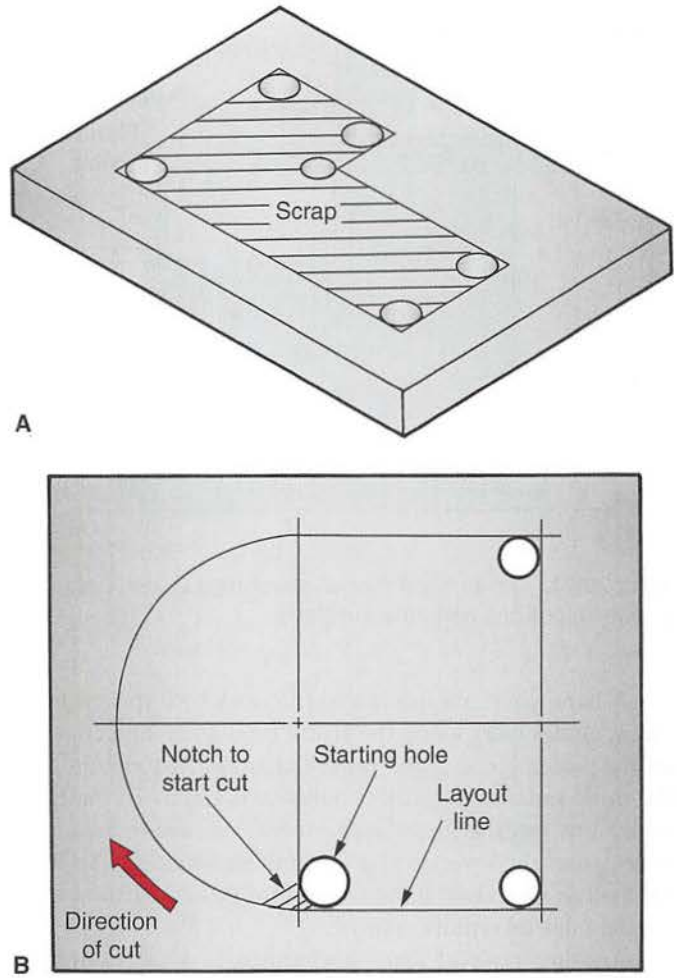
Figure 20-18. Vertical band machine designed for metal machining.



Goodheart-Willcox Publisher

Figure 20-19. An example of two-dimensional band machining. To protect your hands, be sure that a piece of backup metal is in place at the point where the saw blade breaks through.

piece. It is then welded, and the work is maneuvered along the prescribed lines. Additional holes must be drilled if sharp corners are to be made, **Figure 20-20**. It may be necessary to use the blade as a file to get the work into cutting position.



Goodheart-Willcox Publisher

Figure 20-20. Making internal cuts. A—Drilled holes are needed where sharp corners are to be made in an internal cut. B—Start internal cuts by using the blade as a file, and notch the work until the blade can be positioned to start the cut.

After completing the internal cut, cut the band close to the weld so that the entire weld can be cut away prior to rewelding. It is recommended that there be no more than one weld in the band.

20.6 Band Machine Power Feed

Power feed or mechanical pressure attachments are available for band machines. The simplest attachment makes use of weights to pull the work into the blade, **Figure 20-21**. Both of the operator's hands are free to guide the work.

Several types of hydraulic power feed attachments have been devised. On some vertical band machines, the worktable is hydraulically actuated and feeds the work into the blade at a constant rate. Accidental overfeeding is eliminated, greatly extending band life.

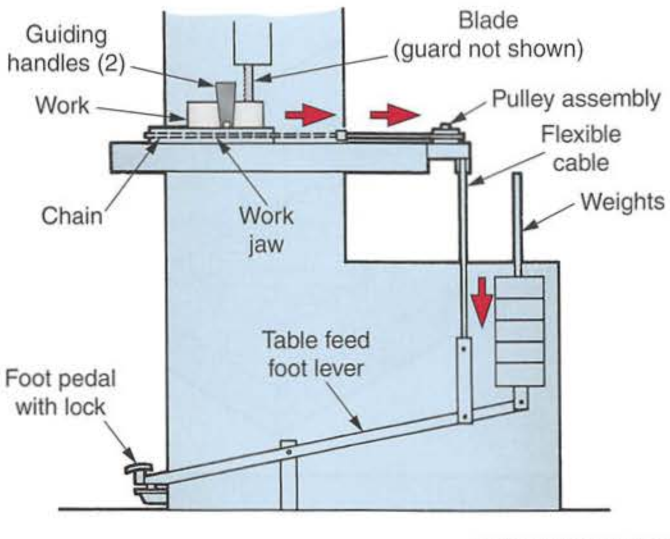


Figure 20-21. The simplest type of power feed system uses weights to pull the work into the blade.

A handwheel, connected to the work by a sprocket and chain, guides work along the layout line. A servomechanism on the lower blade guide senses changes in feed force on the work and automatically counteracts them. To maintain a constant feeding force on the work, the device advances, slows, stops, or reverses the worktable movement. To leave the operator's second hand free, a foot switch permits moving the table by remote control.

Another type of feed mechanism is a self-contained unit attached to the machine. A hydraulic cylinder applies and maintains constant pressure on the work through a sprocket-and-chain system. By turning a handwheel one direction or another, the operator can move the sprocket left or right, which in turn rotates the work. **Figure 20-22** shows the parts of this type of system.

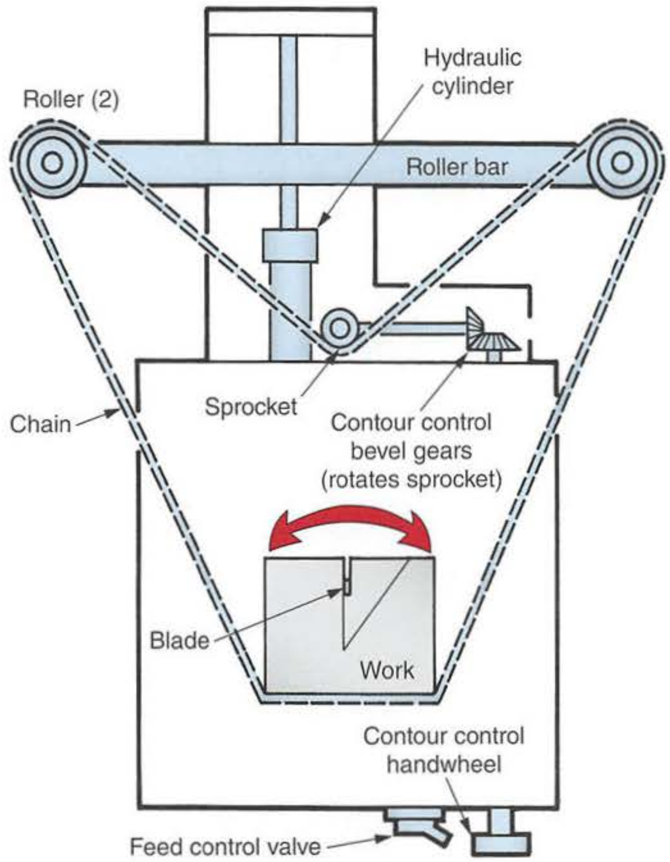


Figure 20-22. A hydraulically actuated power feed unit.

20.7 Other Band Machining Applications

The versatility of the band machine can be further enhanced by the addition of accessories or minor tool modifications. Band machines can be fitted with attachments for filing, polishing, or friction sawing. Non-toothed blades and customized vertical band machines permit other applications.

20.7.1 Band Filing

A smooth, uniformly finished surface may be obtained rapidly and with considerable accuracy on a band machine fitted for filing, **Figure 20-23**. A series of small file segments make up the ***file band***. The individual units interlock to form a continuous file. The segments are fitted to a flexible back. File guides replace the regular saw guides when a band file is used. A variety of file shapes and cuts are available.

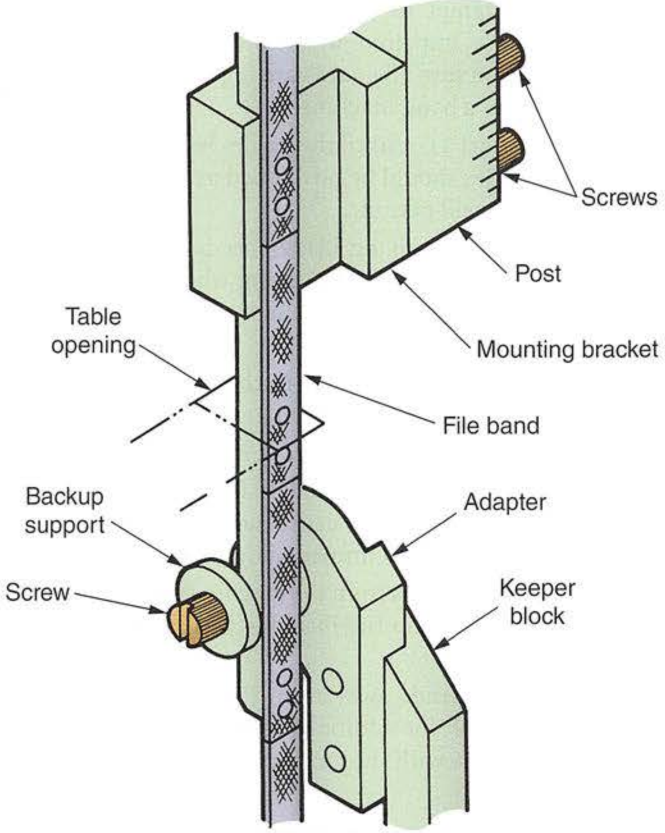
20.7.2 Band Polishing

Parts can be polished on a band machine with a polishing attachment, **Figure 20-24**. A continuous-band abrasive cloth replaces the saw blade. For best results, and to extend the life of the abrasive band, lubricate the back of the abrasive band.

20.7.3 Friction Sawing

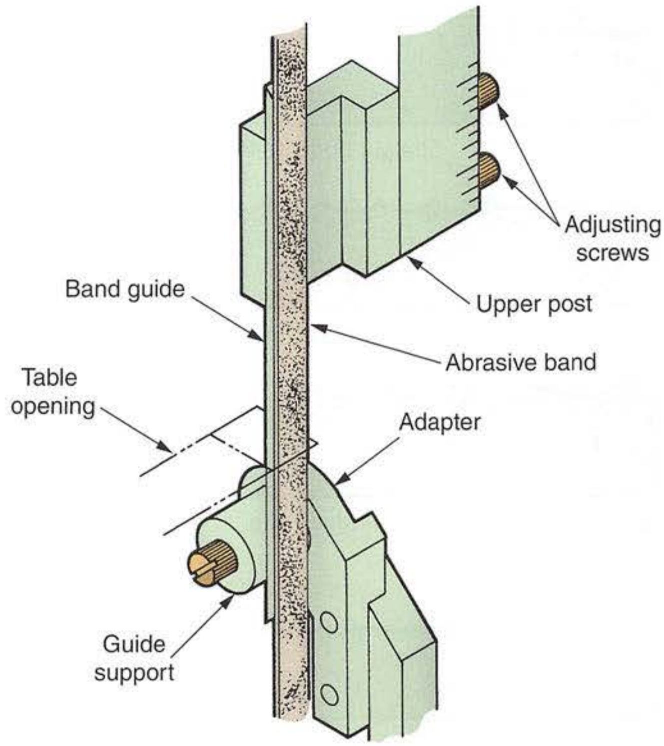
Friction sawing uses extremely high cutting speeds of between 6000 and 15,000 feet per minute (fpm) or 1800 to 4500 meters per minute (mpm) and heavy pressure to cut ferrous metals. In friction sawing, the high speed of the band generates sufficient heat to soften the metal just ahead of the blade. The band removes the softened metal as it moves through the work. Only a small area on either side of the blade is affected by the heat.

The blade teeth do not actually cut. They are used to scoop out the softened metal. As a matter of fact, dull teeth are superior to sharp teeth for this application because they generate heat better. Friction sawing is a spectacular operation that produces a shower of sparks.



Goodheart-Willcox Publisher

Figure 20-23. Band filing requires that special guides be used. In this illustration, the worktable has been removed for clarity.



Goodheart-Willcox Publisher

Figure 20-24. Polishing can be done on a band machine by replacing the saw blade with an abrasive band and using the guide and support shown. In this illustration, the worktable has been removed for clarity.

Friction sawing is a rapid way to cut ferrous metals less than 1″ (25 mm) thick. Thicker stock can be cut if a rocking movement is used. The hardness of the metal being cut does not affect the effectiveness of this cutting method.

The band drive wheels on machines used for friction sawing are usually large in diameter and are carefully balanced for smooth, vibration-free operation. Large wheels are needed to reduce fatigue on the band due to the high operating speed. A larger diameter keeps the band from bending as sharply as it loops around the wheel, reducing stress on the band.

Band machines designed for friction sawing cannot be used for conventional band machining unless they are equipped with variable-speed drives. Most band machines are not adaptable for friction sawing.

SAFETY NOTE

Friction sawing requires a full face shield, leather gloves, and a transparent shield fitted around the cutting area.

20.7.4 Other Band Tools

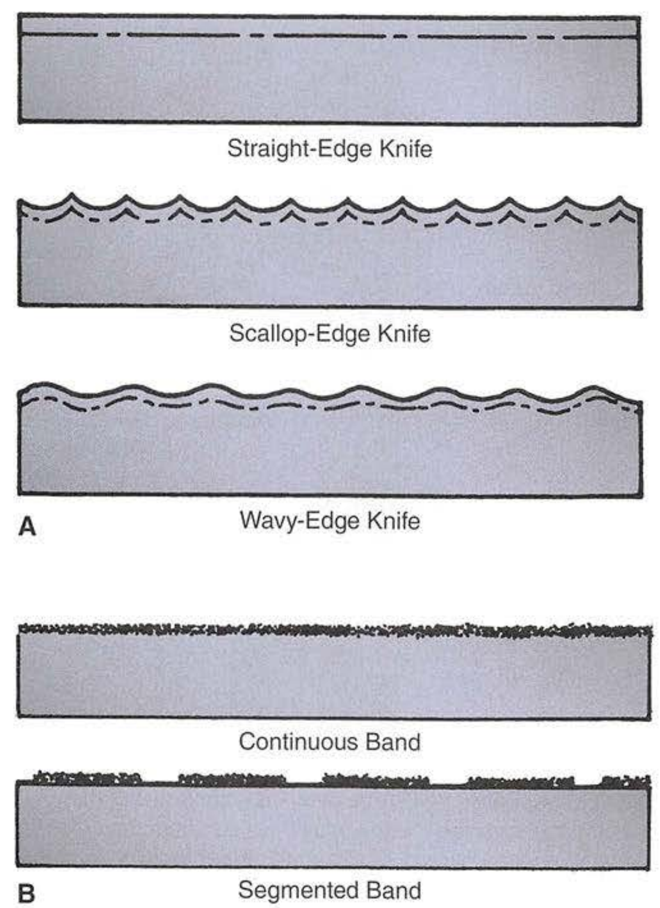
Toothed bands are most commonly used on the vertical band machine. However, other types of blades have been developed for special work, **Figure 20-25**. In addition to the bands mentioned in this section, blades with unusual characteristics are available to meet almost any band machining requirement.

The ***knife-edge blade*** is used to cut material that would tear or fray when machined by a conventional blade. For example, sponge rubber, cork, cloth, corrugated cardboard, and rubber would tear easily.

The ***diamond-edge band*** is designed to cut material that is difficult or impossible to cut with a conventional toothed blade. The diamonds are located on the front edge of the band where the cutting is accomplished. On a wire band, diamonds are fused around the circumference on the band, permitting it to cut in any direction.

20.7.5 Specialized Vertical Band Machines

The vertical band machine is manufactured in a wide range of sizes and has been adapted to do many kinds of band machining. Some large machines have been fitted with closed circuit TV and a remote control console to permit the operator to contour-machine large sections of material or to perform hazardous or dangerous work more safely. For example, this type of machine is used to machine toxic and radioactive materials.



Goodheart-Willcox Publisher

Figure 20-25. Other band tools. A—Types of knife edge blades. B—The cutting edge of a diamond-edge band is impregnated with diamond dust.

Band machines with computer numerical control (CNC) are also available. They have X and Y table movement capability, plus circular interpolation, which allows them to cut circular and elliptical contours.

20.8 Troubleshooting Band Machines

A number of problems can occur during band machining. Use the table shown in **Figure 20-26** to identify various problems and the methods used to correct each of them.

20.9 Band Machining Safety

Follow these guidelines to operate a band machine safely:

- Do not attempt to operate a band machine until you have received instructions on its safe operation.
- Never attempt to operate the machine while your senses are impaired by medication or other substances. Be sure all guards are in place before starting to operate a band machine.
- Do not start a cut until the guides have been set properly. Guides should be positioned as close to the work as the job will permit.
- Other than changing blade speeds (some machines require the band to be running when speed changes are made), make no adjustments until the blade has come to a complete stop.
- Wear eye protection and leather gloves when handling band blades or blade material.
- Get help when handling heavy material.
- Remove burrs and sharp edges from the work as soon as possible. They can cause serious cuts. Have cuts and bruises treated immediately. Report all injuries.
- Do not clean chips from the machine with your hands. Use a brush. Stop the machine before attempting to clean it.
- Keep your hands away from the moving blade. Use a push block for additional safety. Never have your hands in line with the cutting edge of the band.

GREEN MACHINING

Proactive Maintenance

There are two kinds of maintenance—proactive and reactive. Reactive maintenance is maintenance done after a problem already exists (a reaction to the problem). Proactive maintenance means taking steps to prevent a problem before it happens, so it is sometimes called preventive maintenance. Proactive maintenance saves time, money, and other resources in a variety of ways. Performing routine maintenance on machines and replacing parts regularly can help prevent major breakdowns, which often require expensive parts and take extended time to repair. Proactive maintenance keeps machines running efficiently and accurately, preventing wasted work hours and wasted materials.

20.10 Broaches and Broaching Machines

Recall from Chapter 7, *Hand Tools*, that broaching is a manufacturing process for machining flat, round, and contoured surfaces and is ideal for producing irregularly shaped openings. Both internal and external surfaces can be shaped by this process.

With broaching, a multitoothed cutting tool or broach is pushed or pulled across the work, **Figure 20-27**. Each tooth

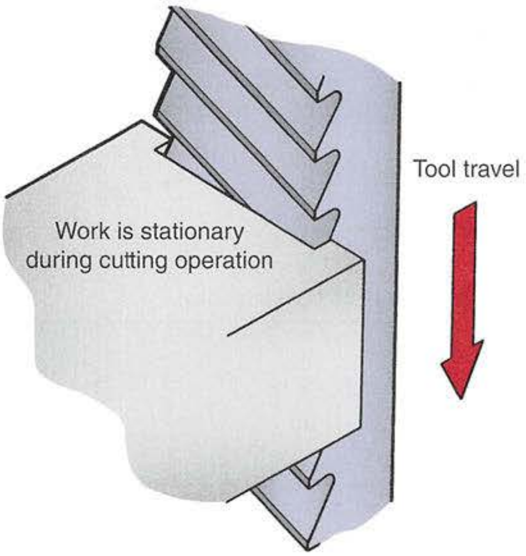
Troubleshooting Band Machines	
Problem	**Correction**
1. Teeth dull prematurely.	a. Use slower cutting speed. b. Replace blade with a finer pitch band. c. Be sure proper type cutting fluid is used. d. Increase feed pressure. e. Check to be sure band is installed with teeth pointing down.
2. Band teeth breaking out.	a. Reduce feed pressure. b. Use finer pitch band if thin material is being cut. c. Be sure work is held solidly as it is fed into band. d. Use a heavier-duty cutting fluid.
3. Band breaks.	a. Change to a heavier band. b. Reduce cutting speed. c. Check wheels for damage. d. If blade breaks at weld, use longer annealing time. Reduce heat gradually. e. Use finer pitch blade. f. Reduce feed pressure. g. Decrease band tension. h. Check blade guides for proper adjustment. i. Use cutting fluid.
4. Cutting rate too slow.	a. Increase band speed. b. Use coarser pitch blade. c. Increase feed presssure. d. Use cutting fluid.
5. Band makes "belly-shaped" cut.	a. Increase blade tension. b. Adjust guides close to work. c. Use coarser pitch band. d. Increase feed pressure.
6. Band does not run true against saw guide backup bearing.	a. Remove burr on back of band where joined. b. If hunting back and forth against backup bearing on guide, reweld blade with back of band in true alignment. c. Check alignment of wheels. d. Check backup bearing. Replace if worn.
7. Premature loss of set.	a. Use narrower band. b. Reduce cutting speed. c. Apply cutting fluid.

Goodheart-Willcox Publisher

Figure 20-26. Common band machine problems and their possible solutions.

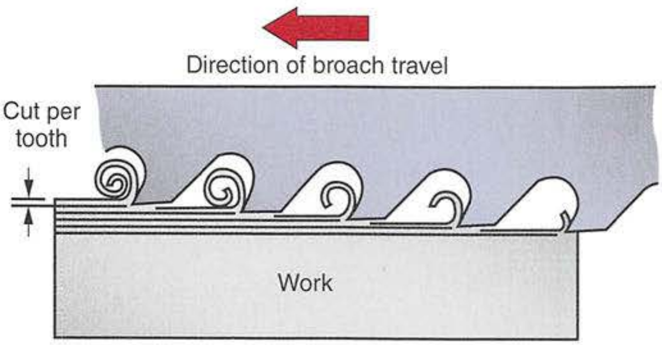
on the broach removes only a small portion of the material being machined, **Figure 20-28**. In many industrial applications, the broach is assembled from units with different tooth sizes. The tooth units are stacked on a mandrel to be pulled through the work, **Figure 20-29**.

There are three basic types of broaching operations: internal broaching, external broaching, and pot broaching. Each type of broaching operation requires a different type of broach.



Goodheart-Willcox Publisher

Figure 20-27. Broaching involves the use of a multitoothed cutting tool (the broach) that moves against the stationary work. The operation may be on a vertical or horizontal plane and may involve making internal or external cuts.



Goodheart-Willcox Publisher

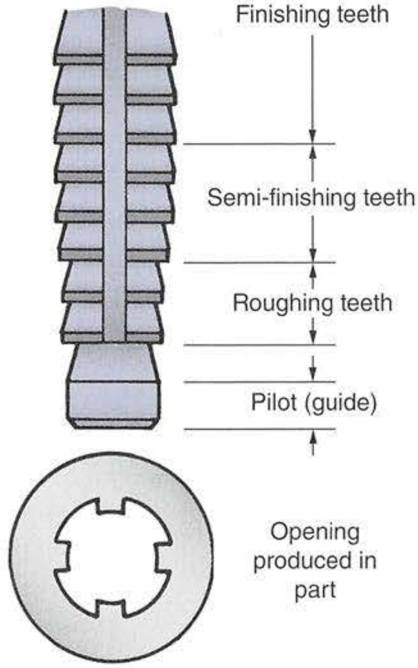
Figure 20-28. Each tooth on a broaching tool removes only a small portion of the material being machined.

20.10.1 Internal Broaching

Internal broaching requires a starting hole so that the cutting tool can be inserted. It uses a tapered pull broach. The first teeth are the smallest. Each tooth is only a few thousandths of an inch larger than the previous tooth. The broaching machine pulls the broach by the pull end, which extends below the first teeth and the pilot guide, **Figure 20-30**.

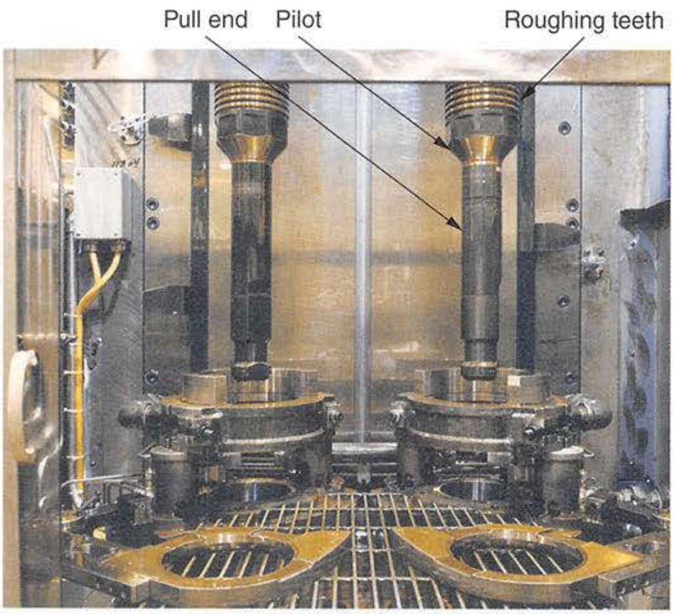
20.10.2 External Broaching

External broaching uses a slab broach, which is a flat-toothed strip that is usually held (singly or in groups) in a slotted fixture. Slab broaches can be several inches wide and several inches thick. External broaching is most often used to machine flat surfaces. However, it can also be used to cut slots or splines on the outside of a workpiece.



Goodheart-Willcox Publisher

Figure 20-29. This drawing shows a greatly shortened section of an internal broaching tool and the splines it cuts into a part. The pilot guides the cutter into a cut or hole previously made in the work. Each tooth of the broach increases slightly in size until the specified size is attained.



National Broach & Machine Co.

Figure 20-30. Two large pull broaches positioned above the clamping fixtures used to hold workpieces in this broaching machine. The pull end, pilot, and first set of roughing teeth are all visible.

20.10.3 Pot Broaching

In ***pot broaching***, the tool is stationary and the work is pushed or pulled through the broach, **Figure 20-31**. Slab broaches or ring-shaped broaches are held in a fixture called a *pot*. The pot is designed to hold multiple cutting tools concentrically. Pot broaching is often used to machine precision gears, **Figure 20-32**.

A variety of broaching equipment is available. The machines range in size from large, hydraulically powered broaching machines, **Figure 20-33**, to small arbor press units designed to cut keyways in gears, pulleys, and similar components.

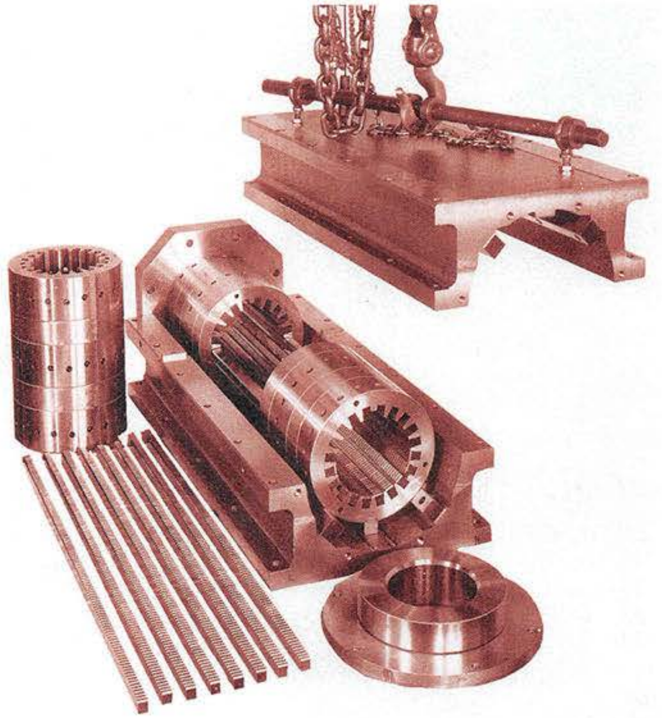
20.11 Advantages of Broaching

The broaching process offers several manufacturing advantages, including the following:

- High productivity.
- Ability to maintain close tolerances.
- Production of good surface finishes.
- Economy (even though initial tooling costs can be high unless standard tooling is used).
- Long tool life, since only a small amount of material is removed by each tooth.
- Freedom to use semiskilled workers, since equipment is automated.

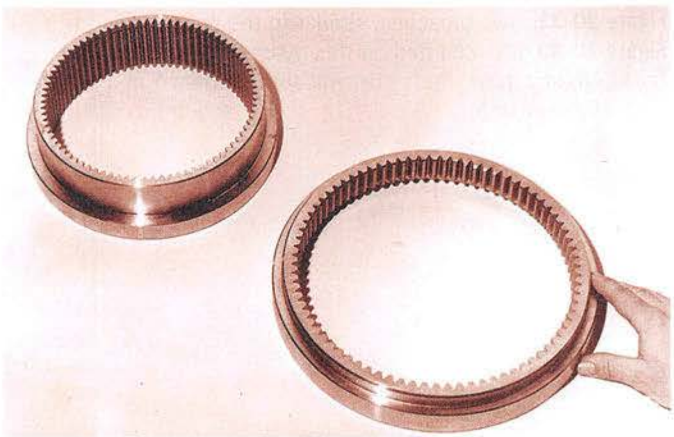
Almost any material that can be machined by other techniques can be broached. Consistently close tolerances can be maintained by the broaching process. While the surface finishes produced by broaching are smooth compared to many other machining processes, they can be further improved by adding ***burnishing*** (noncutting) elements to the finishing end of the broach. Burnishing finishes a metal surface by compressing it, often with steel balls.

The machining operation can usually be completed in a single pass of the cutting tool. When properly employed, broaching can remove metal faster than almost any other machining technique. Small parts can be stacked and shaped in a single pass, **Figure 20-34**, although larger units may require several passes to machine all surfaces.



National Broach & Machine Co.

Figure 20-31. Slab broaches, shown at the lower left, are mounted to the pot and used to cut slots on the outside of the workpiece as it is pushed through the stationary fixture.



National Broach & Machine Co.

Figure 20-32. The internal teeth of these precision gears were machined using the pot broaching process.

National Broach & Machine Co.

Figure 20-33. Two broaches, similar to the ones shown in **Figure 20-30**, are mounted on this hydraulically powered broaching machine. Parts are typically completed in a single pass of the broach.

TheFinalMiracle/Shutterstock.com

Figure 20-34. Typical small parts machined by broaching. Small parts can be stacked and machined in a single pass.

Chapter Review

Summary

- Band machining offers a variety of advantages over other machining operations, including efficient material removal and downward cutting action.
- Blade type and blade characteristics, such as the width, pitch, set, gage, and tooth form, are important factors when choosing the proper blade.
- For a blade to be properly welded, the ends must be squared. Several teeth are removed, the ends are welded together, the flash is ground away, and the blade is annealed.
- Proper band machine preparation and blade maintenance are important for safe band machining operation.
- Various power feed mechanisms are available for band machines.
- Special blades and attachments for band filing, polishing, friction sawing, and other operations make the band machine very versatile.
- Most problems with band machines can be solved by analyzing the problem and taking appropriate corrective action.
- The three basic types of broaching are pull broaching, slab broaching, and pot broaching.
- Broaching has many advantages over other machining processes.

Review Questions

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

1. Band machining makes use of a(n) ______ saw blade.
2. List three advantages band machining has over other machining techniques.
3. What two points must be considered when selecting a blade for a specific job?
4. When making straight cuts, use the ______ blade the machine can accommodate.
5. Blade pitch refers to ______.
   A. the width of the blade in inches or millimeters
   B. the thickness of the blade in inches or millimeters
   C. the number of teeth per inch of blade or tooth spacing in millimeters
   D. All of the above.
   E. None of the above.
6. Tooth form is the ______.
   A. shape of the tooth
   B. thickness of the blade
   C. number of teeth on the blade
   D. All of the above.
   E. None of the above.
7. A resistance ______ welder is used to weld band machine blades.
8. The blade must be ______ after welding because the joint is extremely brittle and cannot be used in this condition.
9. The cutting tool must be installed with the teeth facing ______.
10. Blade tension is the pressure put on the saw band to ______.
    A. cut the metal more rapidly
    B. keep it taut and tracking properly
    C. reduce the power needed to do the cutting
    D. All of the above.
    E. None of the above.
11. What is the simplest form of band machining?
12. The worktable on many vertical band machines is mounted on trunnions so that it can be tilted to make ______ cuts.
13. How can internal cuts be made on a band machine?
14. List three types of power feeds available for band machines.
15. Smooth, uniformly finished surfaces are possible when the machine is fitted for ______.
16. Describe friction sawing.
17. Of what use is a knife-edge blade on a band machine?
18. When are diamond-edge bands used?
19. What is unique about a diamond impregnated wire band?
20. What are the possible remedies if a band machine makes a "belly-shaped" cut?
21. What commonly used tool can help keep your hands away from the moving blade of a band machine as it nears the end of the cut?
22. Broaching is a manufacturing process for machining ______ surfaces.
    A. flat
    B. round
    C. contoured
    D. All of the above.
    E. None of the above.

23. What are the three types of teeth on the cutting tool of a broaching machine?
24. What does internal broaching require that external broaching does not?
25. How does pot broaching differ from internal and external broaching?
26. List three advantages offered by broaching.
27. With broaching, the machined surface can be further improved by adding ______ elements to the finishing end of the broach.

For questions 28–30, match each description with the correct term.

28. Recommended for heavy-duty sawing.
29. Used for high-speed sawing of free-machining nonferrous metals.
30. Applied when the machine is not fitted with a built-in coolant system.

A. Mist coolant
B. Flooding
C. Solid lubricants

CHAPTER **21**

Introduction to CNC Machining

Chapter Outline

21.1 History of CNC
21.2 Advantages and Disadvantages of Using CNC
21.3 CNC Milling Machines
- **21.3.1** Basic CNC Milling Machines
- **21.3.2** Machining Centers

21.4 CNC Turning Machines
- **21.4.1** Basic CNC Lathes
- **21.4.2** Gang-Tool Lathes
- **21.4.3** Turret Lathes
- **21.4.4** Swiss-Type Turning Centers

21.5 CNC Safety
21.6 CNC Coordinate Systems
- **21.6.1** Cartesian Coordinate System
- **21.6.2** Polar Coordinate System

21.7 CNC Movement Systems
- **21.7.1** Stepper Motors
- **21.7.2** Servomotors
- **21.7.3** Lead Screws and Lead Nuts

Learning Objectives

After studying this chapter, you will be able to:

- Describe the development of CNC technology.
- List advantages and disadvantages of using CNC technology.
- Describe the features of CNC milling machines.
- Compare the characteristics of various types of CNC turning machines.
- Identify safety guidelines for CNC machining processes.
- Summarize the use of the Cartesian and polar coordinate systems in CNC technology.
- Contrast the two types of motors that are commonly used to drive CNC machines.

Technical Terms

automatic tool changer (ATC)
backlash
Cartesian coordinate system
closed-loop system
conversational language
coordinate system
dry cycle
encoder
horizontal machining center (HMC)
lead screw
lockout/tagout procedures
machining center
open-loop system
polar coordinate system
servomotor
stepper motor
Swiss-type turning center
turning center
vertical machining center (VMC)

Since their development, computer numerical control (CNC) machines have revolutionized machining. CNC machines can perform complex machining operations faster than manual machines with greater accuracy and ease. This chapter introduces CNC milling and turning machines, including their history, advantages, and disadvantages. This chapter also covers CNC safety and the basics of CNC movement systems.

21.1 History of CNC

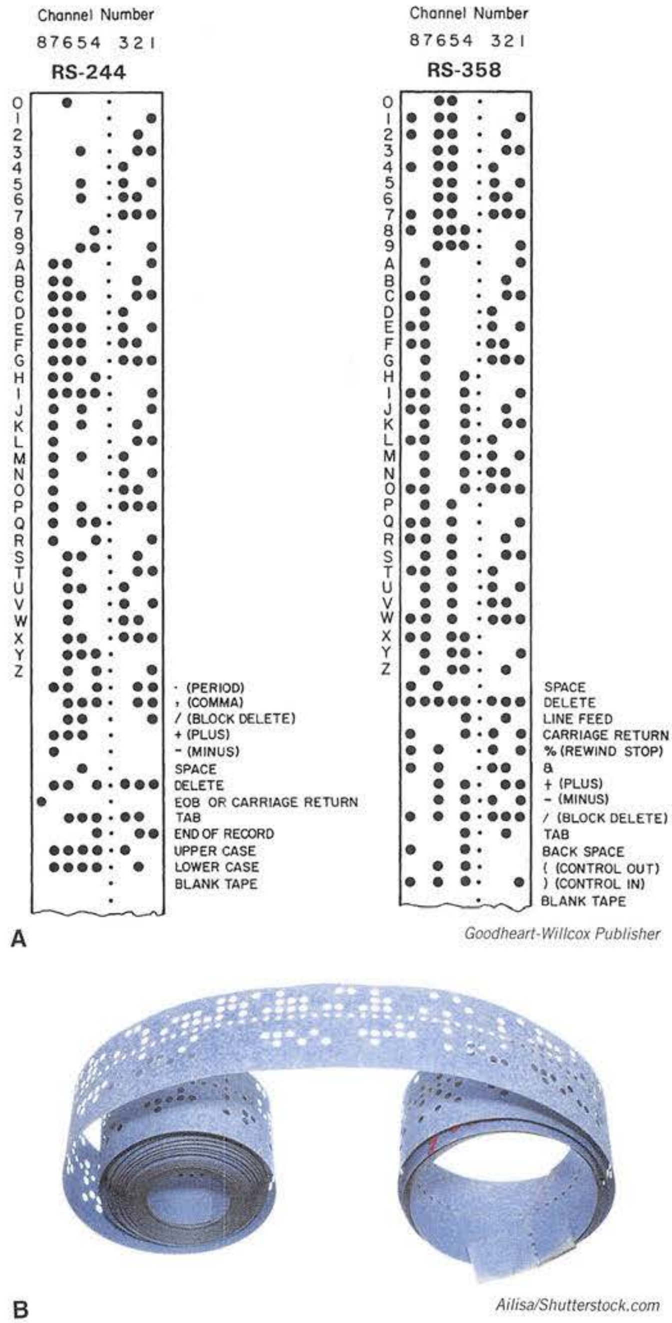
Computer-controlled machining got its start during the 1950s as numerical control (NC). Programs to control machine tools were created and stored on punched paper tape, **Figure 21-1**. Programmers punched holes in the tape at specific locations to produce multiple combinations. Each combination represented a separate letter, number, character, or command. Paper tape readers read the rows of punched holes on the tape at a rate of up to 1,000 lines per second and transferred the information to the computer that controlled the motion of the machine tool.

When these machines were first introduced, the cost of the machines and the skills needed to operate them outweighed the potential cost savings, so many companies were slow to develop the technology. However, by the end of the 1950s, the use of NC equipment was beginning to increase.

The development of the minicomputer in the 1960s dramatically increased the acceptance and use of NC equipment. This technology converted numerical control to computer numerical control (CNC). Computer control lowered the cost of the equipment and made it more cost-effective due to the faster processing capability of the minicomputer.

Also during the 1960s, programming codes were developed to help standardize CNC programming languages. Until then, each machine manufacturer had implemented its own language. These codes, which are still in use today, are described in Chapter 22, *CNC Programming Basics.*

As computer and machine drive technologies continued to develop from the 1970s through the 1990s, equipment became faster and more capable of producing complex shapes, making CNC machining a more cost-effective method, **Figure 21-2**. Also, the use of computer memory to store programs began to replace the use of punched paper tape. Today, the use of punched paper tape is all but obsolete.



Goodheart-Willcox Publisher
Ailisa/Shutterstock.com

Figure 21-1. Punched tape. A—Tape code. Note that every level of RS-244 tape has an odd number of punches for parity check. RS-358 tape has an even number. Parity check is a method of automatically checking to reduce the possibility of tape errors caused by a malfunctioning tape punch. B—A roll of punched tape.

21.2 Advantages and Disadvantages of Using CNC

The primary advantage of using CNC equipment is that it eliminates much of the variation caused by human interaction during the manual machining process. Even the best machinists cannot perfectly duplicate their actions for every part. A CNC machine is not perfect, but its movements are better controlled than those of a human, which reduces variation in the parts that are produced. Another general advantage is the rate at which parts are produced. The faster parts can be machined, the less they cost per unit to manufacture. This allows manufacturers to offer lower prices to customers, making them more competitive in the global marketplace.

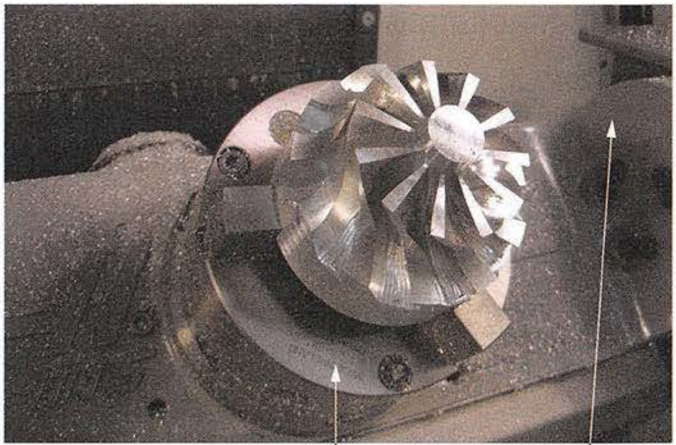


Figure 21-2. A helical impeller machined on a vertical machining center. The complex contours of this part can be machined thanks to the additional axes of rotation provided by the tilting trunnion table and rotary table under CNC control.

Goodheart-Willcox Publisher

Figure 21-3. This automatic tool changer (ATC) holds the tools in a tool magazine and uses a pivot insertion system to quickly and automatically change tools on a horizontal machining center.

The biggest disadvantage of using CNC is the initial cost of the machine. As costs continue to decrease, CNC technology is becoming more affordable. However, some small companies still lack the resources to purchase the necessary equipment.

21.3 CNC Milling Machines

A basic CNC milling machine is a traditional milling machine with built-in CNC capabilities. ***Machining centers*** are CNC milling machines equipped with an ***automatic tool changer (ATC),*** a device that automatically changes and stores the tools, **Figure 21-3**. The machining cycle pauses for the automated tool change process, but the automated tooling change takes much less time than a manual tooling change. Most machining centers are enclosed.

The advantages of CNC milling machines are that they increase part output through faster operation compared to manual machining, and their reduced variation yields a higher quality product. Options such as additional rotating axes can be added to increase the versatility of the machine, but these options also significantly increase the cost.

The greatest disadvantage of CNC milling machines is their high cost. Various types of CNC milling machines are available, with prices ranging from moderately expensive to hundreds of thousands of dollars or more.

21.3.1 Basic CNC Milling Machines

The basic CNC milling machine, like its manual counterpart, has three axes of movement and requires tooling to be changed manually. The machine shown in **Figure 21-4** is an example of a basic CNC milling machine. It can be operated both manually and in full CNC mode.

Bridgeport Machines, Inc.

Figure 21-4. A Bridgeport 3-axis CNC vertical milling machine. This basic CNC milling machine can be controlled both manually and in full CNC mode.

Traditional machinists can operate a basic CNC milling machine manually, or they can be trained to create programs using a method known as ***conversational language.*** In this

method, the control is equipped with software that allows the operator to select operations from menus. The software converts the operator's input into standard programming code for the control, which then drives the equipment to machine the parts. Conversational language is described in Chapter 22, *CNC Programming Basics.*

Basic CNC milling machines are usually equipped with quick-change collets to allow the operator to change tools during the programmed cycle, **Figure 21-5**. A CNC program can be written with pauses to allow an operator to change tools manually. Quick-change collets are less rigid than normal collets and can only be used for light- to medium-duty operations.

The biggest advantage of the basic CNC milling machine is its cost. It is well suited to the toolroom environment and allows traditional machinists to operate it manually or in full CNC mode to create small batches of parts. However, the basic CNC milling machine is not suitable for high-volume production because it lacks the necessary speed, precision, and repeatability.

Royal Products, Division of Curran Manufacturing Corporation

Figure 21-5. Inserting an end mill mounted in a quick-change collet. Quick-change tool systems provide a fast method of changing tools on basic CNC milling machines.

21.3.2 Machining Centers

The spindle of a machining center can be either horizontal or vertical. ***Horizontal machining centers (HMCs)*** have the same features as manual horizontal milling machines. HMCs offer rigid, horizontally mounted spindles and stable workholding methods, **Figure 21-6**.

Vertical machining centers (VMCs) have vertically mounted spindles and feature the same capabilities as manual vertical milling machines with the added benefit of computer control and automatic tool changing, **Figure 21-7**. VMCs can be equipped with additional axes of rotation for machining parts with complex features that cannot be

MHI Machine Tool U.S.A., Inc.

Figure 21-6. A fixed-bed horizontal machining center. This CNC milling machine has a 40-ton automatic tool changer and can handle work weighing up to 17,600 lb (8000 kg).

Mazak

Figure 21-7. A vertical machining center (VMC) with a 24-tool capacity.

machined in a single CNC program cycle within the cubic space provided by a basic 3-axis system, **Figure 21-8**. The additional axes allow for more flexibility in the types and complexity of parts that can be produced.

The advantages of the VMC are tied to its machining capabilities. VMC machines vary in accuracy, precision, speed, power, tool change capabilities, and the ability to add rotational axes. Their biggest disadvantage is cost. As capabilities increase, so do the costs of the machines. Costs can be controlled by selecting equipment with fewer options, then tailoring the machine setups in-house to meet the requirements. However, such practices limit the flexibility of the machines.

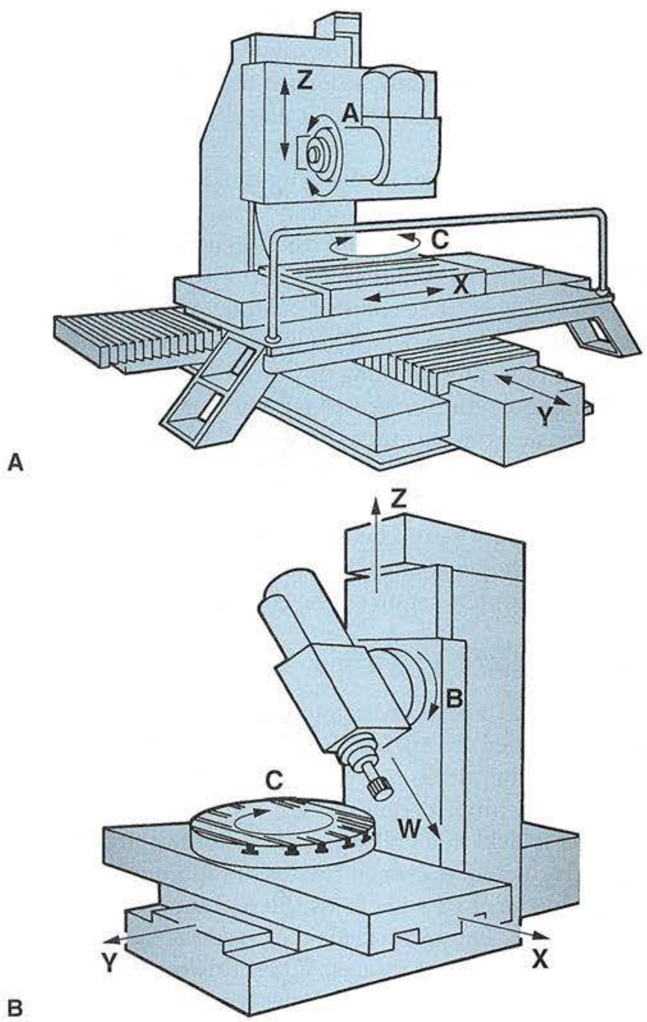
21.4 CNC Turning Machines

CNC turning machines are designed to machine cylindrical parts about an axis of rotation. A CNC turning machine is basically a computer-controlled lathe, and it has the same features as a manual lathe. A CNC turning machine equipped with an automatic tool changer is known as a ***turning center***, **Figure 21-9**.

There are many types of CNC turning machines, each with its own advantages and disadvantages. The four types of CNC turning machines described in this chapter are presented in order from least expensive to most expensive because one of the first things a company studies when considering the purchase of CNC equipment is the cost of the machinery.

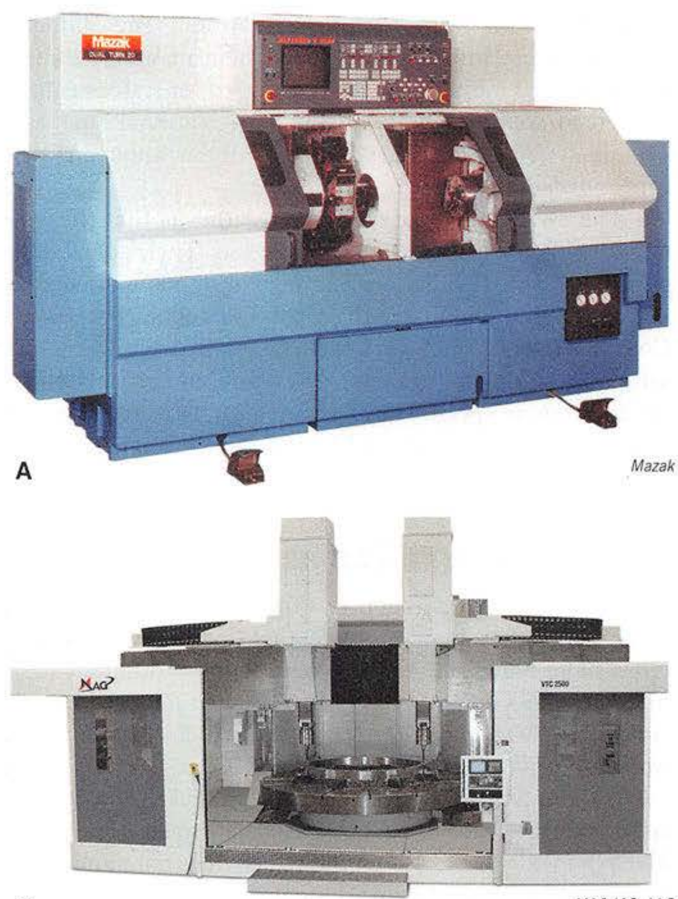
21.4.1 Basic CNC Lathes

Basic CNC lathes are common lathes with CNC capabilities included, **Figure 21-10**. Some CNC lathes can be operated in manual mode, just like a traditional engine lathe, as well as in full CNC mode. The control system built into this machine uses the same type of programming used in more expensive models. Its computer capabilities differ only in the amount of storage memory available. As memory capacity increases, so does the cost of the machine.



Goodheart-Willcox Publisher

Figure 21-8. Multiaxis machining. A—A 5-axis CNC milling machine. B—A 6-axis vertical machining center (VMC).



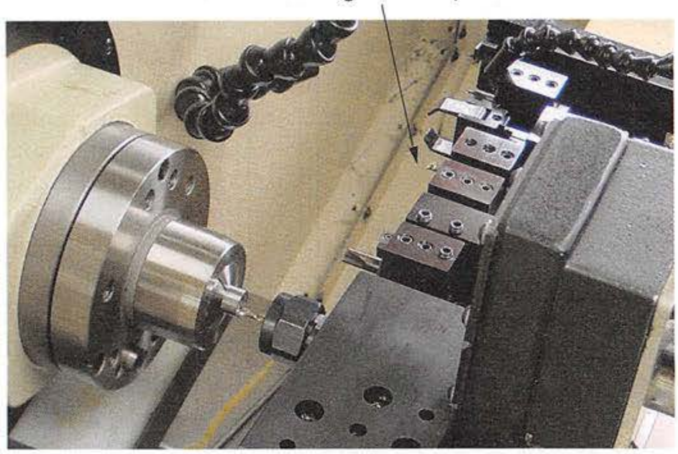
Mazak

MAG IAS, LLC

Figure 21-9. Turning centers. A—This 4-axis turning center permits operations to be performed on both sides of the workpiece, reducing machining time. B—Another example of a 4-axis vertical turning center.

Bridgeport Machines, Inc.

Figure 21-10. A basic CNC lathe with 2-axis CNC control.



Goodheart-Willcox Publisher

Figure 21-11. Multiple tool setup on a CNC gang-tool lathe. The workpiece is held in a collet chuck.

Tooling changes are done by the operator. The biggest advantage of the basic CNC lathe is its cost. Since tool changes are not automated on a basic machine, the cost of the hardware required for such automation is avoided.

CNC lathes work well in a toolroom environment. Traditional machinists can operate them manually or, if a special run of multiple pieces is required, operators can be trained to create programs using conversational language. This allows operators to program machines without an intricate knowledge of CNC programming.

The disadvantage of basic CNC lathes is that they are not suitable for high-volume production. Their speed, precision, and repeatability are better than in manual machining, but they cannot match the capabilities of more advanced CNC machines. CNC lathes can be equipped with gang-tool setups or automated turrets, described later in this chapter, to increase speed. However, their lack of precision and repeatability can create quality problems in high-precision manufacturing.

21.4.2 Gang-Tool Lathes

Gang-tool lathes are a step up from the basic CNC lathe. Gang-tool lathes have multiple tools mounted along the cross-slide, **Figure 21-11**. This setup allows different tools to be used in a single program cycle. The tool to be used is controlled by the position of the cross-slide relative to the workpiece mounted in the spindle and collet.

Although this type of setup can be added to a basic CNC lathe, the cross-slide travel on a basic machine is usually limited to a maximum of 12″ to 18″, which does not allow room for several tools to be set up. Therefore, gang-tool setups typically require a larger, more expensive machine. The need for a larger cross-slide travel and for higher-volume production usually drive the need to upgrade from a basic CNC lathe to a gang-tool lathe.

The advantages of a gang-tool lathe include the ability to produce more parts per hour than a basic CNC lathe. Its cost, while higher than the basic CNC lathe, is still within the range of most manufacturers. The gang-tool setup allows for rapid tool changes and, as long as the different parts being machined are of the same or similar diameter, low changeover times.

The disadvantages of the gang-tool lathe are its limited flexibility and the space required between the tools in the setup. If a changeover is required between parts whose diameters are drastically different, the entire toolholder setup on the cross-slide must be adjusted, adding to the time required to execute the changeover. This is especially true when there are several tools set up along the cross-slide.

Gang-tool setups work well for machining small-diameter parts. Because the part turns between the tools on the cross-slide, the smaller diameters allow for more tools to be set up in gang-type fashion. Larger-diameter parts do not work well with a gang-tool setup, because the space needed between tools for the larger-diameter workpiece restricts the number of tools that can be set up on the cross-slide.

21.4.3 Turret Lathes

CNC lathes that have a rotating turret attached to the cross-slide are called *turret lathes*. The computer controls the rotation of the turret and the movement of the cross-slide based on the information written into the CNC program. The part is held in a collet or chuck on the spindle. As the spindle is rotated, the turret rotates the appropriate tool into place, and the cross-slide feeds the tool into the workpiece, **Figure 21-12**. The apron feeds the cross-slide along the axis of the workpiece as it rotates. When this machining step is complete, the cross-slide withdraws from the workpiece, the turret rotates the next tool into place, and the next machining step begins. This process repeats until the machining process is complete.

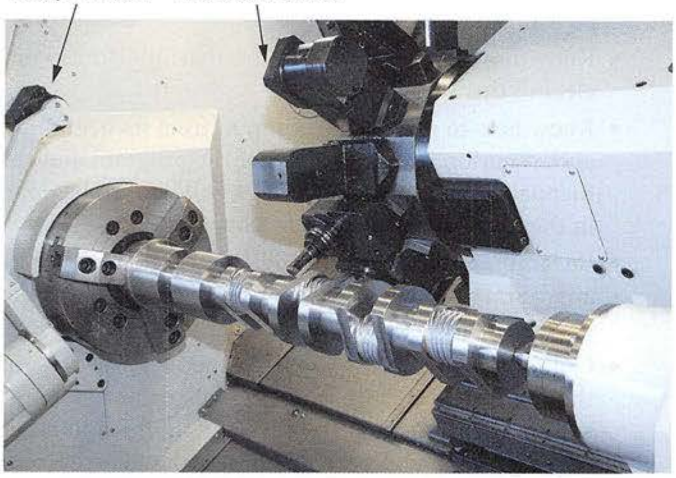


Figure 21-12. A large turret lathe machining a camshaft. The workpiece is held in a 3-jaw universal chuck. The cutting tools are held in a 12-station turret and are rotated into place. The tool presetter measures the offset lengths of the tools held in the turret for locating purposes.

The advantages of a turret lathe are its accuracy and speed. With the proper toolholding technology, the machine can be changed over from one part to another in a minimal amount of time.

The disadvantages of turret lathes include their cost and the skill set needed by operators to operate the equipment effectively. Tooling changes and part setups and changeovers are part of the daily operation of this machine, so the operator must be familiar with the machine controls. Often, the operator has to make frequent adjustments to the program to modify tooling offsets. The level of operator training required is therefore higher than for basic and gang-tool CNC lathes.

21.4.4 Swiss-Type Turning Centers

Complex details require a high level of precision in their machining. When a part to be machined includes elaborate detail, ***Swiss-type turning centers*** are used, **Figure 21-13**. These machines use a guide bushing to hold the workpiece tightly at the point of machining. The tools are not fed along the cylindrical axis of the part being machined. Rather, the main spindle behind the guide bushing moves the part along the cylindrical axis. The tools are fed along a direction perpendicular to the cylindrical axis of the part, as with cross-slide motions on other lathe types, **Figure 21-14**. The tools on a Swiss-type turning center can be set up gang-tool style, or a turret can be used.

The advantage of a Swiss-type machine is its ability to accurately machine small, complex parts with intricate details, **Figure 21-15**. The biggest disadvantage is cost. The machines are extremely expensive, and the raw materials needed to take advantage of the capabilities of this type of equipment are also expensive. The guide bushing diameter is not adjustable. To ensure that the part does not deflect during the machining operation, the stock must be ground to a precise diameter before machining with the Swiss-type turning center can begin. This increases the cost of the raw materials significantly. Swiss-type turning centers should be considered when the parts being produced require a high level of accuracy and precision.

Mack Molding Co., Arlington, Vermont

Figure 21-13. These 7-axis Swiss-type turning centers have 21 tools each and are coupled with bar feeders to automatically load precision-ground stock into the machine.

Mack Molding Co., Arlington, Vermont

Figure 21-14. This Swiss-type turning center comes equipped with an additional axis on the sub spindle, allowing the center to machine both ends of the workpiece simultaneously. This feature, along with a rapid feed rate, significantly reduces cycle times.

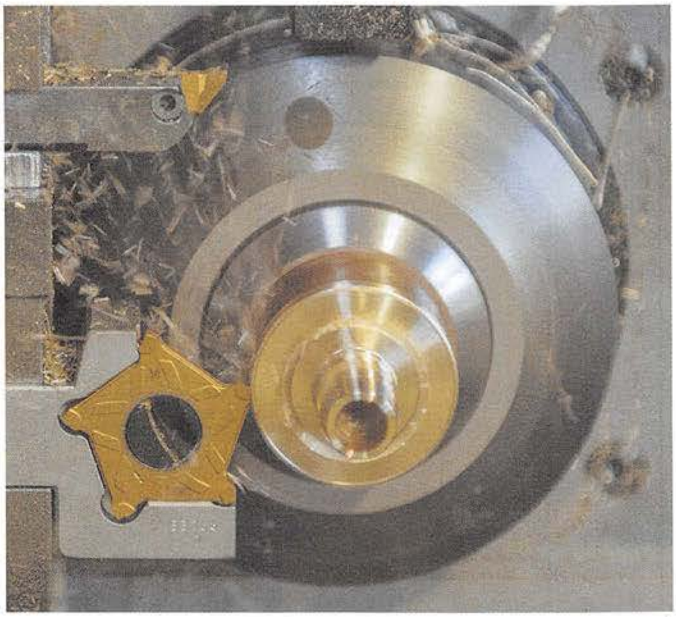


Figure 21-15. A small, intricate component made of brass being machined on a Swiss-type turning center with a gang-tool setup.

21.5 CNC Safety

The only difference between CNC machines and their manually operated counterparts is that the CNC machines are automated. They are controlled by a computer system as it executes a computer program written by a human. Therefore, the safety warnings and cautions related to manual machines are applicable to CNC equipment as well. Review Chapter 3, *Shop Safety*, for more information.

The following are additional safety concerns related solely to CNC equipment:

- Before operating a machine for the first time, obtain training on the machine. You must have a working knowledge of the machine and safe operating practices. Different machines may have different operating mechanisms. Read the operation and maintenance manuals provided with each machine.
- Do not remove or bypass guards, doors, and other safety devices on any CNC machine so that they do not work as they were designed.
- Know the sequence of operations executed by any program run by the machine you are operating. This allows you to recognize problems with the machine when it does something out of the ordinary.
- When changing tools manually, wait for the machine axes and the spindle to come to a complete stop.
- Know where the emergency stop buttons are located. One will always be located on the control panel and others may be located in other areas around the machine. The emergency stop brings the program to an immediate halt.
- Know how to manually jog the machine to a point clear of the workpiece.
- Know how to restart the program from its beginning and, if multiple tools are part of the program, how to manually change the tools. This will have to be done in the event of an emergency stop.
- Be familiar with all error codes and cautions that may appear on the control screen. Know what steps to take to correct each error code.
- Obtain training in the use of all auxiliary devices on the machine. Many machines have coolant systems, automated lubrication systems for spindles and other moving parts, and automated chip handlers. Know how to maintain these.
- Never try to remove chips while the machine is running, even if the chips appear to be clear of the work area.
- Cycle all programs once without a part in place during setup (this is called a ***dry cycle***). It is best to run the program in single-step mode, with the feed rate overridden to its lowest range. This can prevent an accidental machine crash.
- CNC programs can also be verified by machining a sample part from plastic, wax, or a similar inexpensive material, **Figure 21-16.**
- Be careful around automated tool changers. They can move without warning and trap an operator against a part of the machine. Automated tool changers are usually pneumatically powered, so take care around compressed air systems as well.

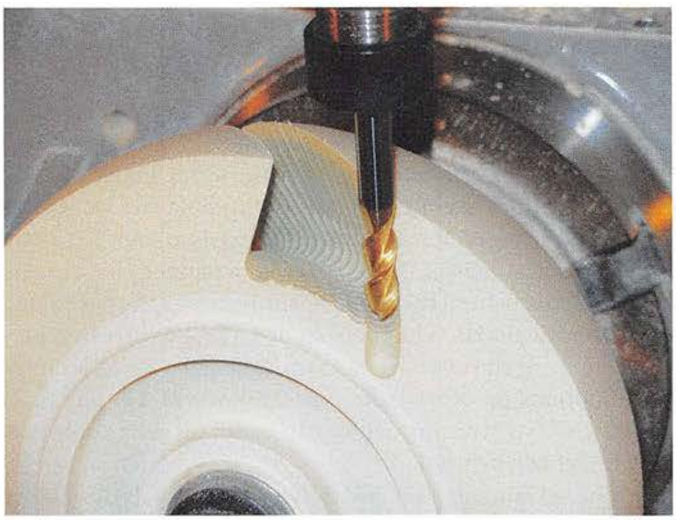


Figure 21-16. CNC programs can be verified by producing a part in an inexpensive material, such as plastic or wax.

- Keep all cooling fans, intakes, and exhausts in the computer control panel clean and free of obstructions. This helps prevent overheating and malfunction of equipment and possibly even equipment fires.
- If the machine crashes or does anything out of the ordinary, stop the machine, follow OSHA's ***lockout/tagout procedures***, and contact the area supervisor. Lockout/tagout procedures ensure the equipment is stopped and its energy source is removed to prevent accidental injuries.

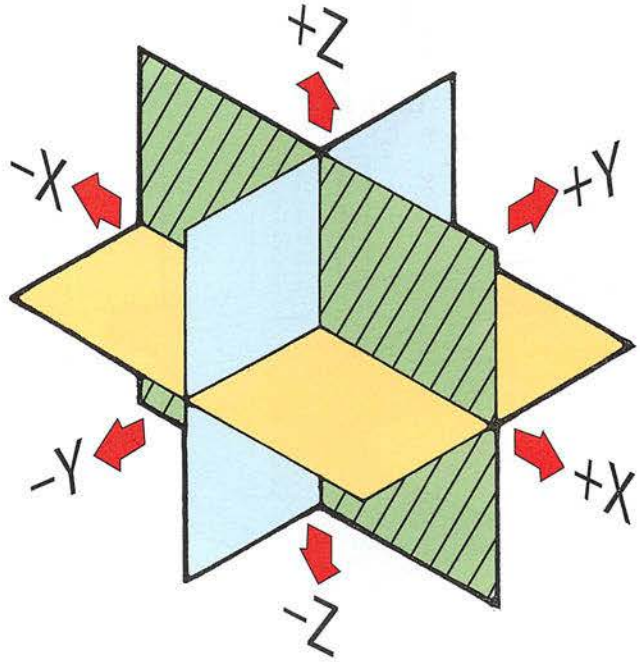
21.6 CNC Coordinate Systems

The CNC program drives the motors that position all of the moving axes. The program uses a ***coordinate system*** to communicate the direction and distance the workpiece or tool must move to the motors and the control. The most common systems in use today are the Cartesian coordinate system and the polar coordinate system.

21.6.1 Cartesian Coordinate System

The ***Cartesian coordinate system***, shown in **Figure 21-17**, forms the basis of CNC programming. Programs, written in either inch or metric units, specify the destination of a particular movement. With the destination established, the axis of movement (X, Y, or Z) and the direction of movement (+ or –) can be identified. To determine whether the movement is positive (+) or negative (–), the program is written as though the tool, rather than the work, is doing the moving.

Spindle motion is assigned the Z axis. This means that for a drill press or vertical milling machine, the Z axis is vertical. For machines such as a lathe or horizontal milling machine, however, the Z axis is horizontal. See **Figure 21-18**.



Goodheart-Willcox Publisher

Figure 21-17. The Cartesian coordinate system is the basis of all CNC programming. Each of the three major axes (X, Y, and Z) is perpendicular (at a 90° angle) to the other two. The arrows indicate the direction of travel, positive or negative.

The system of coordinates used for machine axis designation is specified according to the right-hand rule of Cartesian coordinates, as shown in **Figure 21-19**.

Machining instructions can be programmed directly into computer memory by entering the information through the keypad on the control. Programs can also be created and

WORKPLACE SKILLS

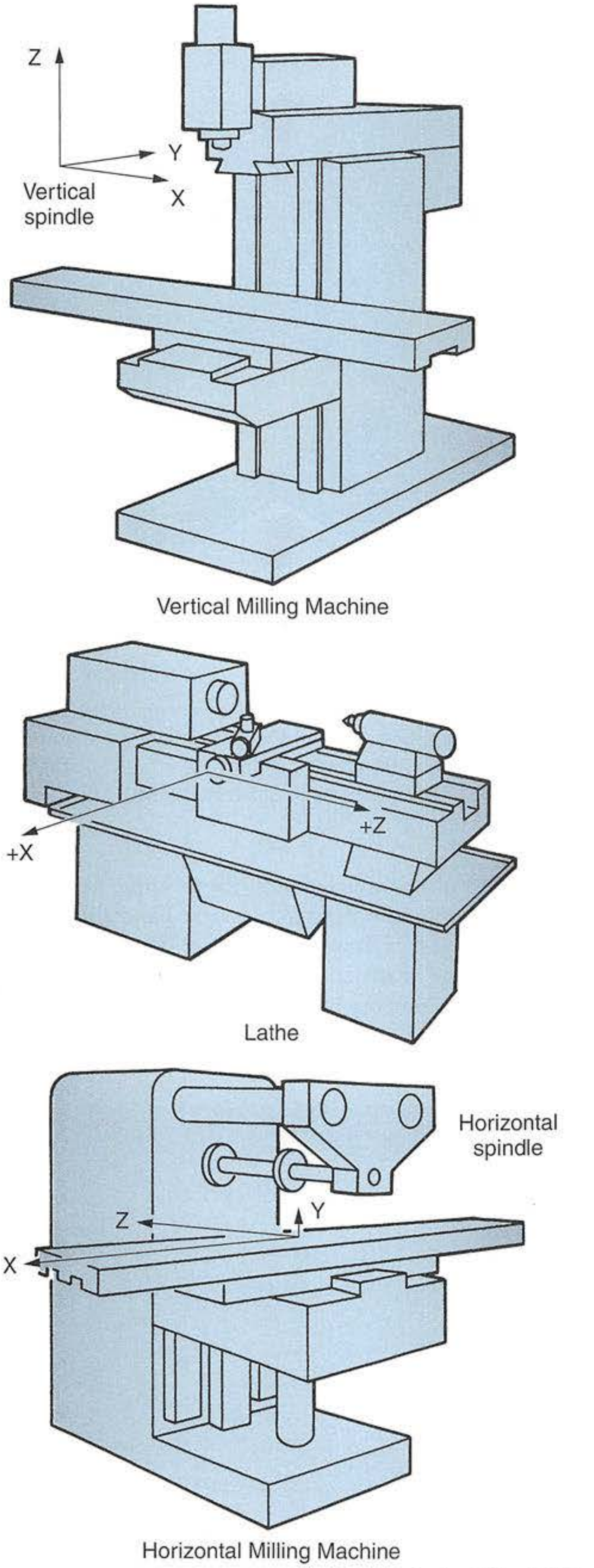
Decision Making and Problem Solving

Employers value workers who have the ability to make sound decisions and solve problems that arise. The processes for making decisions and solving problems are similar:

1. Identify the problem or issue to be decided.
2. Brainstorm possible solutions.
3. Decide which solution to implement.
4. Implement the solution.
5. Evaluate the results.

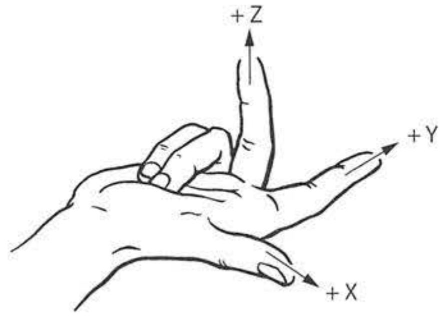
Having the ability to solve problems on the job shows an employer that you are able to handle more responsibility. Solving problems as a group can strengthen the team and help employees feel more pride in their work.

Making decisions and solving problems both require *critical thinking skills*. These are higher-level skills that enable you to think beyond the obvious. You learn to interpret information and make judgments. Supervisors appreciate employees who can analyze problems and think of workable solutions.



Goodheart-Willcox Publisher

Figure 21-18. Axes of machine tool movements. Spindle motion is assigned to the Z axis and therefore differs between vertical spindle and horizontal spindle machines.



IBM

Figure 21-19. Machine tool axes are specified according to the right-hand system of Cartesian coordinates. When the right hand is held as shown, the thumb, forefinger, and third finger point in the positive (+) directions of X, Y, and Z axes.

stored separately from the machine. These programs can then be uploaded using portable storage devices, such as USB memory devices. Sometimes machines are networked into a company's computer systems. In these cases, programs can be uploaded into the control through the computer network.

Most programs are written using the Cartesian coordinate system because defining the location of a given point relative to another point is easier in Cartesian coordinates. This is because distances between important points are usually provided on prints using dimensions that are based on the Cartesian coordinate system.

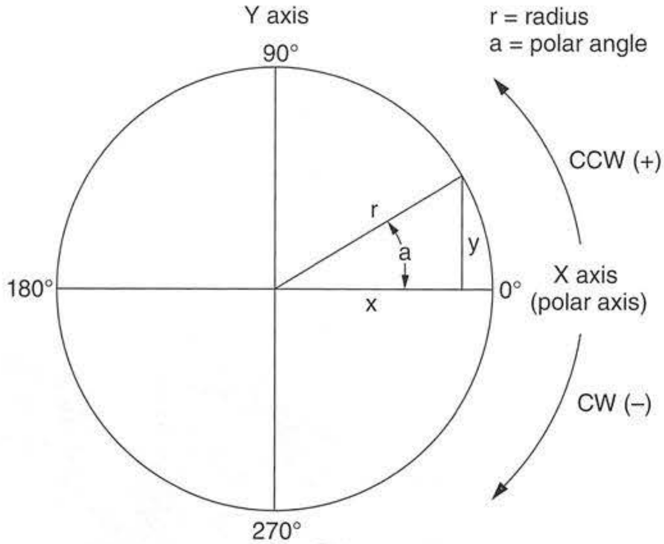
21.6.2 Polar Coordinate System

The ***polar coordinate system*** is used in CNC programs only when a straight-line distance and travel angle from a tool's current point location is known. An example of a perfect application for polar coordinate programming is a circular pattern of holes around a given point, **Figure 21-20**. The locations for the holes are defined using a distance from the center point of the circular pattern and an angle relative to an axis that is parallel to either the X or Y axis of the machine. Cartesian coordinates are not used in these cases because doing so would require the programmer to use trigonometry to convert the polar coordinate information into Cartesian coordinates. This would involve rounding multiple decimal values to the third or fourth place, which would cause errors in programming.

The polar coordinate system is not often used. However, when a part is defined with dimensions using the polar coordinate system, having the capability to create programs using this system reduces time and errors associated with the mathematical conversions that would otherwise be required.

21.7 CNC Movement Systems

CNC axes are positioned using electric motors. The two basic types of motors used to drive CNC axes are stepper motors and servomotors.



Goodheart-Willcox Publisher

Figure 21-20. Polar coordinates describe a new location from the present location (usually the center of a circle) by showing the value of the radius (r) and the polar angle (a). Counterclockwise rotation is positive, clockwise rotation is negative.

GREEN MACHINING

Efficiency in CNC Machining

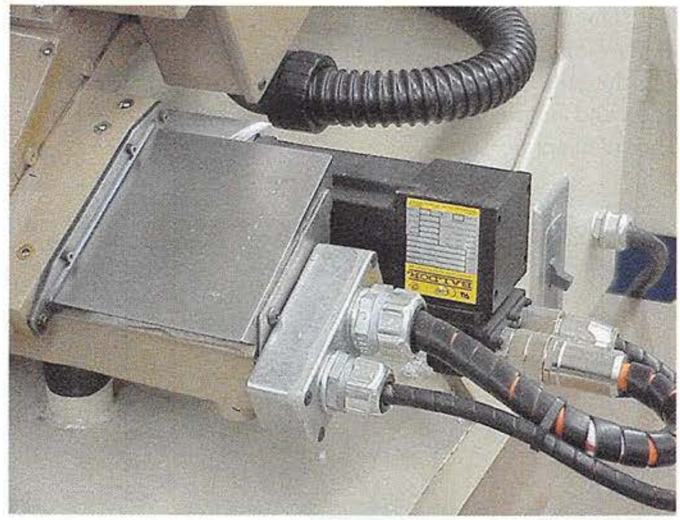
CNC machine shops can ensure energy- and cost-efficiency by streamlining shop operations and maintenance. High-efficiency practices include the following:

- Perform large work volumes on a complicated setup, rather than repeatedly resetting for smaller batches.
- Minimize maintenance costs through regular prevention and troubleshooting.
- Provide easy access to machines to reduce setup and breakdown time.

Special equipment, such as an induction motor with dual winding, can also be used to increase spindle efficiency. When a spindle is not turning in operation, shops can turn off CNC motors, fans, pumps, and conveyors to save energy. In addition, simple shop upkeep, such as adjusting the layout of milling equipment and maintaining a clean shop, can improve a shop's overall efficiency.

21.7.1 Stepper Motors

Stepper motors are often used in less expensive CNC machines, especially machines that are used for woodworking purposes, **Figure 21-21**. They do not require encoders



Goodheart-Willcox Publisher

Figure 21-21. This stepper motor controls the Z axis movement on a small benchtop turning center. Stepper motors are commonly used in open-loop systems.

to keep track of the motor's revolutions relative to the position of the axis they are driving. An ***encoder*** is a transducer that measures the position of moving axes and provides electronic feedback to the control.

Stepper motors are typically used in ***open-loop systems.*** An open-loop system has no means to provide feedback to the control relative to the position of the axis. This requires the machine to be reset to its zero position, often called *machine homing*, more frequently than a machine using servomotors. Another major drawback of stepper motors is that they lose torque at higher speeds, which means their machine cycle times must be reduced when cutting forces require higher torque.

21.7.2 Servomotors

Servomotors do not lose as much torque at high speeds as stepper motors, so machines equipped with servomotors can remove more material faster, **Figure 21-22**. However, because of their design, they require encoders to provide feedback to the control relative to the position of the axis they are driving. This is known as a ***closed-loop system.*** The use of encoders is critical when precision machining is needed.

21.7.3 Lead Screws and Lead Nuts

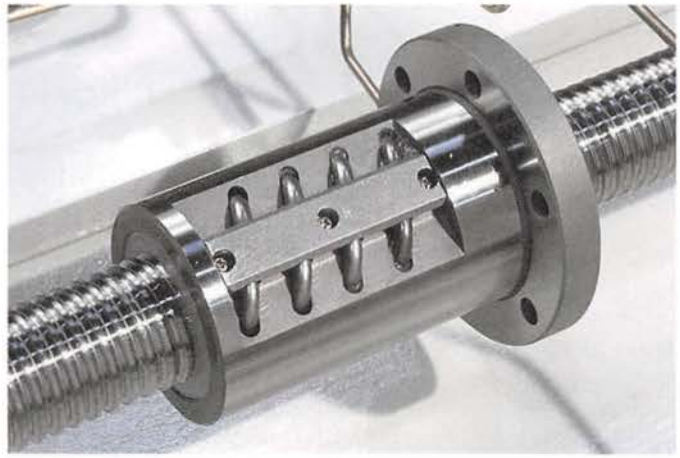
Both stepper and servomotors drive the axes of a machine using lead screws. ***Lead screws*** are designed to translate the rotating motion of an electric motor into straight-line, or linear, motion. As lead screws rotate, they cause a lead nut to move along the threads of the screw, **Figure 21-23**. Lead nuts are attached to the moving table of a milling machine or to the cross-slide of a lathe.

The threaded rods and nuts do not fit tightly together. If they did, you would not be able to turn one while holding the other. The clearance designed into the mating of the rods and nuts is called ***backlash***. Backlash is present in all threaded applications.

Goodheart-Willcox Publisher

Figure 21-22. This servomotor controls the Y axis movement on a horizontal machining center (HMC). Servomotors use a closed-loop system, requiring encoders to provide feedback to the control.

Minimizing backlash is critical in CNC machines to ensure precise positioning as the axis changes direction. Lead screws and lead nuts are precision-ground to exacting tolerances to minimize backlash. Encoders help filter out minor backlash but, as these components wear, they often need to be replaced. Worn lead screws and lead nuts result in too much clearance for the encoder to compensate.



Haas Automation, Inc.

Figure 21-23. A lead screw assembly consisting of a lead screw and a ball bearing nut. The lead screw translates the rotary motion of the motor into the linear motion that moves the nut.

Chapter Review

Summary

- Computer numerical control (CNC) is a computer-controlled version of the numerical control machines developed in the 1950s.
- CNC technology results in faster, more precise machining, but CNC machines typically cost much more than their manual counterparts.
- CNC milling machines are similar to traditional milling machines but have built-in CNC capabilities.
- Machining centers are CNC milling machines equipped with automatic tool changers.
- CNC turning machines consist of computer-controlled lathes, including basic CNC lathes, gang-tool lathes, turret lathes, and Swiss-type turning centers.
- All of the safety guidelines for manual machines should be followed for their CNC counterparts.
- CNC machines have additional safety guidelines related to the computer and the automation of the equipment.
- CNC programs use the Cartesian coordinate system and the polar coordinate system to communicate the direction and distance a workpiece or tool must move.
- CNC axes are driven by stepper motors or servomotors, depending on the speed and cost of the equipment.

Review Questions

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

1. What medium was used to store the programs that controlled machine tools in the original numerical control machines?
2. What technological advance in the 1960s was responsible for dramatically increasing the use of numerical control technology?
3. Name two advantages of using CNC equipment instead of manual methods.
4. What is the biggest disadvantage of using CNC?
5. How many axes of movement does a basic milling center have?
6. Basic CNC milling centers use a method called _____ that allows operators to select operations from menus without having to know a programming language.
7. Why might a milling center need more axes of rotation than are provided on a basic model?
8. Describe the difference between a basic CNC lathe and a gang-tool lathe.
9. Why might you avoid using a gang-tool lathe for parts that have a relatively large diameter?
10. What is the purpose of the turret on a turret lathe?
11. Why do operators of turret lathes require more training than those who operate basic CNC lathes?
12. Which type of CNC turning center would be used to machine small parts that have a high degree of elaborate detail?
13. When changing tools manually, wait for the machine axes and the _____ to come to a complete stop.
14. What is the purpose of a dry cycle?
15. In the _____ coordinate system, movement is specified in a positive or negative direction along the X, Y, or Z axis.
16. The _____ coordinate system specifies movement by giving a straight-line distance and travel angle from a tool's current point location.
17. A(n) _____ is a transducer that provides electronic feedback about the position of the moving axes to the control in a CNC machine.
18. What is the difference between an open-loop system and a closed-loop system?
19. Why must the lead screws and lead nuts in the motors that drive the CNC machine axes be checked and replaced at regular intervals?

CHAPTER **22**

CNC Programming Basics

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Explain the process of planning and developing a CNC program.
- Describe three methods of generating CNC code.
- Summarize the use of CAD and CAM software in CNC programming.
- Classify the types of codes used in the ANSI/EIA 274D code format.
- Identify commonly used CNC modal commands.

Technical Terms

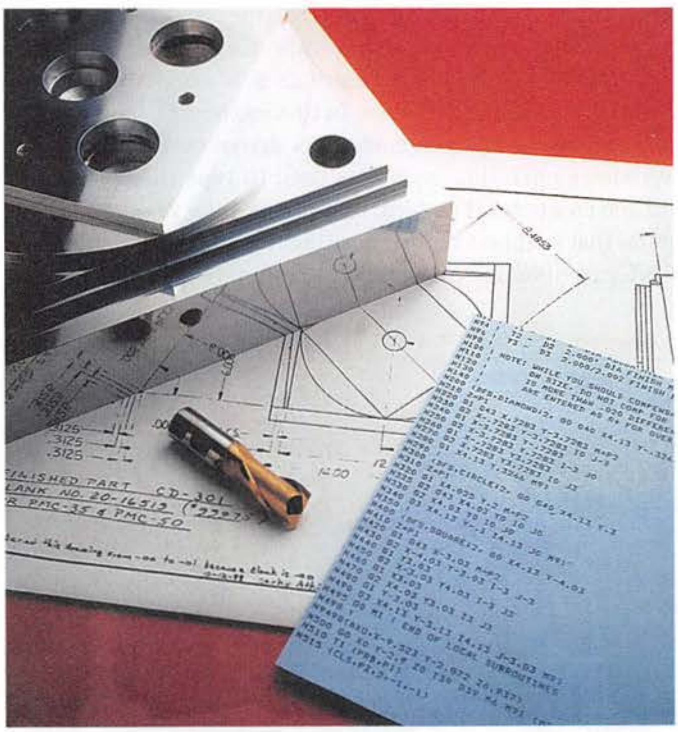
absolute positioning
assigned code
block
circular interpolation
G-codes
incremental positioning
linear interpolation
machine control unit (MCU)
manual data input (MDI)
M-codes
modal command
offline programming
post-processing
unassigned code
word
word address format

CNC programming methods have changed greatly since the days of paper tape storage. However, the programming language that commands the motion of the machine tools has remained virtually the same, **Figure 22-1**. Even so, the tasks of creating and uploading a CNC program have been greatly simplified. Advances in computing power and software have made it easier for operators to create CNC programs without requiring an in-depth knowledge of control codes. These advances have greatly benefited manufacturing because they enable machine shops to utilize CNC technologies accurately without requiring a long learning curve.

However, for safe machine operation, maximum efficiency, and effective troubleshooting, it is essential that programmers and operators understand the codes and their functions. Because most programs will have to be modified at least once during their run cycle, programmers and operators must be able to read and write a CNC program. This chapter covers the basics of CNC program creation, with an emphasis on language and code format.

22.1 Developing CNC Programs

In order to develop a valid CNC program to machine a part, the programmer needs a copy of a print of the part to use as a planning tool. In most cases, the company's quality control department can provide a copy that can be marked up as needed for planning purposes. Making unofficial photocopies of the print is unacceptable. All copies of the print should be officially controlled by the quality control department or its equivalent. Always ask permission before copying a print.



Giddings & Lewis, Inc.

Figure 22-1. A series of coded instructions, called a *program*, is used to control CNC machine tool movements and operations. From left to right: a part machined on a VMC, an end mill, the print of the part, and a printout of the CNC program.

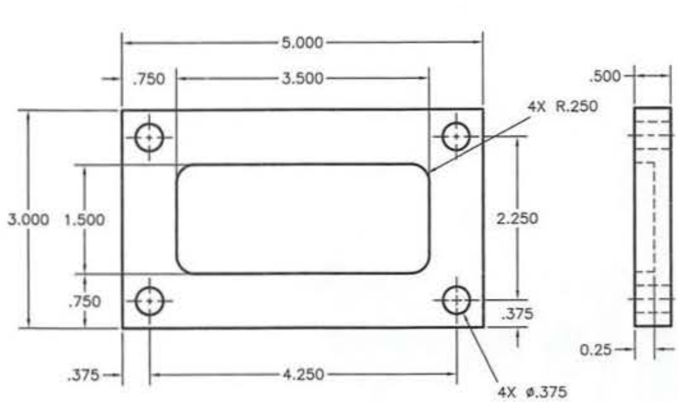
Figure 22-2 shows a part drawing that can be used to demonstrate how CNC programs are planned. After examining the print, the programmer determines the condition and dimensions of the stock to be used in the process. In this example, the plate to be machined is 6061 aluminum that has already been machined to the length, width, and thickness of the finished part.

The programmer then determines the necessary machining processes based on the print. In this case, the machining processes will consist of drilling four holes and milling the pocket in the middle of the plate. After the machining processes and tooling have been determined, the programmer calculates the feed rates and spindle speeds as described in earlier chapters of this textbook.

For this part, the two features (holes and pocket) will be machined using different processes and tools. Notice in **Figure 22-3** that the holes and the pocket of the part have been color-coded. The color codes can be made with colored pencils or markers or even highlighters. Color-coding allows the programmer to readily associate tooling and processes with the colors of the features on the drawing.

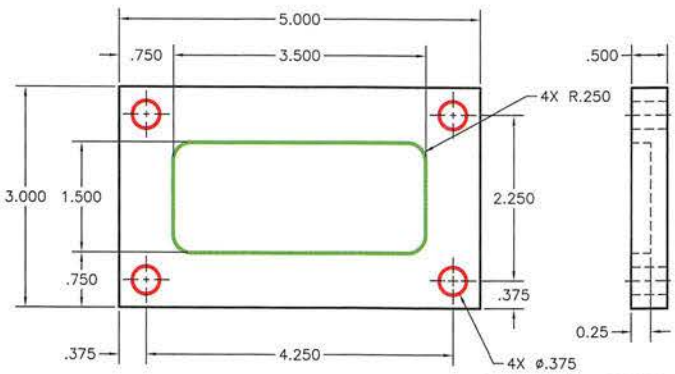
In this example, the holes will be drilled using a center drill and a twist drill. The pocket will be milled using a larger end mill to rough out the pocket and a 0.500″ diameter end mill to create the corners of the pocket, which are filleted at a radius of 0.250″.

At this point, the programmer can begin planning how the drilling and pocket milling processes should be executed. For this uncomplicated part, a prewritten subroutine can be used to machine the holes. The tool path for milling the pocket will have to be written manually, unless a CAM software package is used. Chapter 23, *CNC Milling*, describes the programming techniques for milling processes.



Goodheart-Willcox Publisher

Figure 22-2. A simple part drawing. This plate will be made of 6061 aluminum and machined from stock that has already been machined to the proper length, width, and thickness.



Goodheart-Willcox Publisher

Figure 22-3. The same part drawing from **Figure 22-2** with the holes color-coded red and the pocket color-coded green. Color-coding the print can assist the programmer in associating the tooling and processes with the features on the part while writing the CNC program.

22.2 Programming Methods

There are many methods available to generate CNC machine code today. These methods can be grouped into three distinct categories: manual programming, conversational language programming, and CAM programming.

22.2.1 Manual Programming

Manual data input (MDI) consists of entering the CNC program codes at the ***machine control unit (MCU)***, also simply called a control, **Figure 22-4**. The MCU reads the program from memory and translates it into the electronic signals needed for machine operation. MDI programming is primarily used for small programs with few lines or for inserting a small number of lines into an existing program.

Dmitry Kalinovsky/Shutterstock.com

Figure 22-4. A CNC machine operator using the manual data input (MDI) method of programming to enter a program into the machine control unit (MCU) of a machining center.

SAFETY NOTE

Great caution must be taken when programming using the MDI method to ensure that the code has been entered 100% correctly. Incorrect code could cause a spindle to crash into the workpiece or cause the machine to move in an unexpected direction, such as into an operator or bystander. An incorrect spindle speed could cause the tool and/or workpiece to break. Flying shrapnel can injure or kill bystanders.

Manual data input can be done in two ways. The first method developed was "MDI execute." In this method, the code is entered and executed one line at a time. This is a slow and tedious process that has to be repeated for each part. The improved method of manual data input is "MDI store." The entire code is entered, reviewed, and saved to the machine's on-board memory, and then the whole sequence is executed. Controls with machine graphics can verify the code graphically before any machine movements take place.

The disadvantage of MDI is that it may interfere with the processing of products scheduled for shipment. Some machines allow code for another process to be entered and stored while production parts are being machined. However, this may mean that two people are needed at the machine—one to enter the code and one to manage the production run. An alternative would be to have the operator enter code between machine cycles. However, this typically slows both programming and machining cycle times, and can introduce errors into the code, negatively impacting safety.

Offline programming, **Figure 22-5**, is a better method for manually creating CNC code. In this method, a plain text editor, such as Notepad in computers driven by the Microsoft™ Windows operating system, is used to type the code on an offline computer. The term *offline computer* refers to a computer that is not currently in use controlling the operation of a CNC machine. The program is then stored electronically until

Dmitry Kalinovsky/Shutterstock.com

Figure 22-5. A CNC machine operator using the offline programming method at a computer station to write a CNC program.

needed. The primary advantage of offline programming is it can be done without slowing production machining operations.

22.2.2 Conversational Language Programming

Conversational language programming has become a popular option on CNC machine controls over the last several years. Conversational language is a series of software codes that allow operators to enter information into the MCU without the tedious practice of typing the program using programming codes, **Figure 22-6**. Fewer mathematical calculations are needed, and information can be entered directly from the print. As the operator enters the information, the software converts it into the format needed by the CNC control to execute the program.

The advantages of this method of programming include the ease of inputting multiple programs in a job shop environment where several production runs of small quantities do not warrant the setup needed to generate code in an offline setting. Also, in-depth knowledge of programming is not required in conversational language programming. Unfortunately, the fact that the machine does the bulk of code generation has eroded some of the programming skills of CNC operators to the point that many now find it difficult to read code and make changes when they are required.

22.2.3 CAM Programming

In computer-aided manufacturing (CAM) programming, computer software is used to generate the CNC program. In CAM, the part can be created graphically in either a two-dimensional (2D) or three-dimensional (3D) format. After the part and all of its features have been defined, the programmer defines the various cutting tools and processes needed using the CAM software, **Figure 22-7**.

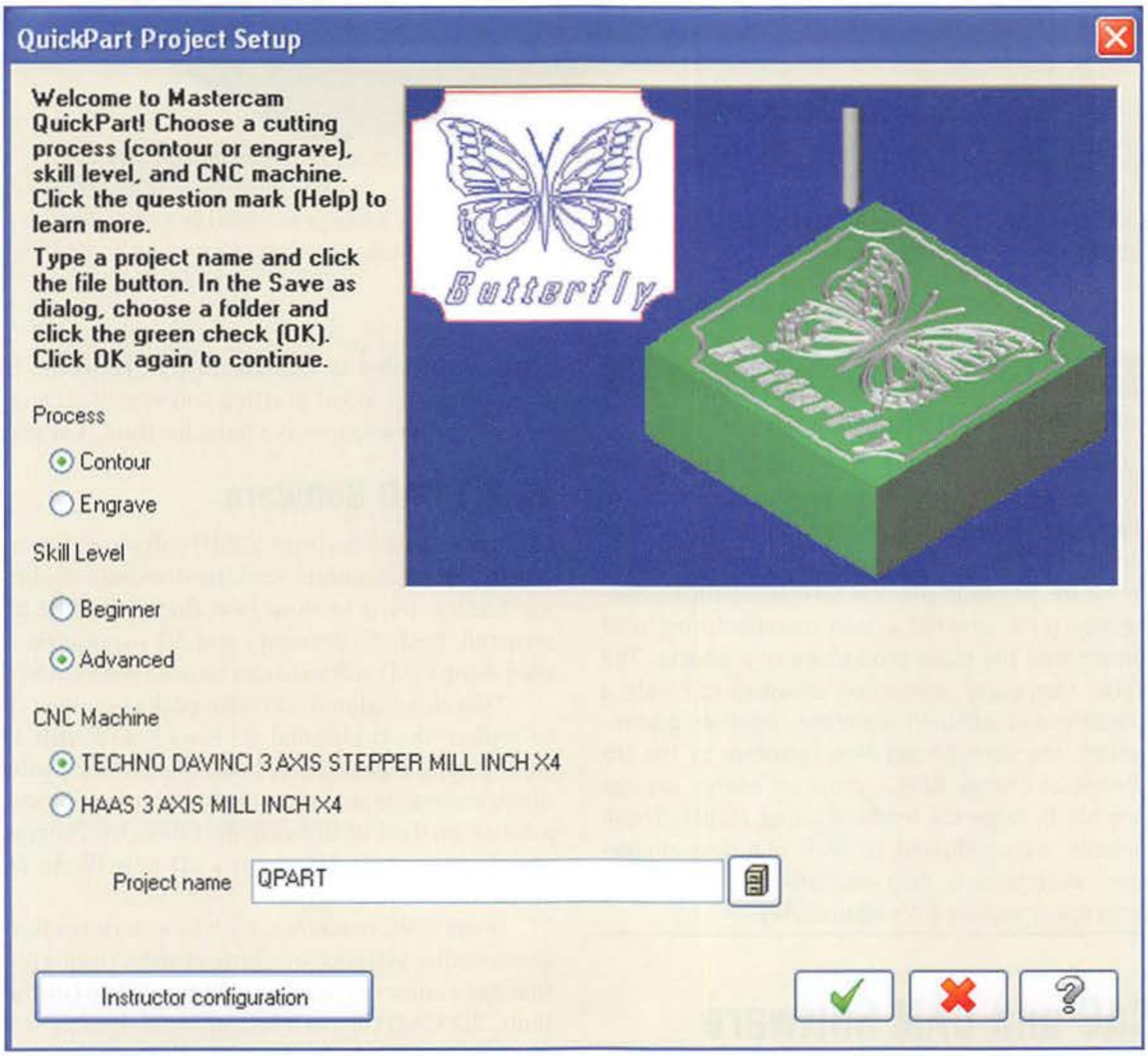


Figure 22-6. Conversational programming allows operators to machine a part by answering a series of questions and prompts instead of writing machine code. Mastercam QuickPart gives students a fast, easy way to design and cut parts while learning the principles of CAD/CAM.

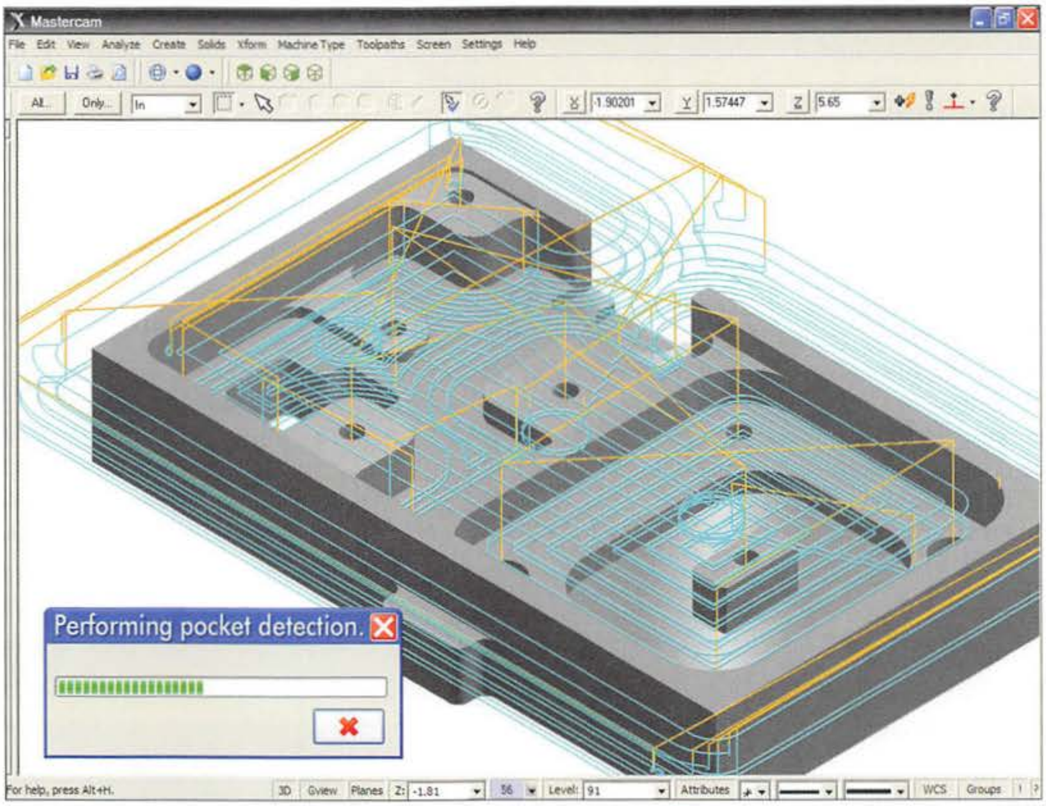


Figure 22-7. Three-dimensional CAD drawings can be imported into a CAM software package to generate a CNC program. Mastercam Feature Based Machining eliminates the manual processes involved in identifying features to program milling and drilling operations on solid parts.

GREEN MACHINING

Government Programs as Green Catalysts

Many government programs have established green initiatives to encourage more sustainable practices in manufacturing. These initiatives aim to reduce energy use, operating costs, and utility bills to promote a cleaner future. For example, the Environmental Protection Agency (EPA) created a lean manufacturing plan that streamlines the mass production of products. The EPA's plan rearranges production activities to create a more orderly and efficient workflow. Another government effort, the Save Energy Now initiative by the US Department of Energy (DOE), provides energy savings assessments to large US manufacturing plants. These assessments are conducted by DOE qualified energy-efficiency experts and help identify immediate and long-term opportunities for sustainability.

22.3 CAD and CAM Software

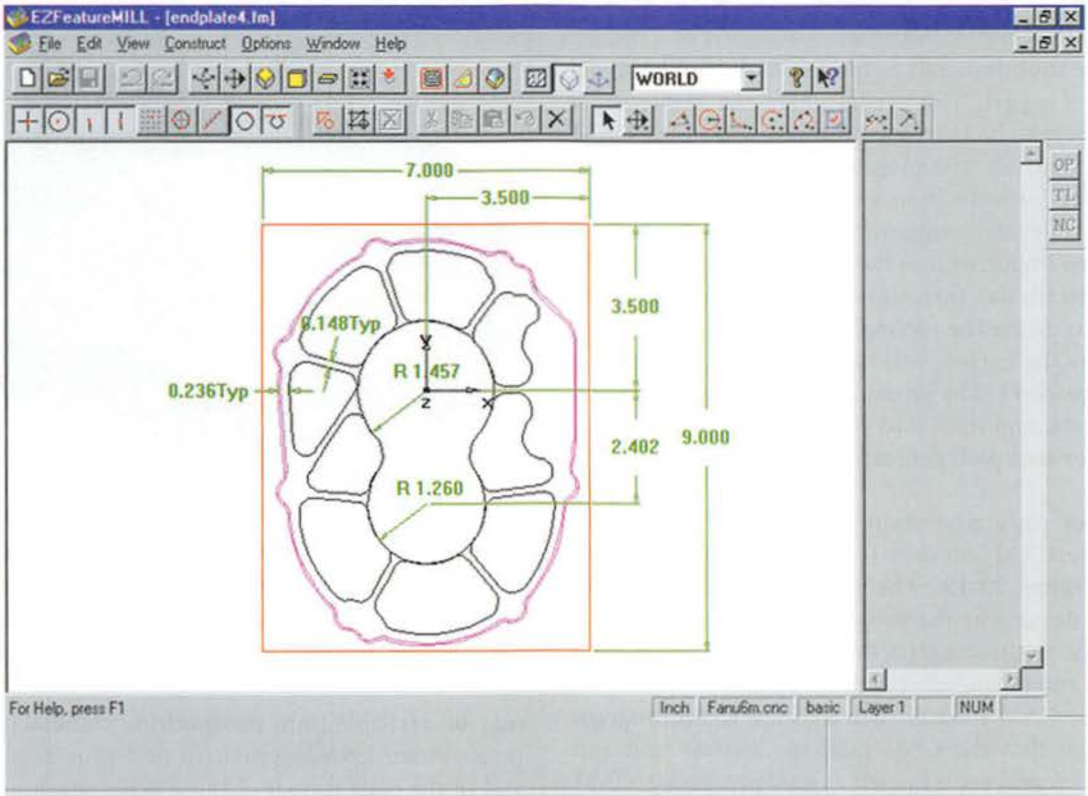
Two types of software are commonly used to support CNC programming. Computer-aided manufacturing (CAM) software is used to define the part and the tools and machining processes needed to manufacture it. However, information from computer-aided drafting software (CAD) is often used by CNC programmers as a basis for the CAM program.

22.3.1 CAD Software

Computer-aided drafting (CAD) software allows drafters to create precise, accurate working drawings of objects such as mechanical parts to show how they should be built or constructed. Both 2D drawings and 3D parametric models created using CAD software can be used with CAM technology.

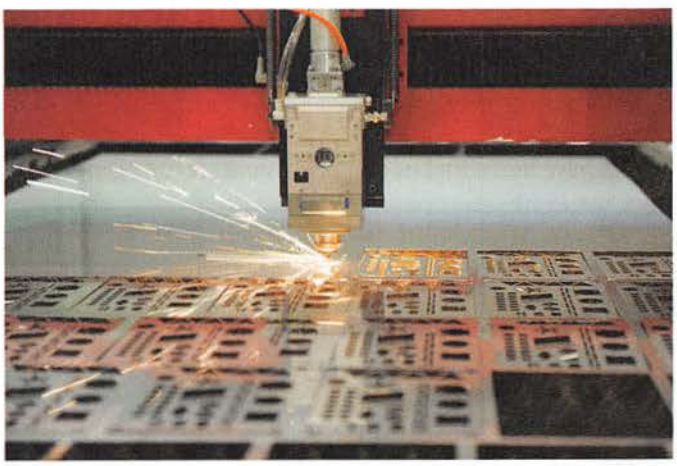
Two-dimensional software packages allow the designer to replace the traditional drafting board with a computerized system, **Figure 22-8**. However, 2D CAD software offers much more potential to the manufacturing process than just a faster method of drawing part designs. Programmers can use the electronic data from a 2D part file to facilitate the creation of CNC programs.

Some CNC machines, such as waterjet cutters and some laser cutting systems, machine profiles from a piece of stock that has a constant thickness, **Figure 22-9**. For these applications, 2D CAD files can be imported directly into the CNC machine's control. To complete the program, the operator needs only to program into the control information regarding material type and thickness, along with a few other commands needed to operate the machine. However, if the design



EZFeatureMILL—Engineering Geometry Systems

Figure 22-8. With some CAD software, the 2D geometry of the part can be created at the machine tool or imported from files. The software defines the features of the part and can generate the code necessary to machine the part.

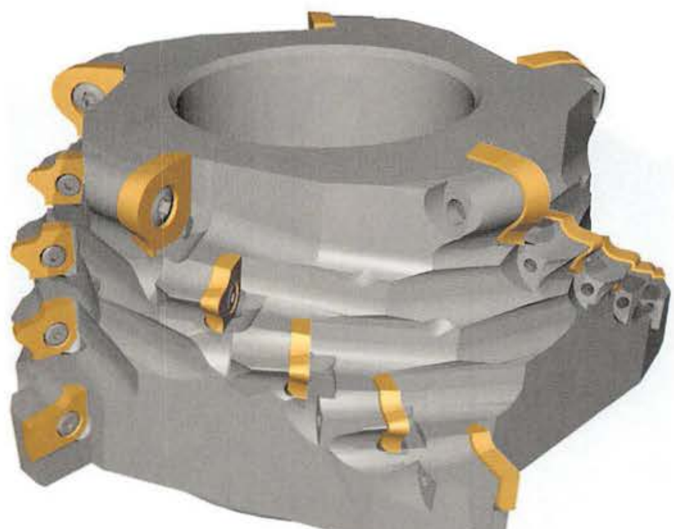


hxdbzxy/Shutterstock.com

Figure 22-9. A CNC laser cutting system cutting 2D shapes from a steel plate.

is more complex and cannot be machined from a piece of flat stock, 3D models are needed.

Created using solid modeling CAD software, 3D parametric models are truly three-dimensional in their design, **Figure 22-10**. Using 3D CAD software, the operator can define materials and features with precise detail. Most parametric modeling CAD programs are also capable of mass properties calculations and various mechanical analyses of parts. These models can be imported into the CAM software and used directly as a basis for the CNC programming.



Lovejoy Tool Company, Inc.

Figure 22-10. A 3D CAD model of an indexable-insert slab mill. The carbide inserts and bolts have also been modeled. Before this model is imported into the CAM software to create a CNC program, the inserts and bolts will be removed.

22.3.2 CAM Software

CAM software uses drawings or models to define the shape and features of a part. The programmer can create a 3D model of the part to be machined directly within the CAM software. Alternatively, the programmer can import 2D or 3D CAD files to form the basis for the programming. If 2D CAD files are used, the programmer adds depth to the part after it has been imported into the CAM software.

CAM software has three functions. First, it allows the programmer to define the tooling and process steps to generate the paths the cutters will take during the machining process, **Figure 22-11**. The programmer supplies the tooling and process data, and the CAM software reads the geometries needed for tool path generation from the 3D model of the part.

When tool path generation is complete, the second function of the CAM software is to simulate the program graphically, **Figure 22-12**. The programmer can include work-holding devices in the program along with the part being machined so that clearance moves around clamps and fixture details can be included in the program.

Finally, the CAM software converts the CAM program into a language that the CNC machine control unit can understand. This process is known as ***post-processing***. CAM systems come with post-processing programs for most of the machines in use today. However, on occasion, the post-processor needed for a specific machine may not be included with the CAM software. When this occurs, post-processors may be available from the machine manufacturer. If not, a programmer knowledgeable in post-processor modification and in the code format of the specific machine will need to write the post-processing program or edit an existing program to work with the specific machine.

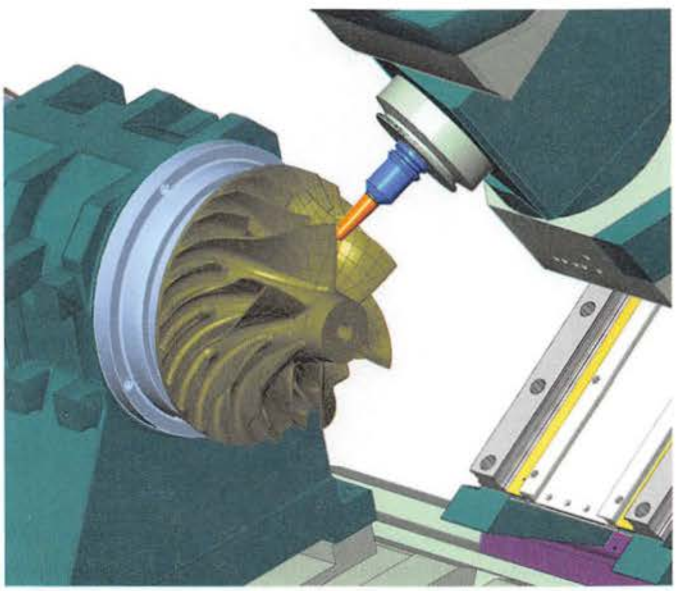


Figure 22-11. CAM software calculates the tool paths for the machining operations and can include specialty applications for specific machining operations. The Mastercam Blade Expert add-on is a powerful and easy-to-use custom application, designed to generate efficient, smooth, and gouge-free tool paths for complex multi-bladed parts.

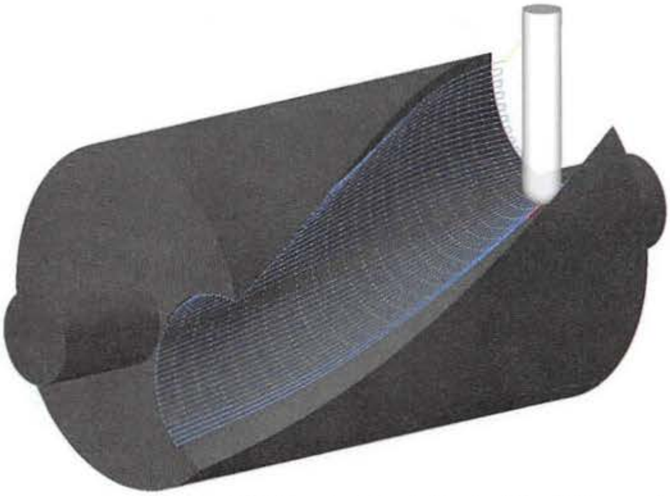


Figure 22-12. CAM software can simulate the CNC program graphically. Cut patterns guide the tool along specified paths. Mastercam's multiaxis machining has many tool path types to govern the cut pattern.

22.4 CNC Programming Codes

The ANSI/EIA 274D code format was the first standard CNC code format. Commonly called *G-code* or *G- and M-code* because of the frequent use of these two letters, codes in this format have been used in numerical control code generation since the 1960s.

G-code uses a ***word address format*** style of programming. The word address format uses combinations of letters (A to Z, except E and V) and numerical data. These combinations are called ***words***. The letters precede the numbers and define the purpose of the numerical data. Each word gives the machine a single instruction. Words that are to be executed together are written on a single line of code called a ***block***.

22.4.1 G-Codes

G-codes are preparatory codes. They control the positioning of the machining cutter relative to several references:

- The path the cutter takes.
- Offsets from a defined path.
- Temporary reference coordinates.

G-codes also include other information used to control the relationship of the cutter to the workpiece. **Figure 22-13** contains a list of the various G-codes used in CNC programming.

G-Codes—Preparatory Functions	
Code	**Function**
G00	Rapid traverse.
G01	Linear interpolation.
G02	CW (clockwise) circular interpolation.
G03	CCW (counterclockwise) circular interpolation.
G04	Dwell (timed delay of established duration). Length is expressed in X or F word.
G20	Inch programming.
G21	Metric programming.
G33	Constant pitch thread cutting.
G81	Simple drilling cycle.
G90	Absolute coordinates.
G91	Incremental coordinates.

Goodheart-Willcox Publisher

Figure 22-13. Examples of CNC programming G-codes. G-codes are preparatory codes and are standardized.

22.4.2 M-Codes

The miscellaneous codes that control machine functions, such as starting and stopping spindle rotation and turning coolant flow on and off, are known as ***M-codes***. The numbers associated with many G- and M-codes are standardized. They perform the same function for all machines that use the ANSI/EIA 274D format. However, several numeric values are not assigned to any particular task or command. This gives individual machine manufacturers the flexibility to use these codes for specialized functions that are specific to their machines. **Figure 22-14** lists the various M-codes used in CNC programming.

22.4.3 Assigned and Unassigned Codes

Most G- and M-codes are assigned codes. ***Assigned codes*** are codes that have the same application no matter what machine control is being used and no matter what type of machining process is being performed. For example, G00 is rapid travel, no matter whether you are programming a milling machine or a lathe. Likewise, M00 is a program stop for both applications.

Unassigned codes are codes that can be used by different types of machine controls to perform tasks for specialty

CAREER CONNECTION

CNC Programmer/Technician

What does a CNC programmer/technician do?

The work of CNC programmers and technicians is very close to that of machinists and tool and die makers. They write and program instruction codes into machines that automate machining processes, such as drilling, grinding, milling, sawing, and turning. The machine then creates a precisely cut part based on the programmed instructions. Although the computer-controlled machine produces the actual cut, the technician sets the path, speed, and feed rate for the job. CNC programmers may write programs to produce parts in batches or for custom use.

What education and skills are needed to be a CNC programmer/technician?

Employers prefer candidates with high school diplomas, previous experience in computer programming, and a strong background in math and science. Other skills relevant to CNC programming work are print reading, drafting, and a general knowledge of metals and plastics. Students interested in professional advancement can also complete apprenticeships or earn associate's degrees at community colleges or technical schools.

What is it like to be a CNC programmer/technician?

Like other machining professions, CNC programmers often work where the machines are located. They are often exposed to the same environmental hazards as other employees on a manufacturing floor, including loud noises, hazardous chemicals, and debris flying from machines.

According to the *Occupational Outlook Handbook*, the median annual wage for CNC programmers has been $48,900 in recent years. They are among the highest paid production workers in their industries.

M-Codes—Miscellaneous Functions	
Code	**Function**
M00	Stop machine until operator restart.
M02	End of program.
M03	Start spindle—CW.
M04	Start spindle—CCW.
M05	Stop spindle.
M06	Tool change.
M07	Coolant on (mist).
M08	Coolant on (flood).
M09	Coolant off.
M30	End program and return to program top.
M52	Advance spindle.
M53	Retract spindle.
M56	Tool inhibit.

Goodheart-Willcox Publisher

Figure 22-14. Examples of standardized CNC programming M-codes. M-codes are miscellaneous codes.

purposes. Every machine's manual includes a table showing how these codes are used for the particular machine.

22.4.4 Other Letter Codes

Other letters are used in generating CNC codes as well. For instance, when the control reads the letter F, it uses the numbers immediately after the letter to set the programmed feed rate for the movement of the machine axes. The letter S signals the control that the numbers immediately after the letter determine the rpm of the spindle. **Figure 22-15** contains a list of these letters and describes their purpose in programming.

22.5 CNC Modal Commands

Modal commands are those G-code commands that, once activated, remain activated until canceled or another modal command is activated. Many of the G-code commands are modal. This section describes the modal commands most commonly used in CNC programming.

22.5.1 G00—Rapid Travel

The G00 command activates the rapid travel setting for all of the positioning axes on the machine. When this command is activated, all axes, when called by the program to move, will move at the fastest rate the machine is capable of moving. This command is for positioning moves, quickly moving the cutter to or away from the areas to be machined. The G00 command enables the CNC program to cycle faster, allowing more parts to be made per time period.

SAFETY NOTE

Great care must be taken with the G00 command. Because it is modal, it remains active until another G-code requiring a different programmed feed rate is read by the control. Also, rapid travel commands do not take the shortest path to the next position, instead moving at the maximum feed rate for each axis independently. This may result in unexpected paths of travel. Mistakes with this command can cause severe damage to the tool, the workpiece, the CNC machine, and possibly the operator.

22.5.2 G01—Linear Interpolation

In CNC programming, ***linear interpolation*** means straight-line movement. The G01 command specifies a linear move to a specified location, **Figure 22-16**. G01 is one of the positioning codes that requires a programmed feed rate. A sample line of code containing a linear interpolation command is shown below:

```
N0010   G01   X0.500   Y0.500   F30
```

Where:

- N0010 addresses the line of code.
- G01 is the call for a linear interpolation move.
- X0.500 is the X axis coordinate for the target location.
- Y0.500 is the Y axis coordinate for the target location.
- F30 is the feed rate at which the machine will move the cutter to the target point.

When this line of code is executed by the control, the machine axes move from their current position to the 0.500,0.500 coordinate in a straight-line motion, at a rate of 30 units per minute. The units may be either inches or millimeters, depending on which unit was specified earlier in the program. The codes for specifying units are described later in this chapter.

Some machines require that all lines of code be addressed in order. Other machines ignore the N sequence number and do not require their use. However, N numbers assist the operator and programmer with troubleshooting and are recommended for most applications.

If the line of code specifying linear interpolation does not have a programmed feed rate, one of two things will happen. The machine will move at a previously programmed feed rate, or it will not move at all and the control screen will signal that a programming error has occurred. Most controls have a registry where information for parameters, such as

Other Letter Codes	
Code	**Function**
A	Designates rotation about the X axis.
B	Designates rotation about the Y axis.
C	Designates rotation about the Z axis.
D	Specifies cutter radius compensation offset.
F	Specifies a feed rate.
H	Specifies tool length compensation offset.
I	Specifies the location of the arc center for a circular move. The I refers to the distance and direction in the X axis. Also specifies the amount of tool "back away" at the bottom of a fine boring cycle.
J	Specifies the location of the arc center for a circular move. The J refers to the distance and direction in the Y axis.
K	Specifies the location of the arc center for a circular move. The K refers to the distance and direction in the Z axis.
L	Used with subroutines to specify the number of times the subroutine should be repeated.
N	Designates a sequence number, line number, or block number in a program.
O	Identifies a program.
P	Specifies the length of time for dwell commands.
Q	Specifies the peck depth for each pass in a peck drilling canned cycle.
R	Designates the radius value in a circular movement. Also specifies the rapid plane height for a canned cycle command.
S	Used to program a spindle speed.
T	Used to select a tool station.
U	Replaces the letter X when doing incremental moves on a turning center.
W	Replaces the letter Y when doing incremental moves on a turning center.
X	Identifies a coordinate position along the X axis.
Y	Identifies a coordinate position along the Y axis.
Z	Identifies a coordinate position along the Z axis.

Goodheart-Willcox Publisher

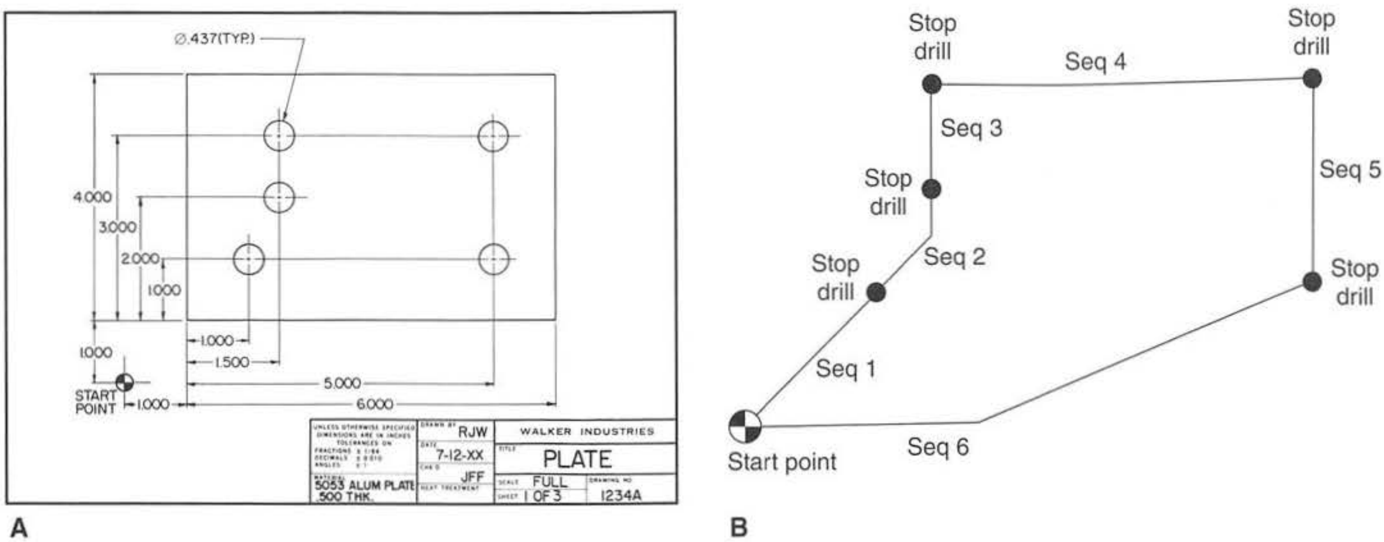
Figure 22-15. Other letter codes used in CNC programming.

feed rate, is stored. The feed rate is pulled from this registry if no reference is found on the line of code.

If a line of code fails to provide a feed rate and the control uses a previous feed rate, the rate may have been programmed for a different tool performing a different machining step. That feed rate may not be suitable for the tool and machining step currently being performed. The controls of some CNC machines hold parameters in their registries from previous programs for other parts. This too could result in a feed rate that is not appropriate for the task at hand. Therefore, it is a good practice to include a feed rate specification on every line of code that calls for a linear or circular interpolation move. This ensures that no incorrect feed rates will be pulled from the control's registry.

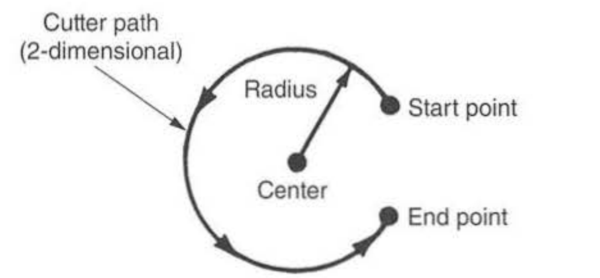
22.5.3 G02 and G03—Circular Interpolation

Just as linear interpolation refers to straight-line movement, ***circular interpolation*** refers to movement in a circular or



Goodheart-Willcox Publisher

Figure 22-16. Linear interpolation is straight-line movement. A—A simple part drawing requiring linear interpolation commands. B—The planned drilling sequence for the above part. Linear interpolation is required to move the drill to the proper locations.



Goodheart-Willcox Publisher

Figure 22-17. When using circular interpolation, only the coordinate location of the start and end points of the arc, the radius of the circle, the coordinate location of the circle's center, and the direction of the cut need to be programmed. The intermediate points required to describe the circular movement are computed by the MCU. On some machines the circle is divided into 90° segments and must be programmed accordingly.

radial pattern, **Figure 22-17**. The G02 command signals the control to drive the axes to move the cutter in a clockwise direction, and the G03 command specifies a counterclockwise direction.

When visualizing the direction of movement, imagine looking down on the machine from the ceiling, **Figure 22-18**. Whether the machine moves the workpiece around a stationary tool, the tool around a stationary workpiece, or any other combination of axis movements, you must visualize the cutter moving around a stationary workpiece. This is how the machine will interpret your program.

A sample line of code containing a G02 circular interpolation command is shown below:

```
N0010   G02   X0.500   Y0.500   R+0.500   F30
```

Where:

- N0010 addresses the line of code.
- G02 is the call for a clockwise circular interpolation move.
- X0.500 is the X axis coordinate for the target location.
- Y0.500 is the Y axis coordinate for the target location.
- R+0.500 is the radius of the circular movement.
- F30 is the feed rate the machine will take to move the cutter to the target point.

Note the plus sign between the letter R and the 0.500 value. This "R address" specifies the location of the center point of the arc. In a circular movement from one point to another point, the center point of the arc can be in one of two places, depending on whether the arc motion is greater than or less than 180°. If the arc to be machined is less than 180°, the plus sign is used. If the arc to be machined is greater than 180°, a minus sign is used. If no sign is placed after the letter R, most controls interpret the arc angle to be less than 180°.

Another method of specifying the center point for the arc to be machined is to use I and J codes instead of the R address, **Figure 22-19**. An example of this method is shown below:

```
N0010  G03  X0.500  Y0.500  I0.000  J0.500  F30
```

In this example, the G02 from the previous example has been replaced with a G03 command, so this arc will be machined in a counterclockwise rotation. The R+0.500 address has been replaced with the code I0.000 J0.500. The I and J values provide the location of the center point of the arc. The letter I specifies distance along the X axis, and the letter J specifies distance along the Y axis. The numeric

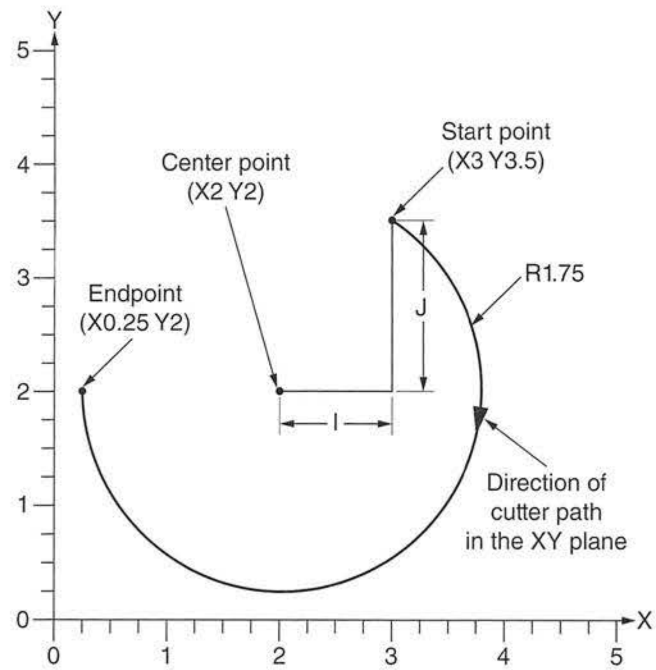


Figure 22-18. Interpret and specify all motion in the XY plane as though you were looking down from the ceiling, directly above the workpiece.

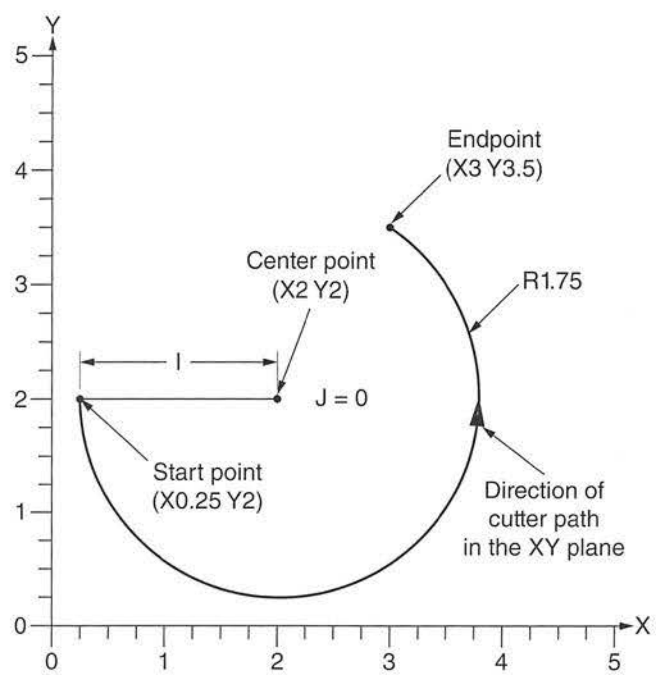


Figure 22-19. The G03 command needs the following information: center point, start point, endpoint, radius, and direction in the XY plane.

values are incremental distances from the point where the machine is currently positioned to the center point of the arc. In this example, the center point of the arc is 0.500 units away from the start point in the Y axis direction. Since the number associated with I is 0.000, there is no distance along the X axis between the start point and the center point. If either I or J is not written into the line of code, most controls assume the value for the omitted address is zero.

As with the G01 command, the G02 and G03 commands require a programmed feed rate. If the rate is not included on the line of code with the circular interpolation callout, the control pulls the last feed rate in the registry. If the registry is empty, the control stops machine movement and displays an error on the control screen.

Both G02 and G03 are modal commands. Once they have been activated, all movements following the activation will be circular motions until they are canceled.

22.5.4 G20 and G21—Units

Another pair of modal commands are the G20 and G21 commands, which set the working units in the control. The standard units recognized by most CNC machines are decimal inches and millimeters. G20 sets the working units in the control to inches, and G21 sets the units to millimeters. If this command is not present in the program before the first command that causes axis movement, the machine will do one of two things when it makes its first move. It will either use the default units specified in the machine control, or it will pull the units used for the previous program. Either of these actions may result in incorrect machining. It is therefore a good practice to include the appropriate code in one of the first lines of a program.

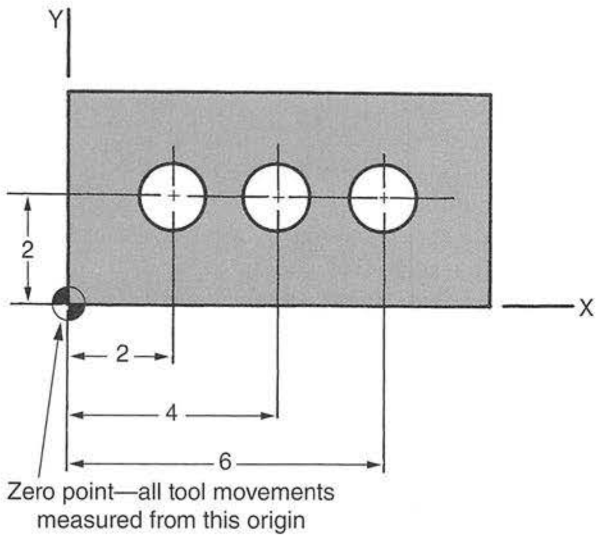
22.5.5 G90 and G91—Absolute and Incremental Positioning

The G90 and G91 modal commands determine the relationship between points in a program based on their coordinate values. Recall from Chapter 21, *Introduction to CNC Machining*, that most programs are written using the X, Y, and Z axes of the Cartesian coordinate system. The X axis specifies table movement to the right and left. The Y axis moves the table forward and backward, or toward and away from an operator standing in front of the machine. Finally, Z axis movement raises and lowers the machining spindle.

The G90 command indicates to the control that absolute positioning is active. In ***absolute positioning***, the coordinate values for any point are interpreted in relation to the X0,Y0 (X = 0, Y = 0) position, **Figure 22-20**. For instance, in **Figure 22-21**, the starting point for the R60 arc is 200 units along the X axis and 40 units along the Y axis, relative to the X0,Y0 position. The ending point for the arc is at X=140 units and Y=100 units. Assuming that the cutter is currently positioned at the starting point of the arc, the line of code to move the cutter to the endpoint of the arc is:

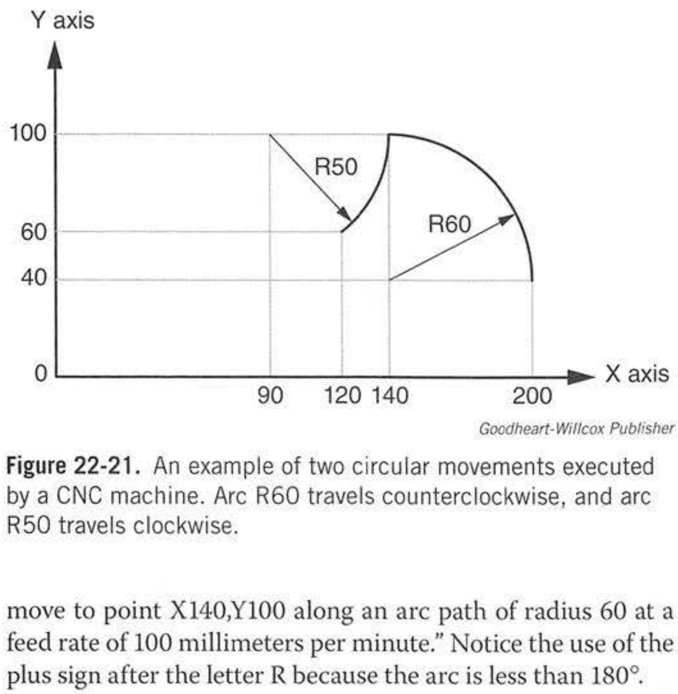
```
N0010  G03  G90  X140  Y100  R+60  F100
```

This command can be read as follows: "Moving in a counterclockwise circular direction using absolute positioning,



Goodheart-Willcox Publisher

Figure 22-20. In the absolute positioning system, all coordinates are measured from a fixed point of origin. The coordinate values for any point are interpreted in relation to the X0,Y0 (X = 0, Y = 0) position.



Goodheart-Willcox Publisher

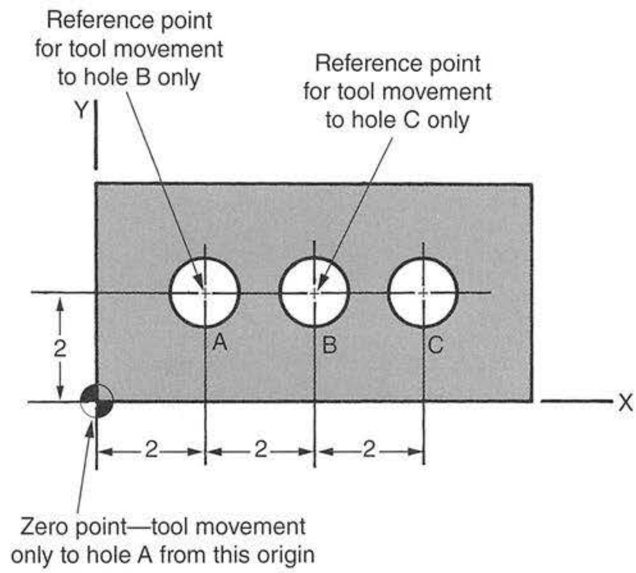
Figure 22-21. An example of two circular movements executed by a CNC machine. Arc R60 travels counterclockwise, and arc R50 travels clockwise.

move to point X140,Y100 along an arc path of radius 60 at a feed rate of 100 millimeters per minute." Notice the use of the plus sign after the letter R because the arc is less than 180°.

Incremental positioning, specified by the G91 command, does not refer to the X0,Y0 position. Rather, it is based on the location of the previous point, **Figure 22-22**. For example, for the same R60 arc shown in **Figure 22-21**, with incremental positioning, the line of code might be:

```
N0010   G03   G91   X–60   Y60   R+60   F100
```

Note the changes in the X and Y coordinate values. The difference between the X200 coordinate and X140 coordinate



Goodheart-Willcox Publisher

Figure 22-22. The incremental positioning system. Each set of coordinates is based on the location of the previous point.

is 60 units. The minus sign indicates that movement is in the negative X direction. Similarly, the difference between the Y100 and Y40 coordinates is 60 units. Because the movement is in the positive Y direction, the value is unsigned.

Programs are not commonly written using incremental positioning because doing so increases the chance that damage could occur as a result of accidentally leaving out negative signs or making mistakes on coordinate values. Another disadvantage of incremental positioning relates to program modification. In the event that a point coordinate defined by incremental positioning must be changed, the coordinates of every point after the changed point must be changed as well, or they will be incorrect. However, incremental positioning is an ideal method in some situations. The best example of a good use for incremental positioning is within subroutines, which will be described in Chapter 23, *CNC Milling*.

22.5.6 Setting the Modal Commands

Although it is not strictly necessary, it is a good CNC programming practice to establish the modal command setting in the first line of CNC code. If you have a control that stores parameters for units and positioning from previous programs in its registries, writing a simple line of code, such as the one below, clears those registries and prepares the new program for operation.

```
N0010   G00   G20   G90
```

This line of code overrides any previous settings and sets the machine to operate at rapid travel, sets the units to inches, and sets the positioning to absolute.

Chapter Review

Summary

- A programmer plans and develops CNC programs by reviewing the print for the part to be created, determining the tools and processes needed to create the part, and planning how the processes should be executed.
- CNC code can be generated by entering the individual codes manually, using conversational language, or using computer-aided manufacturing software.
- Computer-aided manufacturing software allows programmers to create (or input from a CAD file) a 2D or 3D model of a part to be machined and specify the tools and processes needed to machine it.
- Sets of codes in the ANSI/EIA 274D code format, commonly called G-code, are prefaced by letters, such as G and M, that allow the machine control unit to identify the type of command to be executed.
- Commonly used modal commands include those that specify rapid travel, linear and circular movement, units of measurement, and the type of positioning to be used to machine the part.

Review Questions

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

1. Why should all CNC programmers and operators have at least a basic understanding of CNC code formats?
2. What is the first thing a programmer needs in order to develop a CNC program?
3. After a programmer has determined the machining processes and _____ to be used to machine a part, the feed rates and spindle speeds can be calculated.
4. In _____ CNC programming, the programmer types the coded language in a format that can be read directly by the machine control unit.
5. Why is offline programming the preferred method of manual programming?
6. Using a series of software codes to enter information into the CNC control without using programming codes is commonly known as _____ programming.
7. In which type of programming does the programmer use computer software to define the part and the tools and processes needed to machine it?
8. What two types of software are commonly used to support CNC programming?
9. Which type of software allows drafters to create precise, accurate working drawings of objects such as mechanical parts to show how they should be built or constructed?
10. List the three basic functions of CAM software.
11. Converting a CAM program into a format that the CNC machine control unit can understand is known as _____.
12. The first standard CNC code format was the _____ code format, commonly called G-code.
13. In CNC programming, a group of codes called _____ control the positioning of the CNC machining cutter.
14. The miscellaneous codes that control machine functions are known as _____.
15. What do the letters F and S signify in CNC programming codes?
16. In what way are modal commands different from other CNC commands?
17. Which modal command activates the rapid travel setting for the positioning axes?
18. What will happen if a line of code that specifies linear interpolation does not have a programmed feed rate?
19. Explain the difference between the G02 and the G03 commands.
20. Which command would you use to set the units for a machining process to decimal inches?
21. In _____ positioning, the coordinate values for any point are interpreted in relation to the X0,Y0 position of the Cartesian coordinate system.
22. Which command would you use to specify that the coordinates provided should be read from the location of the previous point, rather than the X0,Y0 position?
23. What line of code would you use near the beginning of a CNC program to clear the control's registries and set up the machine for rapid travel, units in millimeters, and incremental positioning? For this question, assume that the address for the line of code is N0010.

CHAPTER 23

CNC Milling

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Identify the purpose of common miscellaneous function codes.
- Describe work-holding devices that are commonly used in CNC machining processes.
- List information the programmer needs to gather before beginning to write a CNC program.
- Describe the codes used to set up the machine properly at the beginning of a CNC program.
- Carry out the procedure for writing a CNC milling program.

Technical Terms

canned cycle
comment code
machine home
peck drilling
program zero position
subroutine

In ANSI/EIA 274D code format, completing a CNC program requires both G-codes and M-codes. Chapter 22, *CNC Programming Basics,* explained G-codes and described the functions of several modal G-codes. This chapter builds on that information by introducing M-code functions and describing work-holding devices for CNC machining. The chapter also includes a step-by-step example of the process of creating a CNC program.

23.1 Miscellaneous Function Codes

As you may recall, M-codes are miscellaneous codes that control specific machine functions. This section defines some of the more frequently used M-codes.

23.1.1 M00—Program Stop

The M00 command causes the machine cycle to stop. When the control encounters this command, all movements, including spindle rotation, table feed movement, and coolant flow, immediately come to a halt.

This command can be used for many applications. If the machine is not equipped with an automatic tool changer, then the operator is responsible for changing tools. The M00 command can be used to stop the machine to allow the operator to change tools. After the tool has been changed, the operator resumes the program by pressing the cycle start button on the control. Other tasks, such as a part or tool inspection or a chip clearing operation, can also be accomplished after stopping the machine with the M00 command, **Figure 23-1**.

Dmitry Kalinovsky/Shutterstock.com

Figure 23-1. The M00 and M01 codes can be used to stop the CNC machine, allowing the operator to make measurements, clear chips, change tools, or inspect the part, tools, or work-holding device. The CNC operation is resumed by pressing the cycle start button.

Note that when the cycle is restarted, the machine does not automatically resume the operations that were stopped by the M00 command. A line of code is necessary to restart the spindle and coolant flow before the machine axes move to begin the next step in the machining processes.

23.1.2 M01—Optional Program Stop

The M01 command allows the programmer to include an optional stop in the machine cycle. Most CNC machines are equipped with an optional stop button or toggle switch. If the button or toggle switch is set to the "on" position, the machine stops when the control reads an M01 command just as it would when reading an M00 command. If the button or toggle switch is set to the "off" position, the machine ignores the M01 command and continues to the next line of code.

The M01 is useful during the initial setup of an operation and for periodic inspections. The command can be used after each tool has finished cutting, allowing the operator to inspect the part after each operation. If everything is running satisfactorily, the toggle switch or button can be set to the "off" position to ignore the M01 code in future cycles.

23.1.3 M02 and M30—Program End

The M02 code is one of two options to signal the machine that the end of the program has been reached. On reading this command, the control stops all machine functions, and the machine signals the operator, through either a program end light or a message on the control screen, that the program has finished its cycle. The M02 command does not cause the cycle to repeat automatically.

Like the M02 code, the M30 code signals the end of the program. Unlike M02, however, M30 also resets the program back to the first line of code so the program can be repeated. For this reason, most programmers use M30 instead of M02. Machines that were equipped with paper tape readers required this code to physically rewind the tape to the beginning of the program. Both codes have specific uses. The M02 code may be used in a machine shop where only one or a few parts need to be machined. The M30 code is better suited to the needs of mass production applications because the operator does not need to manually reset the program to the beginning after each cycle.

23.1.4 M03, M04, and M05—Spindle Operation

The M03 code turns the spindle on to rotate in a clockwise direction as viewed from above the machine. The M04 code also turns the spindle on, but it specifies counterclockwise rotation. The M05 code is the spindle stop code. It cancels both the M03 code and the M04 code.

As with the programmed feed rates discussed in Chapter 22, *CNC Programming Basics,* the spindle's speed must be specified in the line of code, as follows:

```
N0030   M03   S2500
```

The designation for spindle speed definition is the letter S, and the number following it defines the spindle speed. Therefore, the line of code above starts the spindle rotation in a clockwise direction (M03) at 2500 rpm (S2500).

It is considered best practice to include the spindle speed code on every line of code that contains the M03 or M04 command. If the code is not present, the machine function may stop and display an error code, or it may pull a previously programmed spindle speed from the registry. As with feed rates, spindle speeds pulled from the registry are not likely to be correct. Using an improper spindle speed can cause damage to the tool, part, machine, or operator.

Once the spindle has been turned on, it continues to rotate until it is canceled. In addition to M05, codes that cancel spindle rotation include M00, M01, M02, and M30. The M03 and M04 codes cancel each other out, because they are the same code with the exception of the direction of spindle rotation. However, it is unwise to use one of these codes to stop the other. Many spindles, such as high-speed pneumatic spindles, need to come to a complete stop before the command to change direction is activated. Activating the command to change the direction of spindle rotation before the rotation completely stops can damage the spindle.

23.1.5 M06—Tool Change

Another miscellaneous code that is commonly used with CNC machines equipped with automatic tool changers is the M06 code, which initiates a tool change. The M06 code requires an indication of the location of the next tool to be used. The letter T signals the control that the following number designates a position in the automatic tool change system, **Figure 23-2**. In the example below, the T2 defines the tool that is to be loaded based on the tool's location in the turret or magazine:

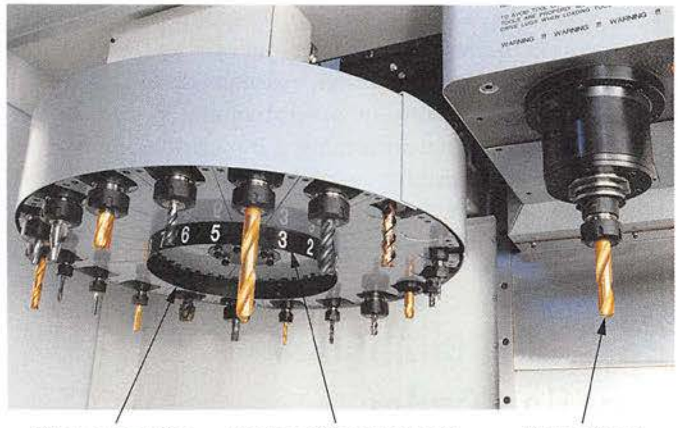
```
N0060   M06     T2
```

SAFETY NOTE

The machine has no way of knowing if the tool is in its proper location. The operator must assume responsibility for proper machine and tool setup. Failing to ensure that tools are in their proper locations may cause damage to the machine or the part or injure the operator.

23.1.6 M07, M08, and M09—Coolant

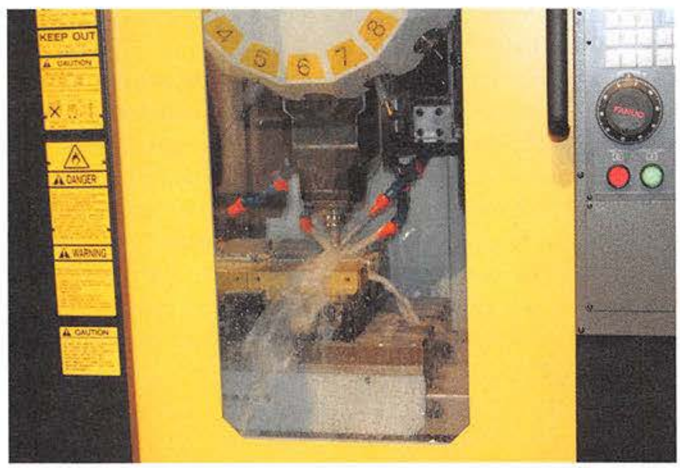
The M07, M08, and M09 codes control coolant flow through the CNC machine, **Figure 23-3**. Both M07 and M08 start coolant flow. M07 starts the flow as a mist, and M08 starts the flow as a flood. M09 stops the flow of coolant through the machine. Some machines are equipped with through-spindle coolant systems, which are activated by different, manufacturer-specific M-codes.



Tool magazine   Tool position numbers   Current tool

Haas Automation, Inc.

Figure 23-2. This CNC machine has a tool magazine capable of holding twenty tools. The numbers of each position in the magazine are clearly visible. An M06 code would signal a tool change designated by the letter "T" and the position number of the tool in the magazine.



Goodheart-Willcox Publisher

Figure 23-3. Sufficient coolant is needed to flush away chips, prevent damage to the tool, reduce workpiece deformation from excessive heat, and improve surface finish.

SAFETY NOTE

Care must be taken when using through-spindle coolant systems. Activating through-spindle coolant when a solid toolholder is in the spindle could damage the spindle bearings.

23.2 Work-Holding Devices

When creating CNC programs, the programmer must consider the work-holding system. During a manual machining process,

the operator can see what is being machined and can avoid moving the work-holding device into the cutter. However, the CNC machine moves from programmed point to programmed point. If part of the work-holding device is in the path of the cutter, the cutter and work-holding device will collide. Knowledge of the work-holding devices and methods allows the programmer to write the program in a way that avoids accidental contact between the cutter and any part of the device. Common work-holding devices for CNC milling processes include the machining vise and specialty jigs and fixtures.

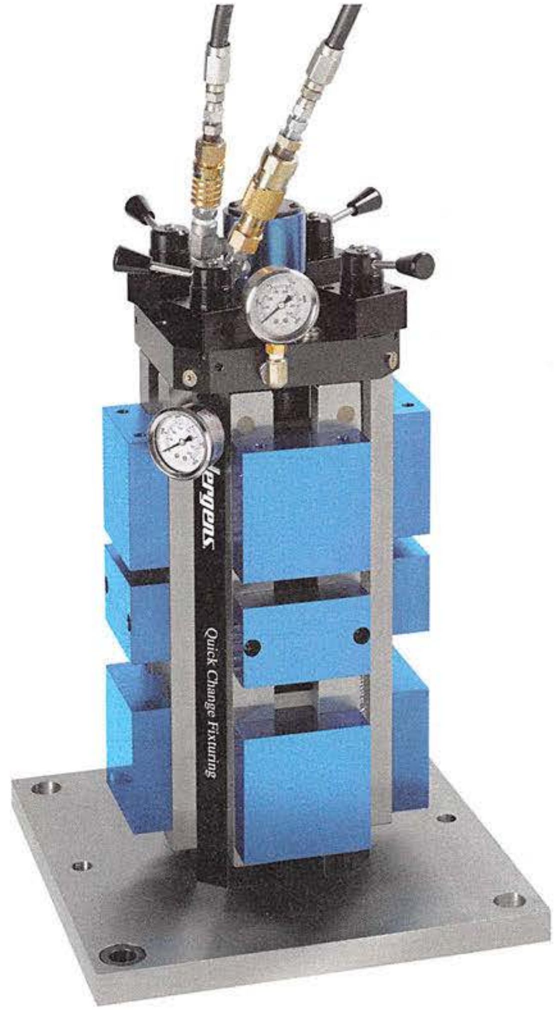
SAFETY NOTE

Check the program carefully to avoid interference between the cutter and work-holding device. A collision can result in serious damage to the cutter, the work-holding device, the part, or the machine. The operator may also be injured by flying debris when a collision occurs.

23.2.1 Machining Vise

In a machine shop, the machining vise is a common work-holding device. It is versatile and can be set up to hold a variety of parts and configurations. The disadvantage of the machining vise is that it takes more time to remove finished parts and load unmachined parts for the next cycle. The operator has to manually open the vise, remove the part, locate the next part, and retighten the vise.

The machining vise is a good option for production runs that involve only a few parts. However, as the volume of parts increases, it becomes more cost-effective to build specialty jigs and fixtures.



© Copyright Jergens, Inc. 2013

Figure 23-4. A four-sided hydraulic vise column. The column has eight stations for holding workpieces. The jaws of the vises are made of machinable aluminum. Pockets can be machined into the jaws to grip oddly shaped workpieces.

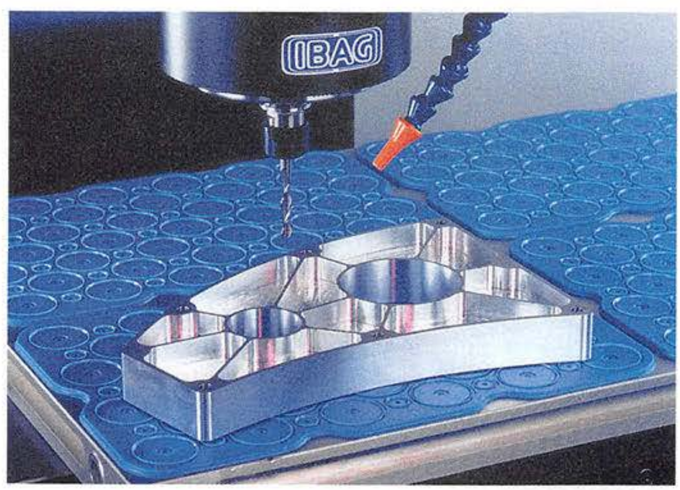
23.2.2 Specialty Jigs and Fixtures

Specialty jigs and fixtures are devices that allow the operator to unload, load, and locate parts faster, saving machining time. In these devices, the clamping mechanisms vary. Commonly used clamping mechanisms for CNC milling include the following:

- Threaded clamps, similar to the movable jaw on a vise.
- Cam locks in which a quarter-turn locks the part into place.
- Toggle clamps.
- Pneumatic or hydraulic vises, **Figure 23-4**.
- Vacuum systems, **Figure 23-5**.
- Magnetic tables.

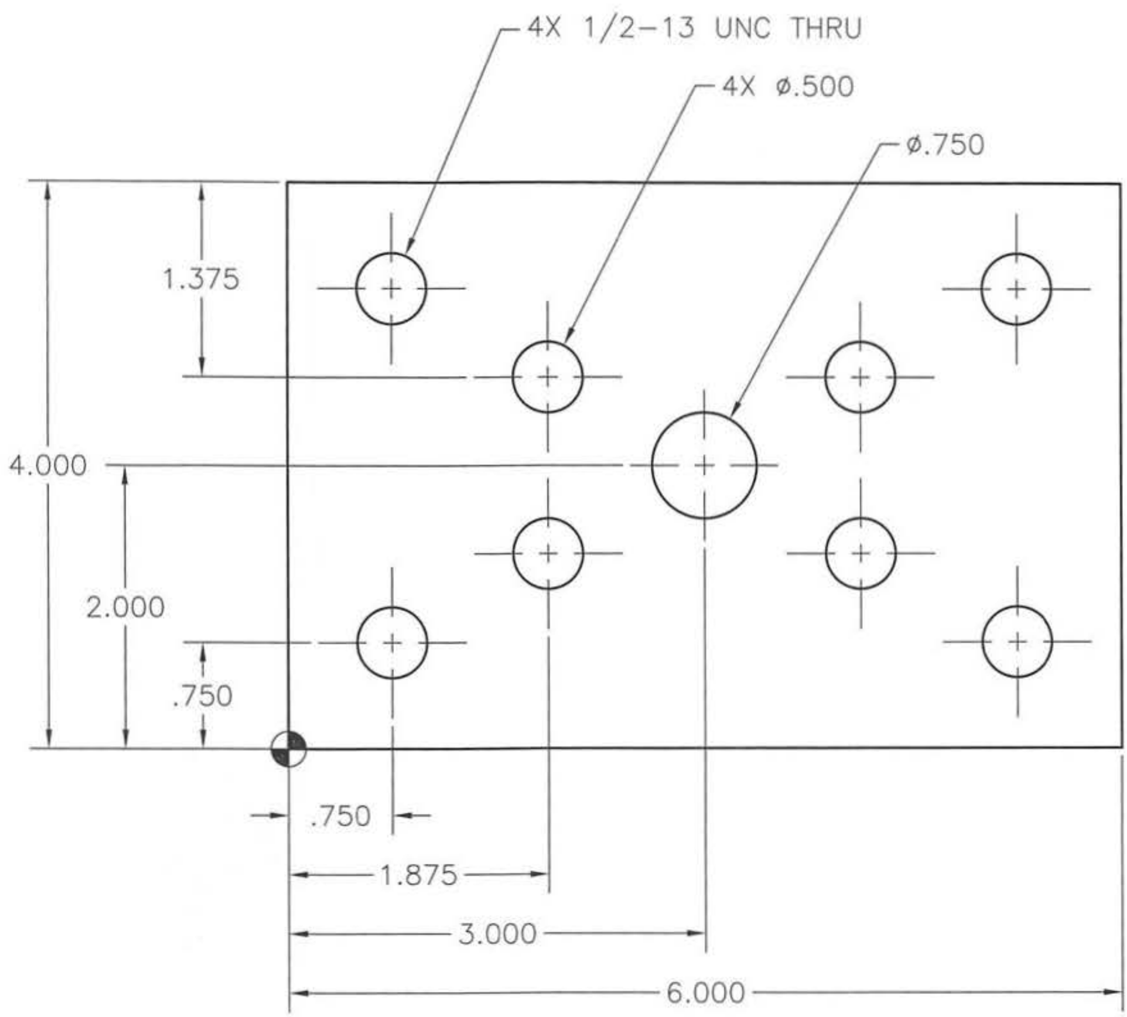
23.3 Planning the Program

Figure 23-6 shows the part to be used in this example of CNC programming. This part is 0.500″ thick and is made from a low-carbon steel. Nine holes will need to be drilled



IBAG North America

Figure 23-5. A vacuum work-holding system, being used here to hold a part on a VMC.



Goodheart-Willcox Publisher

Figure 23-6. This part will be used for all the example CNC programming in the rest of the chapter. The part will be made of 0.500″ thick low-carbon steel, and will require drilling nine holes, four of which will be tapped. The outside edges will be machined.

WORKPLACE SKILLS

Professional Behavior

No matter what career you choose, you will be expected to behave professionally on the job. Professional behavior can help the office run more smoothly. It can also help you advance in your career. Keep the following tips in mind for behaving professionally in the workplace:

- Show respect for your supervisor and coworkers.
- Limit personal conversations and phone calls to break times or lunch.
- Act courteously; remember that others are focusing on their work.
- Arrive on time and do not leave early.
- Offer alternative solutions instead of arguing.
- Practice and encourage safe work practices.

and the outside edges will be machined. All machined features are through features.

23.3.1 Identifying the Work-Holder

As explained earlier in this chapter, the first consideration in writing a CNC program is how the part is going to be held. For this example, a specialty fixture will be used to hold and locate the part. The part sits in the fixture in such a way that the outside edges and all the holes are clear of the nesting system. The fixture is equipped with a magnetic system that can be turned on and off by the operator. It also has a fail-safe device that will not allow the machine to start unless the magnetic device is activated.

23.3.2 Identifying the Required Tools

Next, the programmer determines the tools that will be needed for the process. The example part has the following features:

- The outside edges of a 6.000″ × 4.000″ mild steel block.
- Four 1/2-13 UNC threaded holes.
- One hole that is 0.750″ in diameter.
- Four holes that are 0.500″ in diameter.

The table in **Figure 23-7** shows the suggested tools for each feature. In addition to these tools, a spot drill, or center drill, will be used to locate the centers of the holes. The center drill helps keep the twist drills from "walking" when they come in contact with the base metal. For this part, a #2 center drill will be used to spot the nine holes.

23.3.3 Determining Feed Rates and Spindle Speeds

Next, the programmer determines the feed rates and spindle speeds for each of the identified tools. The information in Chapter 12, *Drills and Drilling Machines*, and Chapter 17, *The Milling Machine*, can be used to specify suitable speeds and feeds for the program. These can always be adjusted as needed to optimize the program. For this example, feeds and speeds for high-speed steel cutters will be used. The table in **Figure 23-8** shows the feed rates and spindle speeds to be used for each of the tools needed to machine the example part.

23.4 Initial Programming and Preparing the Machine

As described in the previous chapter, it is a good practice to begin the program with a line of code that sets the machine up properly to run the rest of the program. The units for this program will be decimal inches, and the coordinates will be absolute. Therefore, the first line of code will look like this:

```
N0010   G00   G53   G20   G90
```

Recall that the first group of numbers specifies the line number in the program. N0010 is the first line in the

Tools, Feeds, and Speeds for Sample Part

Tool #	Tool	Feed Rate (in/min)	Spindle Speed (rpm)
1	.750″ four-fluted end mill	9	1050
2	#2 center drill	3	4000
3	27/64″ drill	6	950
4	.500″ drill	6	800
5	.750″ drill	6	530
6	1/2-13 tap	20	260

Goodheart-Willcox Publisher

Figure 23-8. Suggested feed rates (in/min) and spindle speeds (rpm) for each tool.

Tools Required for Sample Part

Feature	Tools Required
Outside edges	A .750″ four-flute end mill is suggested because it has a large diameter that will allow the edges to be machined faster.
1/2-13 UNC threaded holes	Tap drill using a 27/64″ drill bit, then cut the threads with a 1/2-13 UNC tap.
.750″-diameter hole	A .500″ diameter drill will be needed to drill a pilot hole for the larger drill, because the dead center of the .750″ diameter drill is too large to fit inside the center drilled hole.
.500″-diameter hole	.500″ diameter drill.

Goodheart-Willcox Publisher

Figure 23-7. Suggested tool for each feature.

program. It sets the machine movement to rapid travel (G00), the units to inches (G20), and the coordinate system to absolute (G90). The G53 code changes the coordinate system to the machine's preset coordinate system. Refer to Chapter 22, *CNC Programming Basics*, for more information about these specifications.

23.4.1 Specifying Machine Offsets (G53–G59)

The next task is to set the machine offsets so the machine knows where the workpiece is in relation to the table and spindle. When programs are written offline, the location of the part relative to the machine home position is not considered. ***Machine home*** is the location of the table and spindle, given in Cartesian coordinates. The location of the spindle relative to the table is considered to be X0.000, Y0.000, and Z0.000. A reference point on the part or perhaps a point on the work-holding device is then used to establish a ***program zero position***. Then, when the work-holding device is set up on the table, the program zero position point can be located relative to the machine home position.

The X, Y, and Z locations can be recorded in a machine register under any of six G-codes: G54 through G59. These six codes can all be different. Using multiple zero positions, several parts can be machined using a single program, **Figure 23-9**. Alternatively, some complex machining operations may require multiple zero positions for a single part. When the control encounters one of these codes, it offsets the zero position from the machine home position to the location identified in the machine register. This is now the program zero.

The program zero function is canceled by the G53 code. This code changes the machine parameters to position the axes based on the machine home being the zero point. G53 is non-modal. All subsequent movement commands will use the program zero defined by the G54 code.

For this example, the axes will be positioned so that machine home is the zero point. The program X and Y zero points are directly over the lower-left corner of the block, and the Z zero point is 0.100″ above the top of the block. The line of code looks like this:

```
N0020 G54     X0.000 Y0.000 Z0.100
```

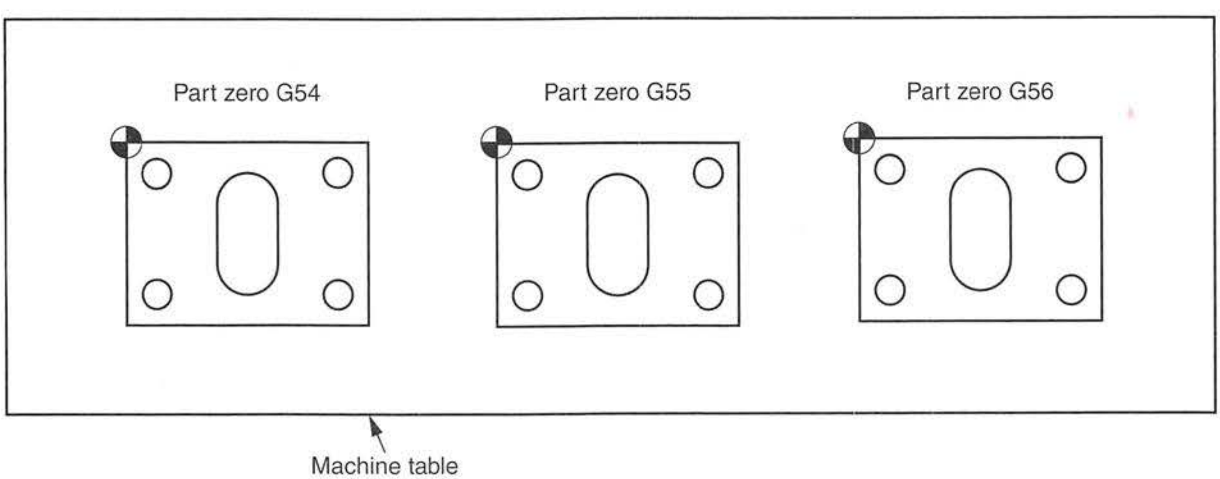
All the information in lines N0010 and N0020 could have been placed on one line. Several years ago, it would have been best practice to do so. Machines were limited in the amount of memory available to store programs, and the size of the program was determined by the number of lines of code it contained. Modern machines are equipped with more memory capacity, so limiting the lines of code is not as critical as it once was. The program is easier to edit or troubleshoot later if each function has a separate line of code, so in this program, the second line specifies the program zero position.

23.4.2 Defining Tool Changes

The next lines in the program define the first tool to be used in the machining process. This will be the 0.750″ end mill that will be used to machine the outside edges of the part. The third and fourth lines of code might look like the following:

```
N0030 M05
N0040 M06     T1 (Change to 0.750" End Mill)
```

The third line of code is a fail-safe operation to stop the spindle rotation. Some programmers may consider this unnecessary. However, it never hurts to think about the many ways a program can malfunction. A crash could cause damage to the machine, the tool, the workpiece, or the operator. A few extra lines of code are well worth the small amount of machine memory to ensure that such malfunctions do not occur.



Goodheart-Willcox Publisher

Figure 23-9. Multiple parts can be machined using a single program by defining multiple zero positions with the G54–G59 codes. The zero positions of three parts, relative to the machine home, are defined using three G-codes (G54, G55, and G56).

The fourth line of code uses the M06 function to begin the automated tool change cycle, **Figure 23-10**. The T1 callout specifies the tool in the turret location designated as location 1. Remember that neither the machine nor the program knows which tool is stored in that location. If this program is being run for the first time after a setup or after a long idle period, it is a good idea to manually remove any toolholders from the spindle and make sure all tools are in their proper locations.

Note the text inside the parentheses in line 4. This is a ***comment code***. The machine ignores any text inside the parentheses, but the text appears on the control screen. Comment codes make it easier for the operator to follow the program. In this case, the comment code serves as a reminder to the operator of which tool should be located in the T1 position. Comment codes also make it easier to locate specific lines of code during editing and troubleshooting applications. Some machines also allow the use of a semicolon to separate comment codes from actual program codes on a program line. However, it is usually easier to locate the comment codes when parentheses are used.

23.4.3 Specifying Cutter Compensation (G40–G42)

If the lower-left corner was used as the zero point of the workpiece and the other three corners were defined using the dimensions of the block, the machine would place the center of the cutter over the zero point. This would result in the dimensions of the block being too small by the diameter of the cutter in both the X and Y directions. To avoid this, the machine must be programmed to compensate for the diameter of the cutter being used.

Goodheart-Willcox Publisher

Figure 23-10. A 180°-rotation automatic tool changer on a VMC. The automated tool change cycle is called by the M06 code. The T1 in this line of code specifies the tool in the corresponding turret location. Here a carbide insert end mill is being placed back into the tool magazine.

The codes G40, G41, and G42 are used to control cutter compensation, **Figure 23-11**. G41 specifies cutter compensation to the left, or counterclockwise. This means that, when envisioning the direction of the cutter moving away from the operator, it will be moving with the edge being machined to the left. G42 is similar to G41 but specifies compensation to the right, or in a clockwise direction. Code G40 cancels both G41 and G42.

For this example, the block will be machined in a counterclockwise direction, so the G41 command is used:

```
N0050  G41    X-.100  Y-.100  Z-.650 D1
```

The letter D defines the diameter of the cutter being used, corresponding to tool T1, and the X and Y coordinates place the edge of the cutter 0.100″ in front of and to the left of the corner of the block. The Z coordinate places the bottom of the cutter 0.050″ below the bottom of the part.

23.5 Programming the Machining Operations

Now that the initial specifications have been programmed and the cutter has been positioned, the next step is to program the actual machining operations. The operations will be performed in the following order:

- Machining the edges.
- Spot drilling the centers of all of the holes.
- Peck drilling all of the holes.
- Tapping the 1/2-13 UNC threaded holes.

23.5.1 Machining the Outside Edges

The first operation will be to machine the outside edges of the part:

```
N0060  M03    S1050
N0070  M08
N0080  G01    X0.000  Y0.000  F9.0
```

Line N0060 turns the spindle on in a clockwise rotation, and the S1050 specifies a speed of 1050 rpm. Line N0070 starts the coolant flow. Line N0080 brings the edge of the cutter into contact with the part and changes the feed from rapid to a programmed feed rate of 9.0 inches per minute.

The next lines of code move the table so that the cutter will machine the outside edges of the part:

```
N0090  X6.000
N0100  Y4.000
N0110  X0.000
N0120  Y0.000
N0130  G00    X-.100  Y-.100  Z.100  M05
N0140  M09
```

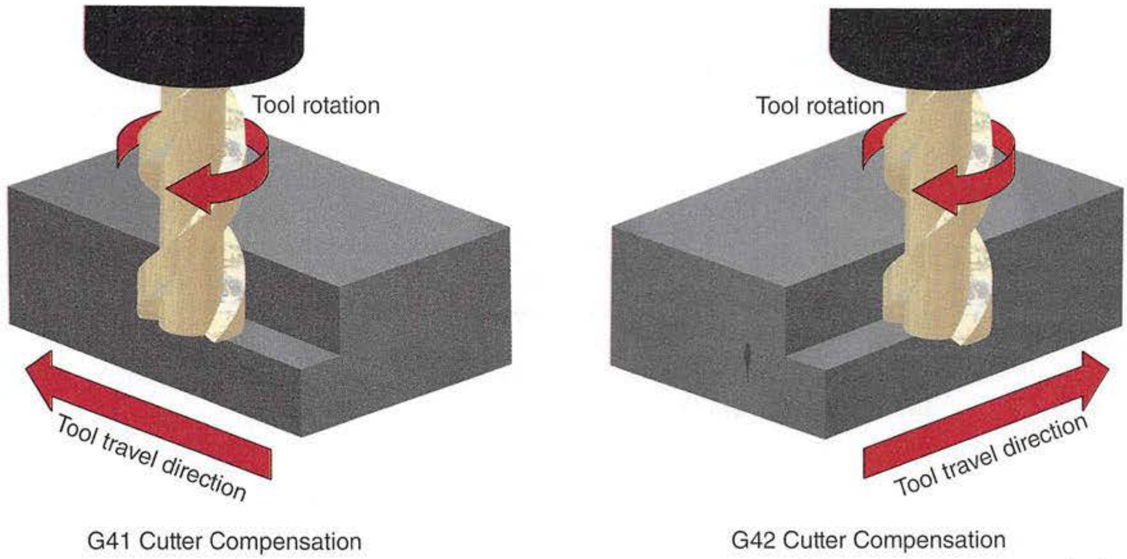


Figure 23-11. Envision the cutter moving away from the operator. Use G41 if the edge being machined is on the left (cutter is left of material). Use G42 if the edge being machined is on the right (cutter is right of material). Use G40 to cancel both G41 and G42.

The last two lines, N0130 and N0140, move the edge of the cutter away from the part after the machining is complete using rapid positioning (G00), stop the spindle rotation (M05), and turn the coolant off (M09).

23.5.2 Spot Drilling

The next machining step is to change the tool to the #2 center drill in preparation for the spot drilling operation. As shown in **Figure 23-8**, the center drill has been defined as tool #2, or T2. The lines of code include the following:

```
N0150  Z.500
N0160  M06    T2 (Change to #2 Center Drill)
N0170  X.750  Y.750   Z.100
N0180  M03    S4000
N0190  M08
```

Line N0160 accomplishes the tool change, and line N0170 moves the cutter to a point 0.100″ above the top of the part and just over the location for the 1/2-13 threaded hole at the lower-left corner of the part. In line N0180, the spindle is again started in the clockwise direction, this time at 4000 rpm, and line N0190 starts the coolant flow. All nine locations for the holes will be spot drilled in this step.

The drilling operation will be performed using a code called a ***canned cycle*** or ***subroutine***, which is a set of commands that follows a prescribed sequence, almost like a miniature program. Based on the information the programmer includes in the line of code that activates the canned cycle, the program follows a prescribed series of moves. A canned cycle remains active and repeats the same sequence until it is canceled using the G80 code.

There are several canned cycles related to the drilling operation: G73, G74, G76, and G81 through G89. G81 is the code for a simple drilling operation. It will be used in this case because it is best suited for the spot drilling operation. The line of code that begins this process will look something like this:

```
N0200  G99   G81   R.100 Z-.075 F100
```

The G99 code directs the machine to retract the Z axis to the location defined with the letter R. The R is usually used to define a programmed radius for circular interpolation, but, in this case, it defines the position for tool retraction. Alternatively, the programmer can use the G98 code, which directs the machine to retract the tool to the original Z position (which in this case would be Z.200).

The G81 code starts the canned drill cycle. This cycle advances the drill at the programmed feed rate (F) to the Z depth defined. The execution of line N0190 spot drills the first hole. The following lines of code define the locations for the rest of the holes:

```
N0210  Y3.250
N0220  X1.875  Y2.625
N0230  Y1.375
N0240  X3.000  Y2.000
N0250  X4.125  Y1.375
N0260  Y2.625
N0270  X5.250  Y3.250
N0280  Y.750
N0290  G80
N0300  G00    X-.100  Y-.100  Z.100  M05
N0310  M09
```

Each of the points defined in lines N0210 through N0280 places the center drill above the part in the proper location. The G81 command is still active, so it executes the canned drilling cycle after each of the linear moves defined in these lines of the code. Line N0290 contains the G80 code that

cancels the G81 canned drilling cycle. Line N0300 moves the cutter to a clearance position away from the part and stops the spindle rotation. Line N0310 stops the coolant flood.

23.5.3 Peck Drilling

The next machining step is to change the tool to the 27/64″ drill, in preparation for the drilling operation for the pilot holes needed for the threading operation. Recall that the 27/64″ drill has been specified as tool #3, or T3. The lines of code to prepare for drilling the pilot holes include the following:

```
N0320  M06   T3 (Change to #3 27/64" Drill)
N0330  X.750 Y.750  Z.100
N0340  M03   S950
N0350  M08
```

The tool change is accomplished in line N0320. Line N0330 places the drill 0.100″ above the part and at the position for the first of the four threaded holes at the lower-left corner of the block. Lines N0340 and N0350 start the spindle and the coolant flow.

The pilot holes will be drilled using the G83 code, which calls a canned cycle for peck drilling. ***Peck drilling*** is used in order to clear the chips away at intervals while each hole is drilled. This command requires more information to define the cycle. This is the correct way to write a line of code for the G83 command:

```
G83   X#   Y#   Z#   R#   Q#   F#
```

Where:

- G83 is the command to activate the peck drilling canned cycle.
- X# and Y# are the XY coordinates for the hole position.
- Z# is the Z coordinate for the bottom of the part, with additional clearance to allow for the included angle at the tip of the drill.
- R# is the position of the clearance plane to which the drill retracts at the end of each cutting feed.
- Q# is the depth of cut for each cutting feed.
- F# specifies the feed rate in the drilling operation.

Figure 23-12 illustrates how this canned cycle works. The drill begins a programmed feed rate from the clearance position above the part (R position) downward a distance equal to the value assigned to the letter Q. It then retracts to the clearance plane to allow any buildup of chips to break away and clear the hole. It then feeds down to the second level, which is a depth of two times the value for the letter Q. Again, it retracts to the R position. The cycle continues until it reaches the Z depth specified in the line of code. The tool then retracts to either the Z axis of the location defined with the letter R or to the original Z axis position, depending on whether G98 or G99 is used.

In the example program, the tip of the drill is already positioned at a point directly above the location of the hole at a height of 0.100″ above the top surface. To determine the value of Q, the programmer adds the 0.100 value for the clearance location, the 0.500″ part thickness, and 0.060″ for clearance below the part so that the included angle of the drill extends beyond the bottom surface. A full 0.060″ clearance is not necessary, but, when added to the other two numbers, it equals 0.660″, which is easily divisible by three. This program will use three cutting feeds, or pecks. Each peck will be equal to 0.220″. This is the value for Q. Therefore, the next line of code will read:

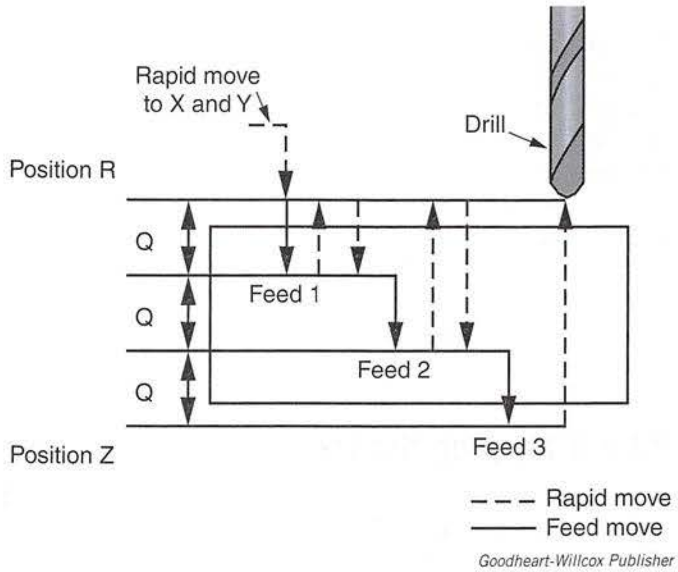


Figure 23-12. Drill movement in the G83 peck drill canned cycle.

```
N0360  G99  G83  Z-.560 R.100 Q.220 F6.0
```

Since the tool is already positioned in the proper XY location, the coordinates for those two axes do not need to be specified. Line N0360 drills the first hole. The following lines of code define the other three hole positions:

```
N0370  Y3.25
N0380  X5.25
N0390  Y.750
N0400  G80
N0410  G00  X-.100  Y-.100  Z.100  M05
N0420  M09
```

The code to drill the four 0.500″ holes and the pilot hole for the 0.750″ diameter hole in the middle is similar to the code used for these four holes. Including code for the tool changes, the remainder of the peck drilling cycles will look like this:

```
N0430  M06    T4 (Change to #4 1/2" Drill)
N0440  X1.875 Y1.375 Z.100
N0450  M03    S800
N0460  M08
N0470  G99    G83   Z-.560  R.100  Q.220  F6.0
N0480  Y3.25
N0490  X5.25
N0500  Y.750
```

```
N0510  X3.000 Y2.000
N0520  G80
N0530  G00    X-.100  Y-.100  Z.100  M05
N0540  M09
N0550  M06    T5 (Change to #5 3/4" Drill)
N0560  X3.000 Y2.000 Z.100
N0570  M03    S530
N0580  M08
N0590  G99    G83     Z-.560  R.100  Q.220  F6.0
N0600  G80
N0610  G00    X-.100  Y-.100  Z.100  M05
N0620  M09
```

23.5.4 Tapping the Holes

The requirements for tapping holes are different from other types of drilling, **Figure 23-13**. First, the feed rate and spindle speed are much slower. Second, the feed and the spindle speed must be synchronized so that the feed rate equals the lead of the thread multiplied by the rpm of the spindle. The lead is the distance from the crest of one thread to the crest of the thread next to it. Finally, when the depth of the thread is reached, both the spindle rotation and the feed must stop, then reverse direction to back the tap out of the newly threaded hole. The G84 code accomplishes these tasks based on the information provided. The next few lines of code start the tapping process:

```
N0630  M06   T6 (Change to #6 1/2-13 Rigid Tap)
N0640  X.750 Y.750  Z.100
N0650  M03   S260
N0660  M08
```

Notice that the spindle speed has been programmed at 260 rpm. This was done for two reasons. First, this rpm is on the low end of recommended tapping spindle speeds. If the trial runs indicate that the spindle speed and feed rates can be increased, this can be done before the program is released for production runs. Second, the lead of the thread must be considered. In a 1/2-13 thread, there are 13 threads per every inch of length on the tap. Since the tapping feed rate must be an even ratio of the thread lead and the spindle speed, the formula for the tapping feed rate is as follows:

$$F = \text{Lead of tap} \times \text{rpm}$$

Where:

F = tapping feed rate

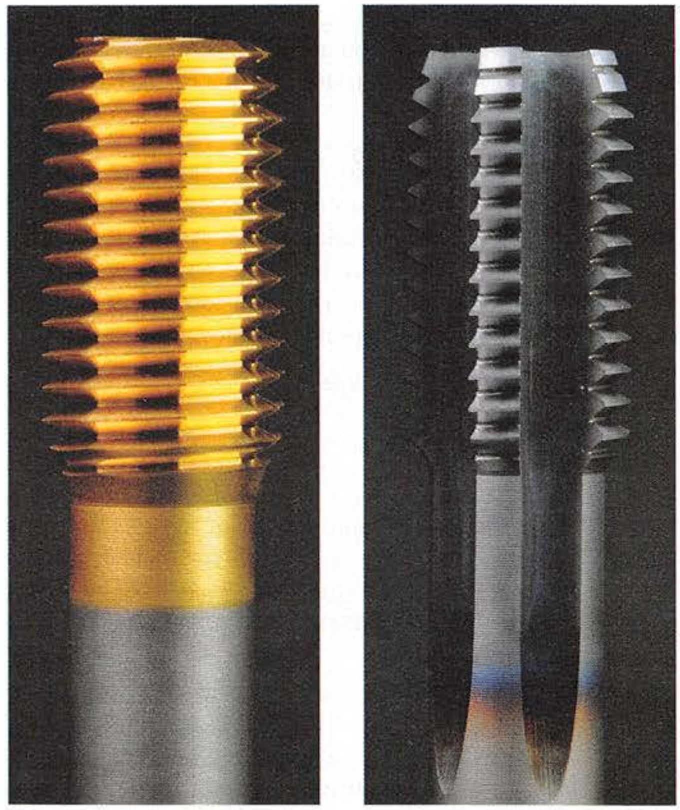
Lead of tap = 1/thread per inch

rpm = spindle speed

The tapping feed rate is as follows:

$$F = 1/13 \times 260$$

$$F = 20 \text{ inches per minute}$$



A B

OSG Tap & Die, Inc.

Figure 23-13. Threading tools. A—A threadforming tap for soft metals, in which chip formation is a problem. B—A threadcutting tap. The flutes in the threadcutting tap allow chips to fall away from the cutter.

A spindle speed of 390 rpm could have also been used. When plugged into the formula above, this would have yielded a feed rate of 30 inches per minute. However, it is best to start slow. Now that the spindle speed and feed rate have been determined, the code to thread the four holes can be written as follows:

```
N0670  G99  G84  Z-.560 R.100 F20.0
```

This line of code threads the hole, then reverses the feed and spindle rotation to retract the tap from the hole. The next step is to complete the threading cycle by specifying the XY locations of the other three holes.

```
N0680  Y3.25
N0690  X5.25
N0700  Y.750
N0710  G80
N0720  G00  X-.100  Y-.100  Z.100  M05
N0730  M09
N0740  M30
```

As with the other canned cycles, the G80 cancels the tapping cycle. Line N0720 retracts the tool and stops the spindle. Line N0730 stops the coolant flow. The last line of the program contains the M30 code, which ends the program and resets it to the beginning. The complete code is shown below:

```
N0010  G00    G53     G20 G90
N0020  G54    X0.000  Y0.000  Z0.100
N0030  M05
N0040  M06    T1 (Change to .750" End Mill)
N0050  G41    X-.100  Y-.100  Z-.650  D.750
N0060  M03    S1050
N0070  M08
N0080  G01    X0.000  Y0.000  F9.0
N0090  X6.000
N0100  Y4.000
N0110  X0.000
N0120  Y0.000
N0130  G00    X-.100  Y-.100  Z.100   M05
N0140  M09
N0150  Z.500
N0160  M06    T2 (Change to #2 Center Drill)
N0170  X.750  Y.750   Z.200
N0180  M03    S4000
N0190  M08
N0200  G99    G81     R.100   Z-.075  F3.0
N0210  Y3.250
N0220  X1.875 Y2.625
N0230  Y1.375
N0240  X3.000 Y2.000
N0250  X4.125 Y1.375
N0260  Y2.625
N0270  X5.250 Y3.250
N0280  Y.750
N0290  G80
N0300  G00    X-.100  Y-.100  Z.100   M05
N0310  M09
N0320  M06    T3 (Change to #3 27/64" Drill)
N0330  X.750  Y.750   Z.100
N0340  M03    S950
N0350  M08
N0360  G99    G83     Z-.560  R.100   Q.220  F6.0
N0370  Y3.25
N0380  X5.25
N0390  Y.750
N0400  G80
N0410  G00    X-.100  Y-.100  Z.100   M05
N0420  M09
N0430  M06    T4 (Change to #4 1/2" Drill)
N0440  X1.875 Y1.375  Z.100
N0450  M03    S800
N0460  M08
N0470  G99    G83     Z-.560  R.100   Q.220  F6.0
N0480  Y3.25
N0490  X5.25
N0500  Y.750
N0510  X3.000 Y2.000
N0520  G80
N0530  G00    X-.100  Y-.100  Z.100   M05
N0540  M09
N0550  M06    T5 (Change to #5 3/4" Drill)
N0560  X3.000 Y2.000  Z.100
N0570  M03    S530
N0580  M08
N0590  G99    G83     Z-.560  R.100   Q.220  F6.0
N0600  G80
N0610  G00    X-.100  Y-.100  Z.100   M05
N0620  M09
N0630  M06    T6 (Change to #6 1/2"-13 Rigid Tap)
N0640  X.750  Y.750   Z.100
N0650  M03    S260
N0660  M08
N0670  G99    G84     Z-.560  R.100   F20.0
N0680  Y3.25
N0690  X5.25
N0700  Y.750
N0710  G80
N0720  G00    X-.100  Y-.100  Z.100   M05
N0730  M09
N0740  M30
```

Chapter Review

Summary

- Miscellaneous function codes, or M-codes, include machine functions, such as stopping the program execution, ending the program, turning the spindle and coolant flow on and off, and performing automatic tool changes.
- Work-holding devices for CNC machining include machining vises and specialty jigs and fixtures that are designed to hold specific workpieces efficiently.
- Before a CNC program can be written, the programmer needs to know how the workpiece will be held in place, what tools will be needed, and the feed rates and spindle speeds required for the tools.
- The first lines of a CNC program set up the machine, including specifying units, type of coordinates, machine offsets, tool change specifications, and cutter compensation.
- A CNC program consists of lines of code that guide the CNC machine to select the proper tools and perform the required processes in a logical, efficient order.

Review Questions

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

1. The code most likely to be used by a programmer to stop the machine for a required manual tool change is ______.
2. What is the difference between the M00 code and the M01 code?
3. The ______ code is used to stop the machine at the end of the program and to reset the program to the first line of code, ready for reuse.
4. To turn on the spindle and specify clockwise rotation, the programmer uses the ______ code.
5. What is the meaning of the following line of code: N0030 M04 S3000?
6. Briefly explain how an automatic tool change can be written into a program.
7. Why is it important to identify the work-holder to be used before beginning to write a CNC program?
8. Under what circumstances might a machining vise be preferred instead of a specialty jig or fixture as the work-holding device for a machining operation?
9. Explain why it is good practice to include a spot drill step to locate the centers of holes before the holes are drilled.
10. The location of the spindle relative to the table, given in Cartesian coordinates, is known as the ______ home.
11. Which codes provide the X, Y, and Z coordinates of the program zero location?
12. Text enclosed in parentheses that is not read as part of the program is known as a(n) ______.
13. Which code would a programmer use to specify cutter compensation to the right (clockwise)?
14. A canned cycle, or ______, is a set of commands that follows a prescribed sequence.
15. Suppose you have programmed a canned drilling cycle to drill six holes at specific locations on the workpiece. What line of code would you write to move the cutter away from the part to a location of $X = -0.200$, $Y = -0.200$, $Z = 0.100$ using rapid positioning; stop the spindle rotation; and turn off the coolant? Assume that the line number is N0540.
16. In the code that specifies a peck drilling operation, what does the value of Q determine?
17. How is the value of Q calculated?
18. In what three ways are the requirements for tapping holes different from other types of drilling?
19. What would be the feed rate for a tapping operation for a 1/4-20 thread with a spindle speed of 400?
20. Which code is used at the end of a tapping cycle to end the cycle?

CHAPTER **24**

CNC Turning

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Identify work-holding devices that are commonly used with CNC lathes and turning centers.
- List the information needed to prepare for writing a program for a CNC turning center.
- Explain how to set the machine offsets and prepare the CNC machine for a turning operation.
- Carry out the procedure for writing a program to machine a part using a two-axis CNC lathe.

Technical Terms

automatic accumulator
bar puller
part catcher
repetitive cycle

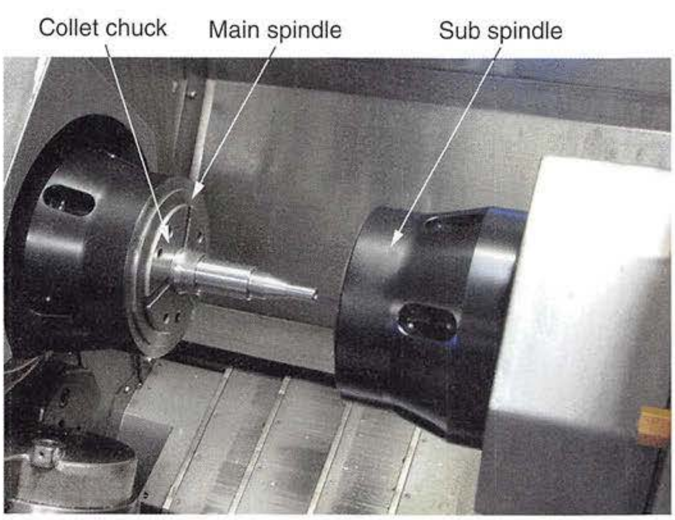
This chapter introduces the creation of CNC programs for turning centers. The machining process is different from a CNC machining center, but there are many similarities. Once you understand programming for a CNC machining center, as described in Chapter 23, *CNC Milling*, it is fairly easy to learn how to program a CNC turning center.

24.1 Work-Holding Devices for CNC Turning Centers

Work-holding devices used with CNC turning centers do not differ significantly from those used with manual lathes. Differences are often dictated by the application of the technology. For example, in a toolroom setting, parts are typically made in small quantities. Sometimes only a single part may be needed. In this application, three-jaw and four-jaw chucks are often used.

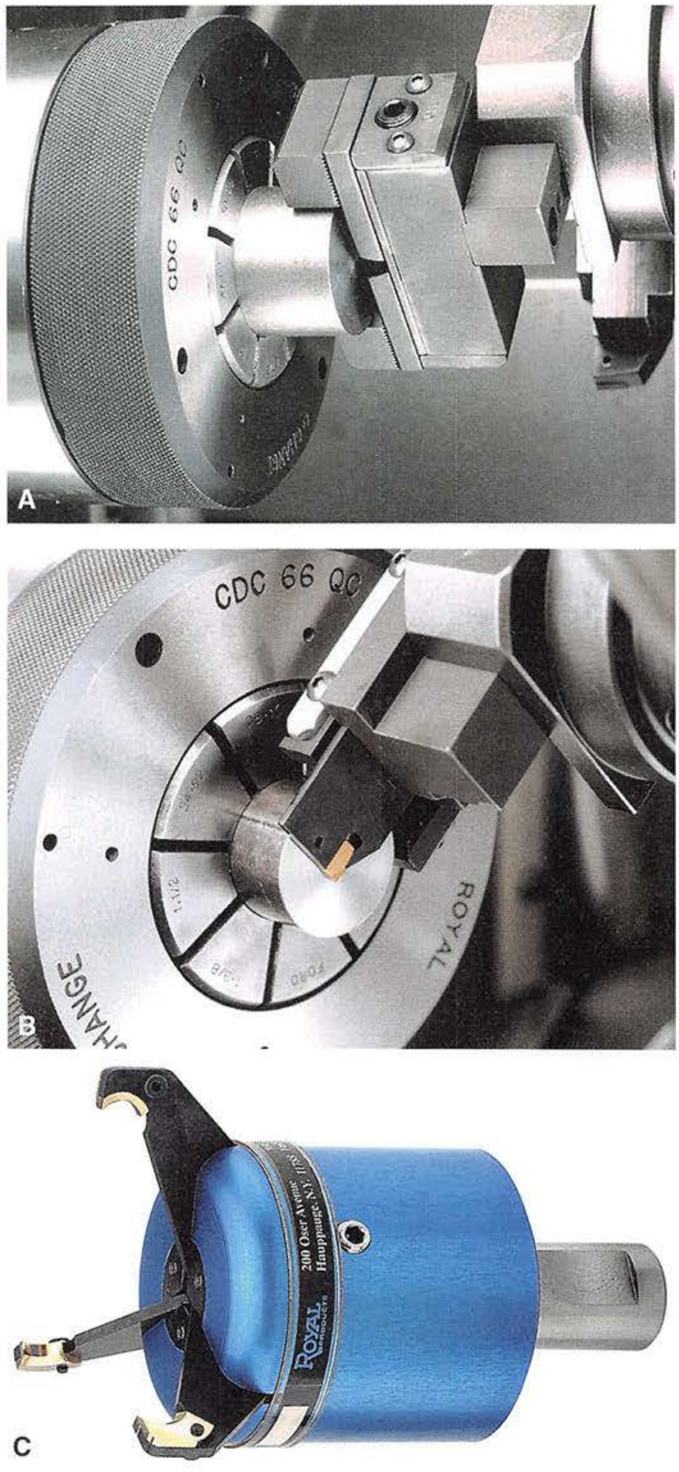
It is not unusual to find collets being used in a toolroom setting as well, **Figure 24-1**. Because collets reduce loading and unloading times, the use of collets best suits applications in which large volumes of parts are being manufactured. In a production setting, chucks are seldom used because of the extra time it takes to load and unload parts. Also, with collets, automated systems can be used to load and unload parts and to activate the collets.

A turning center can be equipped with a bar feeder, bar puller, and part catcher to automate the machining process. Bar feeders feed bar stock into the turning center. ***Bar pullers*** are mounted in the turret with the cutting tools, **Figure 24-2**. Bar pullers clamp and pull bar stock through the spindle for machining. ***Part catchers*** are programmable devices used to catch finished parts as they are cut off the bar stock. Combined with an ***automatic accumulator*** to collect finished parts, **Figure 24-3**, turning centers can be completely automated.



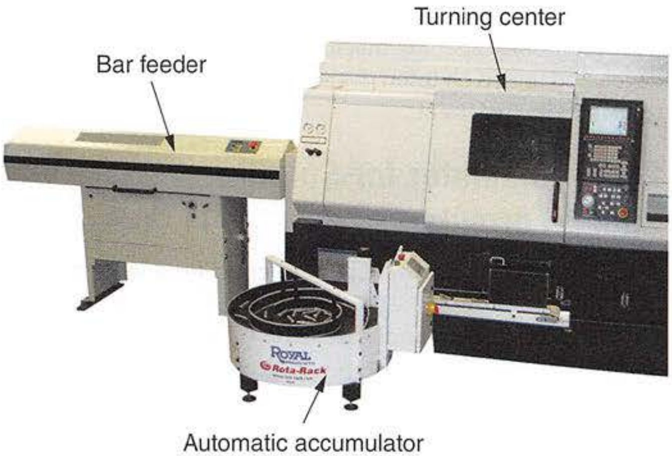
Royal Products, Division of Curran Manufacturing Corporation

Figure 24-1. A turning center with collet chucks in both the main and sub spindle. The part can be held by the collet in the sub spindle so the rear end of the part can be machined.



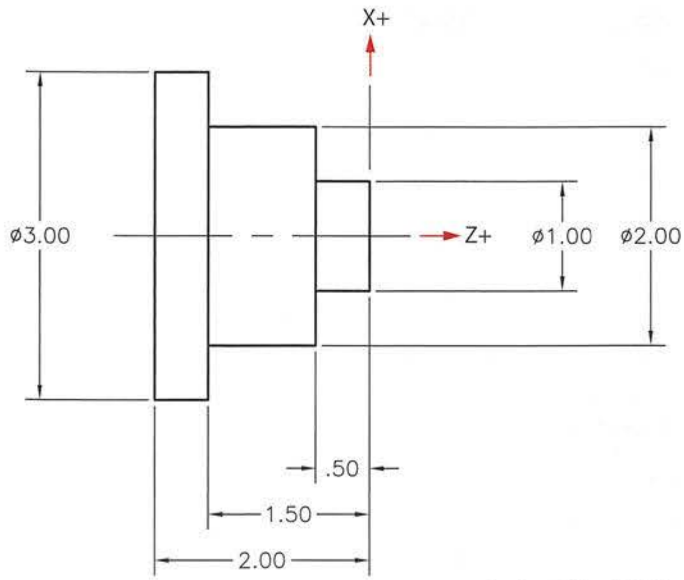
Royal Products, Division of Curran Manufacturing Corporation

Figure 24-2. Bar pullers. A—Heavy-duty CNC bar puller. B—Combination bar puller and parting tool; reduces cycle time by combining the cutoff and bar pulling operations. C—Bar puller actuated by the through-spindle coolant system.



Royal Products, Division of Curran Manufacturing Corporation

Figure 24-3. Combined with a bar feeder and automatic accumulator, this turning center is completely automated. The spiral-shaped Rota-Rack™ automatic accumulator collects finished parts as they are ejected from the turning center and indexes the parts toward its center.



Goodheart-Willcox Publisher

Figure 24-4. This chapter describes how to create the program to machine this part on a two-axis turning center.

24.2 Planning for a CNC Turning Program

This chapter describes how to create a program to machine the part shown in **Figure 24-4**. The part is a cylindrical piece with three different diameters. The base material is bar stock 3.00″ in diameter.

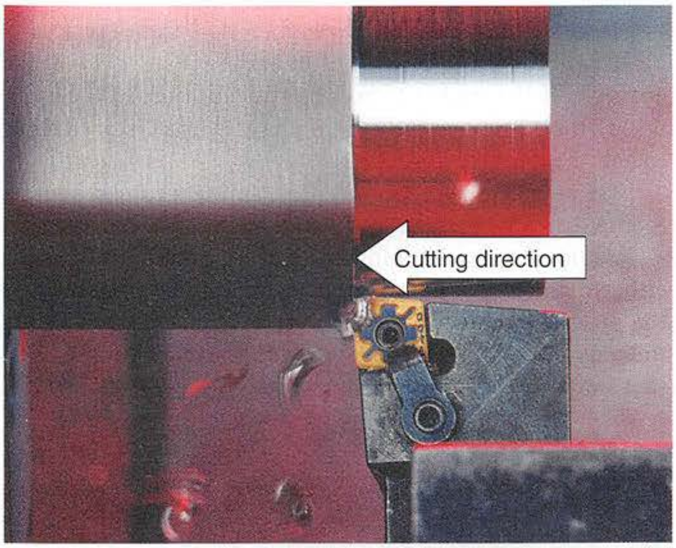
24.2.1 Identifying the Processes and Tools

As when developing a milling program, the first step in developing a turning program is to determine the processes to be performed and the tools that will be needed. Because the processes determine the tools to be used, begin by identifying the processes. For the part to be machined in this chapter, the following four processes are required:

1. Facing the end of the part.
2. Roughing out the basic shape, **Figure 24-5**.
3. Making a finish cut to smooth the surfaces.
4. Performing a cutoff operation to separate the finished part from the bar stock.

To perform these steps, the following three tools will be needed:

- A roughing tool to remove the majority of the material (Steps 1 and 2).
- A finishing tool to remove a small amount of material to create a smoother finish (Step 3).
- A parting tool to cut the finished part away from the bar stock (Step 4).



Carboloy

Figure 24-5. Using a rough pass to machine a basic profile with an indexable insert cutting tool.

Many machinists use the same tool for the roughing and finishing operations. However, the roughing operation dulls the cutter faster than the finish operation because of the amount of material being removed. In a high-volume production operation, using a second tool for finishing allows the tool for the roughing operation to be used longer. This may result in fewer cutters needed per production run.

24.2.2 Coordinate System Orientation

Before writing a program for a CNC lathe or turning center, it is useful to visualize the positioning of the part along the coordinate system axes. The Z axis is the cylindrical axis that passes through the part, as shown in **Figure 24-4**. The positive direction of the Z axis runs from the headstock, which turns the part, toward the tailstock end of the machine, if a tailstock is being used. The X axis runs perpendicular to the Z axis.

Set up the tool post so the tip of the tool travels along the X axis as the cross-slide is advanced and retracted. The tip of the tool should intersect the Z axis. The program created in this chapter is for a two-axis lathe like the one shown in **Figure 24-6**.

The Y axis typically is not used with a standard two-axis CNC lathe equipped with a cross-slide. It is more commonly used with CNC turning centers in which the tools are held in a tool turret. See **Figure 24-7**. The Y axis moves in a vertical motion, with the positive direction upward and the negative direction downward.

24.2.3 Radius vs. Diameter

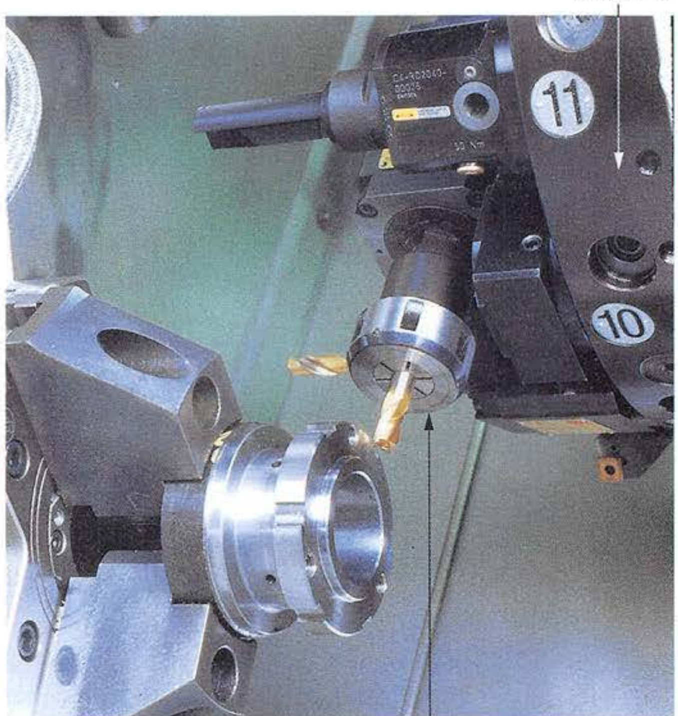
In a turning operation, the part turns around the Z axis. As the cutter advances along the X axis, the tool reduces the radius of the part.

When programming, remember that the diameter of a cylindrical part is two times its radius, **Figure 24-8**. For example, if the goal is to reduce the diameter of a part by 0.100″, the cutter must advance into the part a distance equal to half the diameter, or 0.050″.

Older machines required programmers to compensate mathematically for the difference between the diameter and radius. Modern equipment can be programmed using either diameter or radius input data. This is controlled in the machine's registers and cannot be adjusted from within individual programs. These settings are usually protected so that only advanced programmers have access to them. It is up to the programmer to verify these settings when creating a program for a specific machine. For the example program developed in this chapter, the equipment is assumed to be set to take diameter input.

24.2.4 Diameter for Spindle Speed Formula

The spindle speed formula in a program for a lathe or turning center requires a cutting speed (CS, based on the material being machined) and a diameter (D) component:

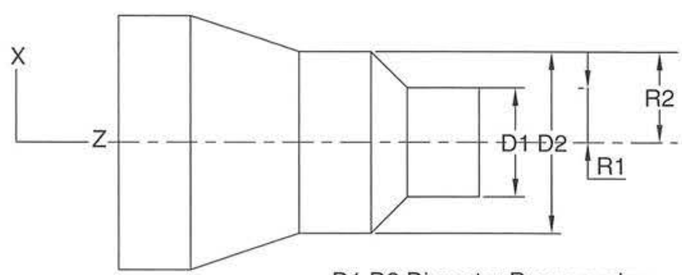


Sandvik Coromant

Figure 24-7. The tool turret on this three-axis turning center allows vertical movement along the Y axis. The milling head attachment is being used to machine slots in this part.

Hardinge

Figure 24-6. A slant-bed two-axis turning center.



Goodheart-Willcox Publisher

Figure 24-8. The difference between diameter and radius programming.

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

In the milling operation, the diameter used in the formula is the cutter diameter. However, in turning, the diameter of the workpiece is used. It is a good idea to calculate the changes as material is removed on each pass. As the part is machined and the diameter is reduced, the formula may need to be changed. However, if the changes in diameter are small, the changes in spindle speed may be negligible.

24.3 Initial Programming

The first few lines in a program for a CNC lathe or turning center are similar to those for a milling operation. They set up the machine and prepare it for the current operation.

24.3.1 Setting the Machine Offsets and Cutter Compensations

Typically, the machine zero position is set so that the tip of the tool is in line with the cylindrical axis and flush with the end of the workpiece. During the initial setup, the tool is manually positioned at this point and the value from the digital readout is recorded in the machine register under G-codes 54 through 59. Recall from Chapter 23, *CNC Milling*, that these codes are used to specify machine offsets.

Cutter compensation settings may also be included in the initial program section. Although cutter compensation in a turning operation is similar to that for milling operations, the visualization technique is different. In turning operations, the programmer must visualize the workpiece looking from the tailstock end (or where the tailstock would be if used) toward the workpiece, with the spindle in the background. If the cutting tool is mounted to the right of the workpiece from this perspective, the G42 code is used to compensate the cutter to the right. If the cutting tool is mounted to the left of the workpiece, the G41 code is used to compensate the cutter to the left. As with milling operations, the G40 code is used to cancel all tool offsets.

24.3.2 Preparing the Machine

The first lines of code set the appropriate program defaults and move the cutter to a point away from the part:

```
N0010  G00  G53     G20 G90
N0020  G54  X-3.25  Z.25
N0030  M05
N0040  M00 (Manually change to a roughing tool)
```

Line N0010 sets the travel speed to rapid traverse (G00), clears any machine offset position that may be loaded in the register (G53), sets the units to inches (G20), and sets positioning to absolute (G90). This clears any information that may be active from a previous program or from a previous cycle in the current program that did not run to completion.

Line N0020 sets the machine zero position. Notice that the coordinates in this case are stored under code G54. Line N0030 is a fail-safe operation to stop the spindle rotation.

SAFETY NOTE

Some programmers consider it unnecessary to stop spindle rotation at this point. However, it never hurts to think about the many ways a program can malfunction and cause damage to the machine, the tool, or the workpiece. A few extra lines of code are well worth the small amount of machine memory to help ensure that such malfunctions do not occur.

Line N0040 pauses the program with the miscellaneous function M00. Recall from Chapter 23 that M00 is a program stop. It is used to pause a program when needed to allow an operator to clear chips away from a part, inspect the tool for wear, change tools manually, or measure a feature, **Figure 24-9**. In the example in this chapter, the machine requires the operator to manually change the tools. The text inside the parentheses is a comment prompting the operator to change the tool to a roughing tool for the first machining operation.

24.4 Programming the Machine Operations

After the correct tool has been loaded, the machine is ready to begin the first machining step, which is facing the end of the part. At this point, the machine is still paused from the execution of the M00 command. To restart the machine, the operator presses the cycle start button on the control panel. Pressing cycle start returns the machine to the state it was in

Marcelo Costa/Shutterstock.com

Figure 24-9. An M00 code can be used to stop the program to inspect a machined feature.

before the M00 command was executed. For example, if the spindle was rotating and the coolant was flowing, they will restart using the same settings as before the M00 command was read by the control.

24.4.1 Facing the End of the Part

The following lines of code direct the machine to perform the facing operation, **Figure 24-10**:

```
N0050  G97  M03     S1500
N0060  G00  Z-.050
N0070  G01  X0.0    F.019
N0080  G00  X-3.05  Z.05 (End of facing operation)
```

Line N0050 starts the spindle in a clockwise rotation at 1500 rpm. Note the use of the G97 code. The G97 code cancels the G96 constant surface speed code, which will be explained in the next section. The G97 code ensures that the spindle is revolving at a constant rate. The next line moves the tip of the cutting tool to a position just off the outside diameter of the workpiece, 0.050″ away from the tip of the part. Line N0070 feeds the tip of the cutter across the end of the part to the part's center at a feed rate of 0.019 inch per revolution. This removes 0.050″ stock from the end of the part. Line N0080 moves the cutter away from the part at a rapid travel rate, ending the facing operation.

24.4.2 Constant Surface Speed

When turning parts in a lathe, making adjustments to the surface speed of the cutting process is critical to maintaining the surface finish of the part and optimizing the tool life of the cutters and inserts. The formula for surface speed is

$$\text{sfm} = \frac{\text{rpm} \times D}{3.82}$$

Sandvik Coromant

Figure 24-10. Facing removes material from the end of the workpiece. Typically, facing is one of the first operations performed when turning a part.

Where:

- sfm equals surface feet per minute.
- rpm equals the revolutions per minute of the workpiece.
- D is the diameter of the workpiece.
- The number 3.82 is a constant.

With each cutting pass of a lathe, the diameter of the workpiece changes. If the changes are subtle, the surface speed setting may not need to be changed. However, if the diameter changes significantly, the programmer needs to change the surface speed for each cutting pass. To simplify the programming process and optimize machine cycle time, the example in this chapter uses the G96 code for constant surface speed (CSS). This code directs the CNC machine's controller to alter the speed of the spindle to compensate for the reduced diameter of the workpiece, maintaining a constant surface speed for the machining process. As the cutter edge approaches the zero point on the Z axis, its speed increases as the radius distance from the cutter to the Z axis decreases.

However, before adding the G96 code to the program, the programmer must insert the G50 code to define the maximum rate at which the spindle can turn while in CSS mode. The G50 function ensures that the machine will not increase the spindle speed beyond this maximum setting. The following two lines of code set the maximum spindle rate to 3000 rpm and activate the CSS function using a CSS of 600 sfm:

```
N0090  G50  S3000  (Max rpm while in CSS mode)
N0100  G96  S600   (Set CSS at 600 and activate)
```

24.4.3 Stock Removal Cycles

The next step in the machining process is to remove the majority of the material using a rough cutting operation. CNC lathes have codes that perform operations similar to the canned cycles on CNC machining centers. These operations are called ***repetitive cycles***. The G71 and G72 codes remove large amounts of material. They program the tool to start at a certain point and machine down by the specified amount using a series of passes. The G71 command removes material by feeding the tool along the Z axis during the cutting feed. G72 removes material by feeding the tool along the X axis during the cutting feed. Since the maximum travel for this part is 0.500″ along the X axis and 1.500″ along the Z axis, using the G71 code is the more efficient code in this case. See **Figure 24-11**.

The G71 code activates the repetitive cycle. A repetitive cycle can be called using either one or two lines of code. For simplicity, this example uses the two-line option. The first line is:

```
N0110  G71  U.125  R.063
```

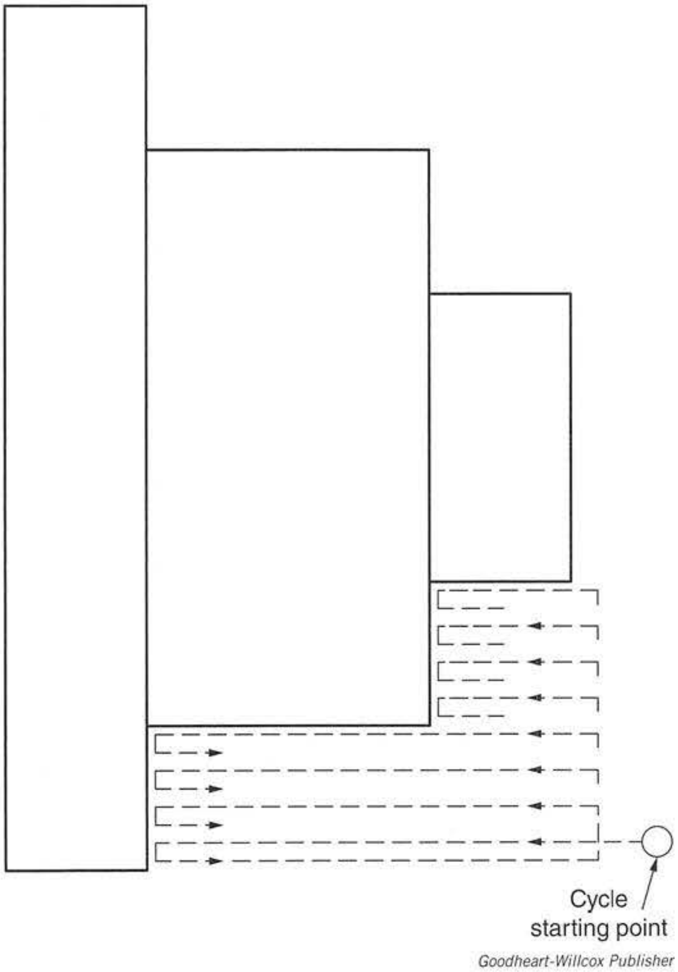


Figure 24-11. The tool path taken while performing a rough turning operation with the G71 repetitive cycle.

On this line, the letter U defines the amount of stock to be removed on each roughing pass, and the letter R defines the retracting distance for the tool after each pass.

SAFETY NOTE

The letter U will be used again in the next line of code, and it will mean something different. Do not get these two confused! The result could crash the machine.

The next line defines the rest of the repetitive cycle:

```
N0120   G71     P###    Q###    U.030  W.030 F.020
```

Line N0120 contains two unknown variables. The letter P defines the line of code that begins the cycle, and the letter Q defines the line of code that ends the cycle. Number (pound) signs can be inserted following the letters P and Q to fill in space until the lines of code for the beginning and end of the cycle are known.

The letters U and W on line N0120 contain offset values that define the amount of stock to be left for the finishing pass. The control references the lines of code containing the finishing pass using the U offset value for the X direction and the W offset value for the Z direction. The control automatically calculates the 0.030″ offset defined to leave the material for the finishing cycle. The letter F defines the feed rate in inches of feed per revolution of the part.

The G71 code requires a tool start point away from the part. In this example, the cycle will start from the position defined on line N0080.

Next, the finish profile must be defined. **Figure 24-12** shows the start point for the cycle and the start point for the final cut. Notice that the final cut begins at the end of the part at the point where the Z axis intersects the part.

First, position the tip of the cutter at the proper X axis dimension and just off the part in the Z direction:

```
N0130   G00     X0.0    Z0.0
```

Recall that during the facing operation, 0.050″ of material was removed from the end of the part, so now the tip of the cutter is 0.050″ from the end of the part at the Z = 0.000″ position. The programmer also plans to leave 0.030″ of stock for the finishing cycle. The next few lines of code specify the path of the cutter to define the final shape of the part.

```
N0140   G01     Z-.08   F.015
N0150   X-1.00
N0160   Z-.580
N0170   X-2.00
N0180   Z-1.580
N0190   X-3.01
N0200   G00     X-3.25  Z.25
```

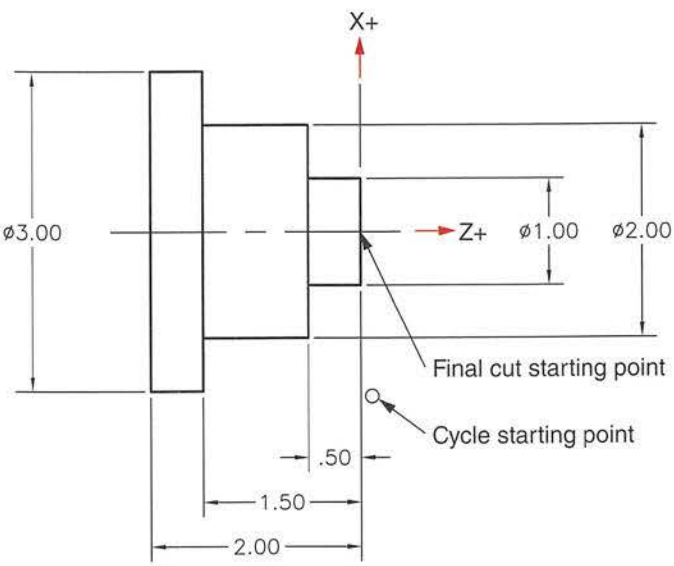


Figure 24-12. The cycle starting point defined by line N0080 and the starting point of the final cutting pass. The tool will retreat to the starting point when the cycle is complete. The starting point of the final cutting pass helps define the entire repetitive cycle.

The dashed line in **Figure 24-13** shows the cutter path produced by these lines of code. The space between the dashed line and the finished shape represents the 0.030″ of stock left from the stock removal cycle to be removed by the finishing cycle. In line N0190, the movement extends beyond the diameter of the part to ensure that the entire surface is machined. The cycle ends by returning the tool to the starting point.

Now that the finish profile has been defined and the lines have been numbered, go back and fill in the values for variables P and Q in line N0120:

```
N0120  G71    P0130  Q0190 U.030 W.030 F.020
```

Notice that the value for Q is 0190, not 0200. N0200 is not part of the final pass. It merely moves the tool to a clearance position. Also notice that the N in the line numbers is not used in the P and Q values.

24.4.4 Finish Turning Cycle

After the roughing cycle, it is necessary to change to a finishing tool before running the finish turning cycle. The next line of code is needed for the tool change:

N0210 M00 *(Manually change to a finishing tool)*

Line N0210 uses the M00 code to stop the tool for a manual change to the finishing tool. When the cycle start button is pressed, the spindle resumes rotation.

After the tool change, the finishing pass can be activated. The G70 code is used to activate a finish turning cycle, **Figure 24-14**. It uses the same start point, cycle start line number, cycle stop line number, and finish point as the roughing cycle.

```
N0220  G70   P0130  Q0190
N0230  G00   X-3.25  Z.25
```

Line N0220 starts the finishing cycle. The line numbers for the start and end of the cycle are the same numbers used in line N0120 for the roughing cycle. Line N0230 moves the cutter to a clearance position.

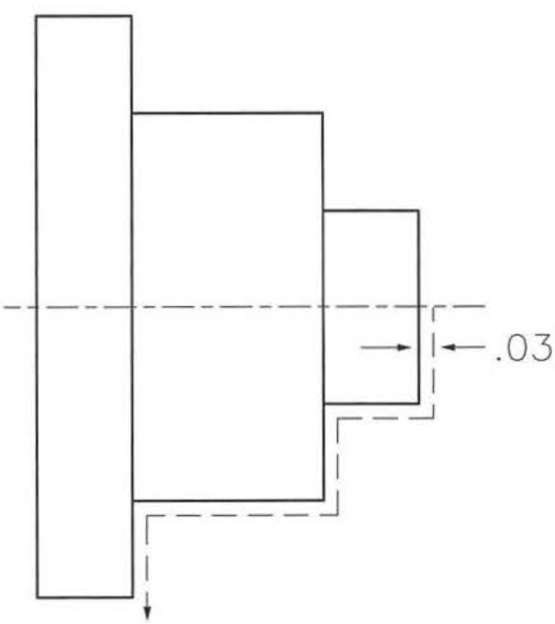
24.4.5 Cutoff Operation

The cutoff operation is the final step for many procedures performed on a CNC lathe or turning center, **Figure 24-15**. For the final machining step, the program is paused again so the tool can be changed to a parting tool. Then the finished part is cut away from the bar stock:

N0240 M00 *(Manually change to a parting tool)*
N0250 G00 X-3.050 Z-2.08
N0260 G01 X0.0 F.019

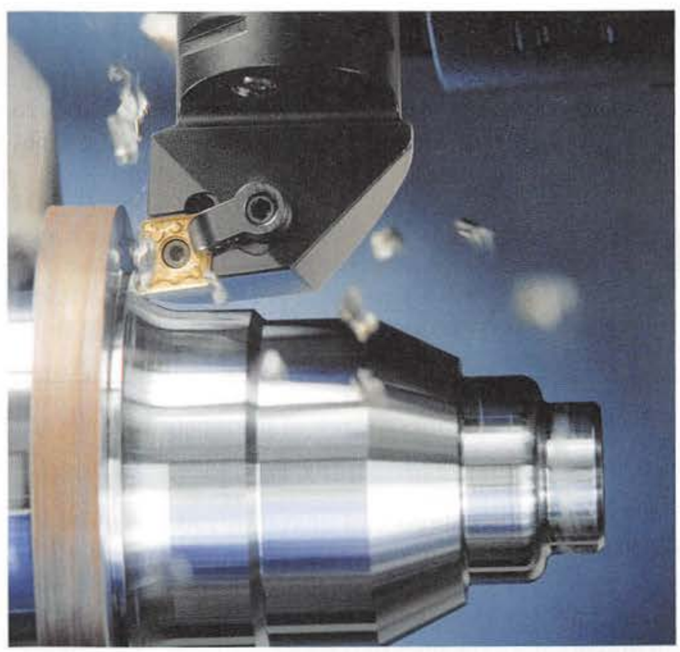
Line N0240 pauses the program for the tool change. Line N0250 uses rapid positioning to position the cutter for the final cut, and line N0260 performs the final cutting operation.

The G75 code can also be used to perform a cutoff operation. This code calls for a pecking cutoff or grooving cycle used for both inside and outside diameters. The G75 code requires an X axis coordinate (X), the peck distance (I), and the feed rate (F).



Goodheart-Willcox Publisher

Figure 24-13. The dashed line shows the cutter path defined by the finish profile. The 0.030″ gap between the part and the line represents the stock left to be removed by the finishing cycle.



Sandvik Coromant

Figure 24-14. Finishing cuts use fine feed and a shallow depth of cut to machine the workpiece to size and provide a good surface finish.

WORKPLACE SKILLS

Lifelong Learning

Companies look for employees who are eager to improve their skills, seek additional education and training, and keep up-to-date in their field of expertise. Continually updating your knowledge and skills is known as *lifelong learning.* The term implies that your need for learning will never end and you cannot assume that the skills you have will be all you will ever need during your career.

Obtaining industry certifications or credentials is a good way to pursue lifelong learning and advance your career. For example, the National Institute for Metalworking Skills (NIMS) offers credentials through an examination system based on set standards. Credentials from NIMS are available in several areas of machining, including CNC Milling and CNC Turning.

Employers usually provide some training or incentives to pursue training. In fact, some companies offer tuition reimbursement plans. However, employees may have to attend classes on their own time. People who enjoy their work will view lifelong learning as an exciting challenge.

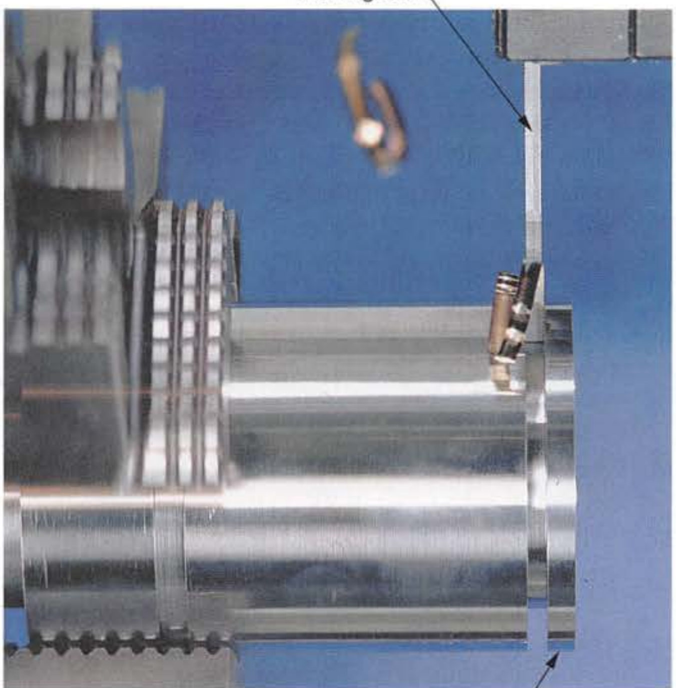


Figure 24-15. The parting tool used to separate the finished part from the bar stock should be as short as possible.

24.4.6 Ending the Program

The last two lines finish up the program:

```
N0270  G00   X-3.25  Z.25 M05
N0280  M30
```

Line N0270 uses rapid positioning to move the cutter out of the way and stops the spindle. The M30 code in line N0280 ends the program and resets it to the beginning. The complete code is shown below:

```
N0010  G00  G53     G20 G90
N0020  G54  X-3.25  Z.25
N0030  M05
N0040  M00 (Manually change to a roughing tool)
N0050  G97  M03     S600
N0060  G00  Z-.050
N0070  G01  X0.0    F.019
N0080  G00  X-3.05  Z.05 (End of facing operation)
N0090  G50  S3000   (Max rpm while in CSS mode)
N0100  G96  S600    (Set CSS at 600 and activate)
N0110  G71  U.125   R.063
N0120  G71  P0130   Q0190  U.030  W.030 F.020
N0130  G00  X0.0    Z0.0
N0140  G01  Z-.080  F.015
N0150  X-1.00
N0160  Z-.58
N0170  X-2.00
N0180  Z-1.580
N0190  X-3.01
N0200  G00  X-3.25  Z.25
N0210  M00 (Manually change to a finishing tool)
N0220  G70  P0130   Q0190
N0230  G00  X-3.25  Z.25
N0240  M00 (Manually change to a parting tool)
N0250  G00  X-3.050 Z-2.08
N0260  G01  X0.0    F.019
N0270  G00  X-3.25  Z.25   M05
N0280  M30
```

Chapter Review

Summary

- Work-holding devices for CNC turning centers are similar to those used for manual lathes and may include chucks and collets, depending on the application.
- The first step in creating a CNC turning program is to identify the processes and tools needed for the operation.
- CNC lathes and turning centers may use two axes with a cross-slide or three axes on turning centers in which tools are held in a turret.
- Before writing the program, the programmer must determine whether the machine is set up to accept radius or diameter input data.
- The initial lines of a program for a turning center set the machine offsets, position the cutter, and perform any other necessary tasks, such as stopping the program for a manual tool change.
- Stock removal is performed in multiple steps using repetitive cycles.
- The final step in a turning procedure is to cut the finished part away from the bar stock.

Review Questions

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

1. What type of work-holding device is most likely to be used in a production setting in which large volumes of parts are being manufactured?
2. What two things must be determined before a programmer can begin writing a program for a CNC turning center?
3. Rough cuts and finish cuts can be performed by the same tool. Why do many machinists prefer to use two separate tools?
4. Along which axis of the Cartesian coordinate system is the cylindrical axis of the part aligned?
5. Which axis of the Cartesian coordinate system is typically not needed for two-axis CNC lathes equipped with a cross-slide?
6. Why is it important to know whether a machine is set up to take radius or diameter input before programming?
7. The diameter of the _____ is used in the spindle speed formula for a lathe or turning center.
8. Where is the machine zero position typically set for a CNC lathe?
9. Why do many programmers automatically insert the M05 code before pausing the program?
10. Which code would you use to pause the program so that a tool can be changed manually?
11. What action is required to restart the machine after a pause and return it to the state it was in when the program paused?
12. Stock removal cycles are a sequence of steps called _____ that make several passes to cut away the defined amount of material.
13. Which code removes material by feeding the tool along the Z axis during a cutting feed?
14. Which code removes material by feeding the tool along the X axis during a cutting feed?
15. In the following line of code, how much stock is to be removed in each pass, assuming the program is set up to use inches?
    ```
    N0130 G71 U.130 R.087
    ```
16. When a G71 code is used, what do the letters P and Q specify?
17. In the following line of code, what does the number after the U specify?
    ```
    N0320 G71 P0240 Q0310 U.030 W.025 F.015
    ```
18. What is commonly the final step in the procedure to machine a part using a CNC lathe or turning center?

CHAPTER 25

Automated Manufacturing

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Define the term *automation.*
- Summarize the traits of a flexible manufacturing system.
- Explain the advantages of using robots to perform certain tasks in a manufacturing application.
- Identify the safety issues associated with automated manufacturing.
- Explain the similarities and differences among various rapid prototyping techniques.
- Discuss the future of automated manufacturing.

Technical Terms

additive manufacturing
computer integrated manufacturing (CIM)
direct shell production casting (DSPC)
flexible manufacturing system (FMS)
fused deposition modeling (FDM)
just-in-time (JIT) inventory system
laminated object manufacturing (LOM)
lean manufacturing work cell
programmable logic controller (PLC)
robot
smart tooling
stereolithography
work envelope

A*utomation* is a term coined in 1947 by a Ford Motor Company engineer. It is not a revolutionary new form of manufacturing but a method of manufacturing that has evolved over many years. Automation is a system for the continuous automatic production of a product, **Figure 25-1**. It relies on a machine or group of machines activated electronically, hydraulically, mechanically, or pneumatically (or a combination of these means) to automatically perform one or more of the five basic manufacturing processes:

- Making.
- Inspecting.
- Assembling.
- Testing.
- Packaging.

The principles of automation have been known for many years. An automated flour mill was in operation in the late 1700s near Philadelphia. It was able to continuously mill grain into flour. The mill used many of the elements found in modern automated operations.

The integration of the computer, **Figure 25-2**, with specially designed machine tools and equipment, has revolutionized production technology, resulting in increased productivity, improved product quality, and reduced manufacturing costs. Automation has also minimized human involvement in many phases of the manufacturing process.

Hirata Corporation of America

Figure 25-1. A robotic system used to assemble the components of automobile engines is an example of an automated system.

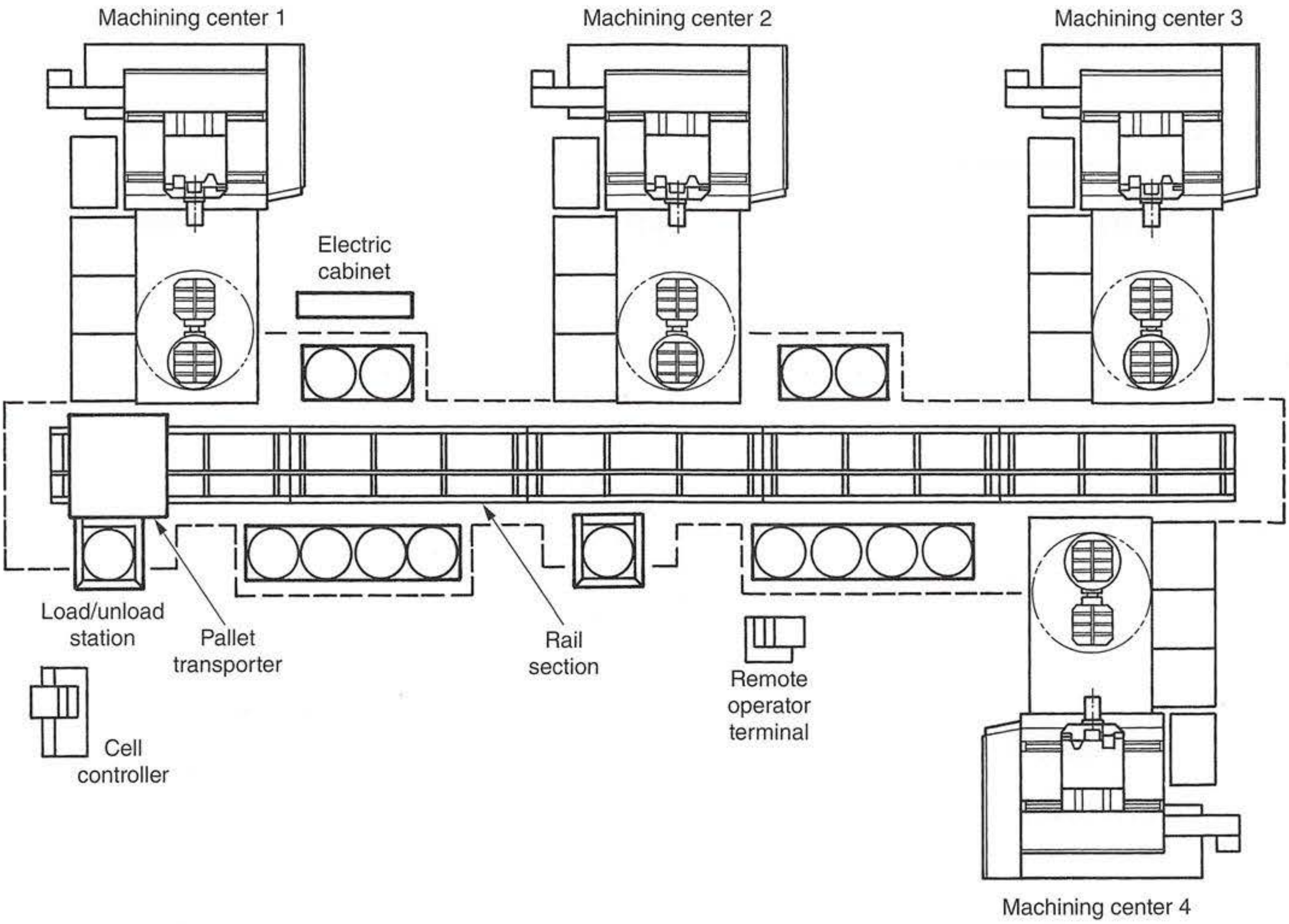
25.1 Flexible Manufacturing Systems

Due to its extensive use of computer-controlled machinery and adaptive tooling, a ***flexible manufacturing system (FMS)*** can be quickly adapted to changes in the product or the manufacturing process. This general category of machining/manufacturing technology is also widely referred to as ***computer integrated manufacturing (CIM)***. A flexible manufacturing system is made up of one or more groups of machines that are used to perform multiple operations automatically. Such a group of machines is often called a *flexible manufacturing cell (FMC)*.

A flexible manufacturing cell brings together workstations (machine tools), automated handling and transfer systems, and computer control in an integrated manner, **Figure 25-3**. It is capable of performing multiple manufacturing actions simultaneously. Work is transferred to and from the individual machines in the flexible manufacturing cell by automated fixture carts, conveyors, or specially designed loaders, **Figure 25-4**. Robots may also be employed in some operations.

Manufacturing Technology, Inc.

Figure 25-2. The integration of computers and specially designed machine tools and equipment into manufacturing systems has revolutionized automation technology. Computerized systems improve product quality and reduce manufacturing costs.



Cincinnati Milacron

Figure 25-3. Flexible manufacturing cell that uses a pallet transporter to link the machines. A cell controller automatically queues work for immediate delivery to the next machine available.

Westech Products Group/Gantrex Machine Tool Loaders

Figure 25-4. A conveyor (foreground) transfers workpieces to an unattended horizontal machining center in this manufacturing cell.

An improvement to the straight-line work area layout is the ***lean manufacturing work cell*** design. This design was developed as part of the Toyota Production System, with a U-shaped layout that allows fewer operators to efficiently operate smaller, automated equipment. See **Figure 25-5.** The smaller layout and the close proximity of the machines enable parts to flow through the work cell one at a time, with a minimal amount of inventory building up between operations. This layout also allows for the processing of smaller batches of equipment by using simpler machines that allow for faster changeover times between different product lines.

The machine tools typically used in flexible manufacturing systems are CNC vertical and horizontal machining centers. Other CNC machine tools, such as grinders and automatic gaging equipment, may also be included in flexible manufacturing systems.

Each step of the manufacturing process is computer-controlled and is linked to the next step. This link is created using ***programmable logic controllers (PLCs).*** A PLC is a small, digital computer that acts as a communications hub for flexible manufacturing cells. Using various sensors and

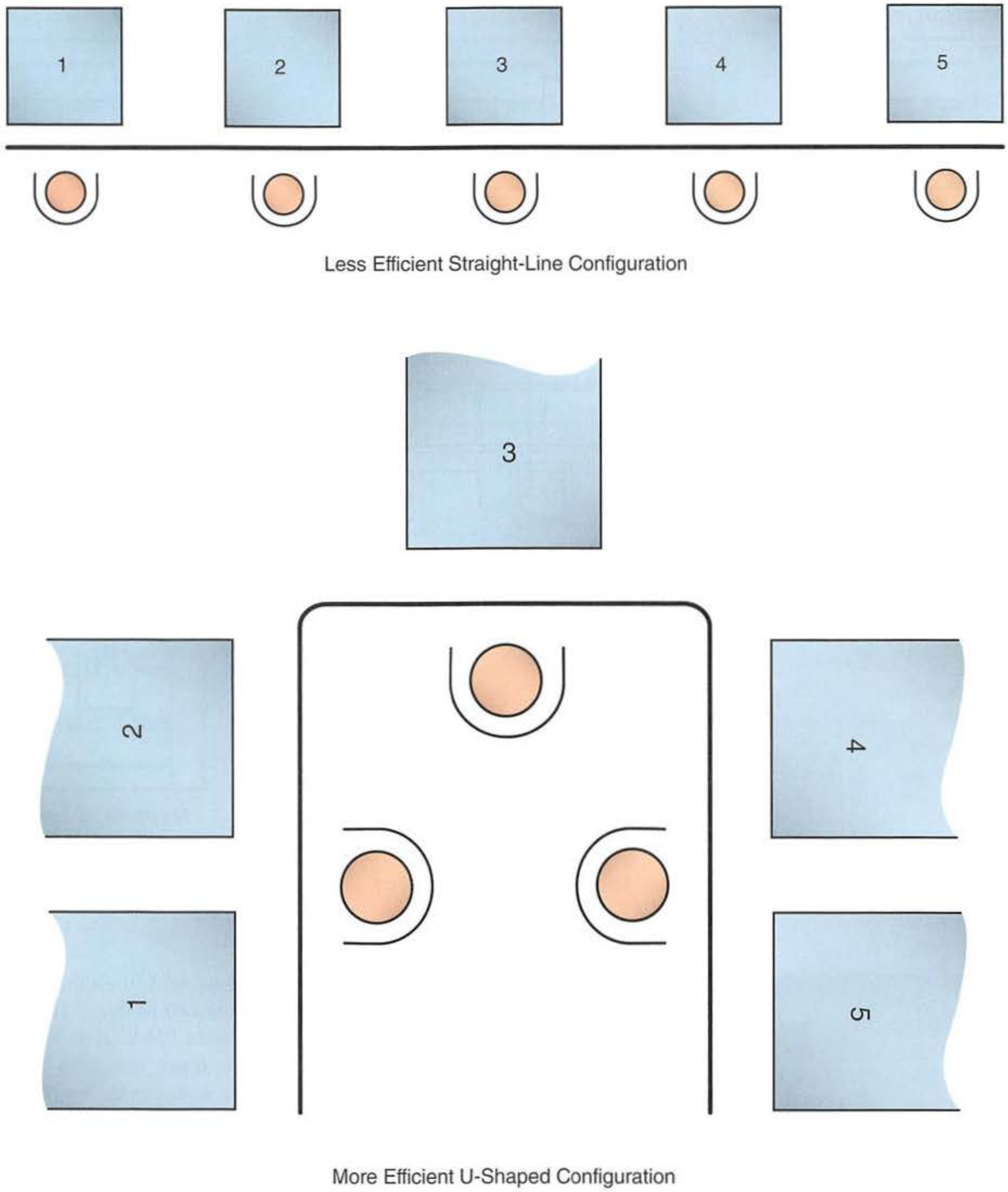


Figure 25-5. The lean manufacturing work cell concept is built around a U-shaped work area, which places workers in each cell in close proximity to improve the efficiency of work flow and minimize inventory needs.

machine-specific codes from the CNC equipment, the PLC manages the communications process between different machines (such as CNC machines, robots, and conveyance systems) within a flexible manufacturing cell. Measuring sensors, often utilizing lasers, are sensitive enough to detect tool wear as it occurs and automatically compensate for it.

Nevertheless, every part is inspected individually. Problems such as tool malfunction or tool breakage are immediately identified so corrections can be made before additional parts are produced that do not meet specifications.

Flexible manufacturing systems can also make use of several other technologies. These include smart tooling; just-in-time (JIT) inventory of materials, parts, and subassemblies; and robots.

Smart tooling consists of cutting tools and work-holding devices that can be easily reconfigured to produce a variety

WORKPLACE SKILLS

Computer Ethics

In most jobs today, the computer is an essential tool. The employer provides a computer for your use as a tool for research or to accomplish tasks. It is unethical to use the computer, without permission, for personal activities, such as playing games, shopping, or other activities that are outside of your assignments. Using your computer for personal activities while on the clock can be considered stealing company time.

Organizations commonly use social media to reach existing customers and find new ones. Those who are writing communications for the organization must be prudent when using sites, such as Facebook or Twitter, for business purposes. Keep your messages honest, and return messages from those who have taken time to respond to your communication. Use good judgment and represent the organization in a professional manner.

It is also unethical to access confidential information, download copyrighted material, or harass others while using company equipment. Unapproved use of computers may open up the computer network for viruses and other issues that may jeopardize the integrity of the network.

Many organizations monitor computer users to make certain computer activity is ethical and legal. Users may also be required to sign an agreement that the computer will only be used for specific purposes.

of shapes and sizes within a given part family. This makes it economically feasible to manufacture products in smaller lot sizes.

A ***just-in-time (JIT) inventory system*** is an inventory management system that eliminates the need to store large quantities of materials and parts. The parts and materials are scheduled for arrival at the time they are needed, and not before. While JIT inventory systems lessen (and often eliminate) storage costs, production can be reduced or stopped if the delivery of the needed items is delayed by bad weather or strikes.

25.2 Robotics

The Robotic Industries Association (RIA) has adopted the following definition for industrial robots: "A ***robot*** is a reprogrammable, multifunctional manipulator designed to move material, parts, tools, or specialized devices through variable programmed motions for the performance of a variety of tasks." Robots can be programmed to do many types of jobs that are hazardous or tedious for human workers. Spot welding, paint spraying, and loading parts onto machines are examples of these jobs, **Figure 25-6**.

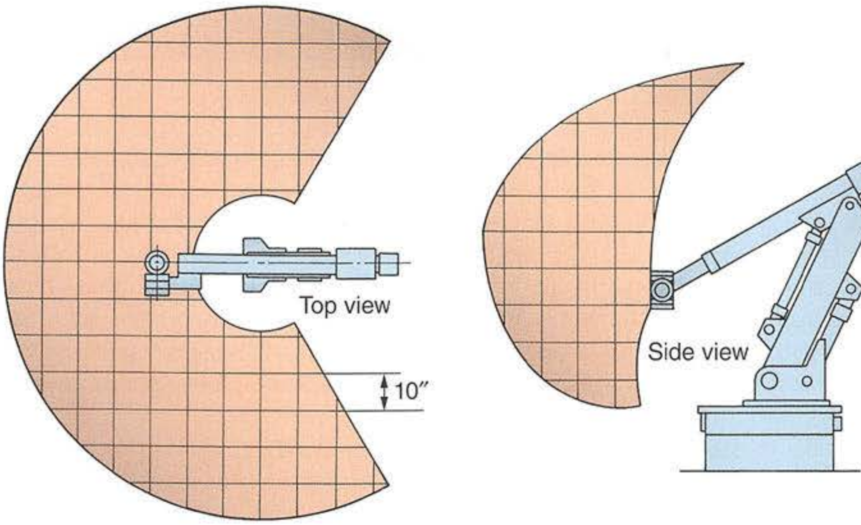
U.S. Amada, Ltd.

Figure 25-6. Robots can be programmed to do many types of jobs that are hazardous or tedious for human workers. This robot is paired with a press brake to load and remove sheet metal parts.

25.2.1 Robotics in Automation

The RIA definition of an industrial robot contains two implied requirements:

- The robot must be a multiaxis machine (three or more axes) that is capable of moving parts or tools to any given location within its work envelope with a high degree of repeatability and precision. The ***work envelope*** of a robot is the volume of space defined by the reach of the robot arm in three-dimensional space. See **Figure 25-7**.
- There must be a control system that can be programmed to drive the robot through a series of specified motions and be capable of interacting with other machines and equipment.



Goodheart-Willcox Publisher

Figure 25-7. The work envelope of a robot is the volume of space defined by the reach of the robot arm in three dimensions.

GREEN MACHINING

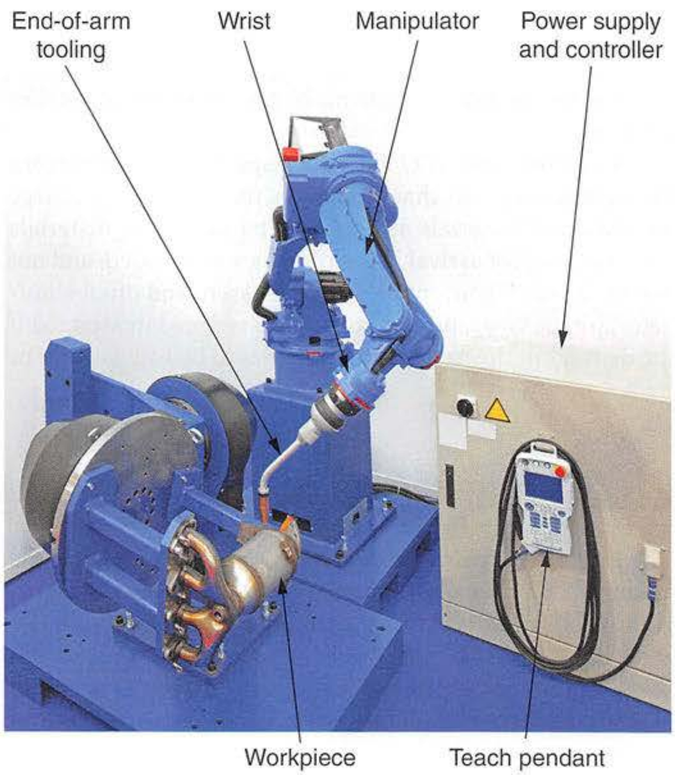
Leadership in Energy and Environmental Design (LEED)

Established by the US Green Building Council, LEED is the industry standard in green construction. LEED rates green building practices in areas including energy conservation, building materials, water use, and air quality. Green practices are evaluated from construction through the life of the building project. Buildings must meet certain criteria to achieve LEED certification, which offers numerous benefits. LEED certification can boost public image and qualify buildings for certain tax benefits and zoning allowances. Green practices are becoming more prevalent in factories where smart choices for the environment can also promote efficiency, thereby reducing cost. Green machining practices and construction methods are an important part of LEED alongside recycling and environmentally friendly cleaning methods.

25.2.2 Industrial Robot Design

As shown in **Figure 25-8**, an industrial robot consists of four basic components:

- **Controller.** Performs computations for controlling the movement of the waist, arm, and wrist to the proper location.
- **Manipulator.** The articulated "arm" of the robot. The end of the arm is fitted with a "wrist" capable of rotational motion around one or more axes.
- **End-of-arm tooling.** A device attached to the robot wrist for specific applications, such as a gripper, welding head, or spray gun. Grippers can be physical, vise-like manipulators, as well as magnetic or vacuum-powered grasping devices.



Baloncici/Shutterstock.com

Figure 25-8. An industrial robot consists of four basic parts: a controller, a power supply, a manipulator, and end-of-arm tooling. The end-of-arm tooling on this robot is a gas metal arc welding gun.

- **Power supply.** Supplies the power to operate the robot. May be hydraulic, pneumatic, or electric.

Robots that are designed to lift relatively light loads are typically powered by electric drive systems. Pneumatic robots can handle medium weight loads, while heavier loads

require the use of hydraulic power systems. However, weight limits are not the only factors considered in selecting the proper robot. The maximum reach of the arm while carrying a load is critical, because the weight limits are established according to how much the robot can handle precisely at its maximum reach. Less precise operations can be performed with electric or pneumatic powered units. However, if the weight is heavy and the required precision is tight, hydraulic systems are preferred.

Cost is another factor companies must consider when purchasing a robot. Electric robots are least expensive, and hydraulic units are most expensive.

25.2.3 Industrial Robot Applications

Manufacturers have replaced human workers with industrial robots in a number of different applications to increase worker safety, to increase capability, or both. Industrial robots are often used in the following applications:

- **Hazardous and harsh environments.** Fumes produced by spray painting and welding, heat in foundry or forging operations, or crush hazards when feeding material into punch presses. See **Figure 25-9.**
- **Tedious operations.** Repetitive operations such as feeding, loading, and unloading of parts and materials, and some assembly operations. See **Figure 25-10.**
- **Precision operations.** Precisely repeated positioning operations, maintaining consistent tool speed, following complex welding and cutting paths without patterns, and quality control using lasers.
- **Handling heavy or unwieldy materials.** Lifting material onto or from a stack, moving material beyond the normal reach of a human, or mounting heavy or bulky workpieces on machine tools, **Figure 25-11.**

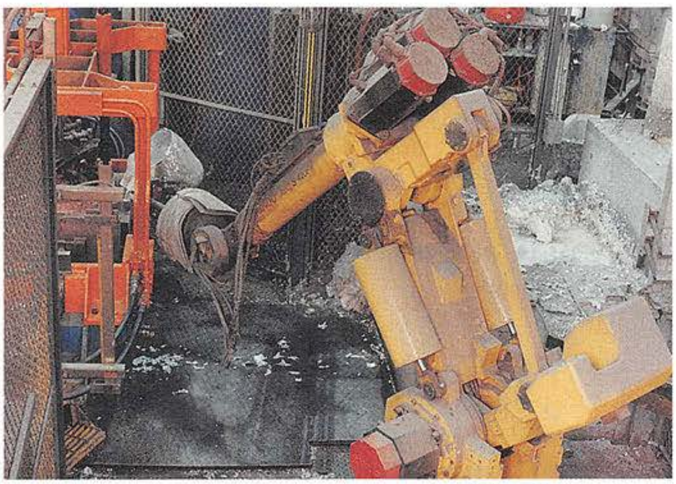
Many types of robots have been developed, but almost all can be classified into one of the four basic geometric configurations shown in **Figure 25-12.** Advanced technology robots can be programmed to serve a wide range of automated manufacturing applications. They can be interfaced with testing devices to perform various types of measurements.

Some robots are capable of selecting and positioning complex parts for machining or storage. A laser is utilized to "see" and define the part outline so the correct item can be selected and positioned for machining.

25.3 Safety in Automated Manufacturing

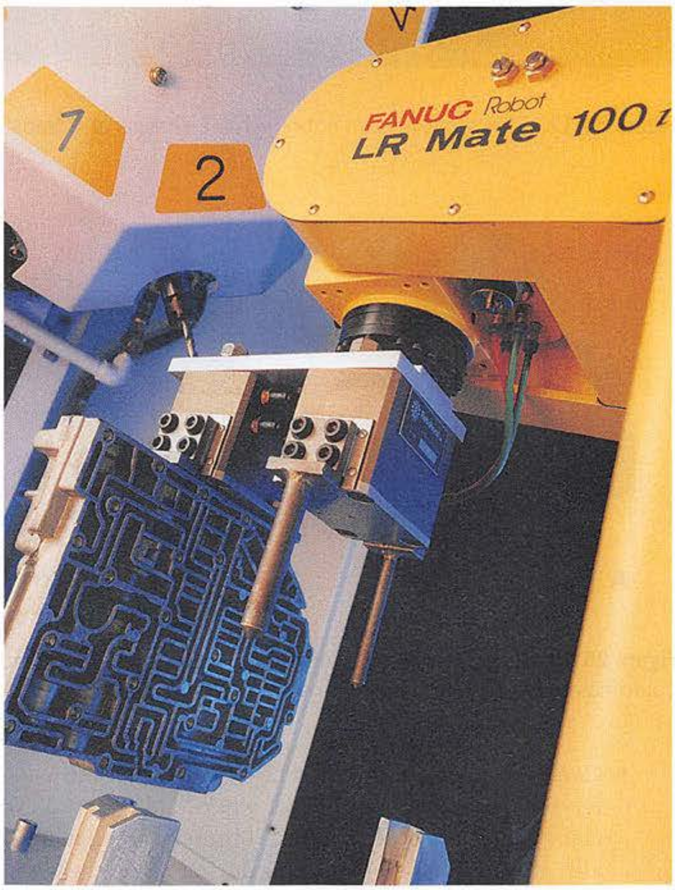
When working with an automated manufacturing system, follow these safety guidelines:

- Observe the same safe operating procedures that are used with traditional machine tools and machining.
- Wear approved eye protection and a snug-fitting apron or shop coat.
- Be sure work-holding devices are positioned correctly and are securely fastened to the worktable.



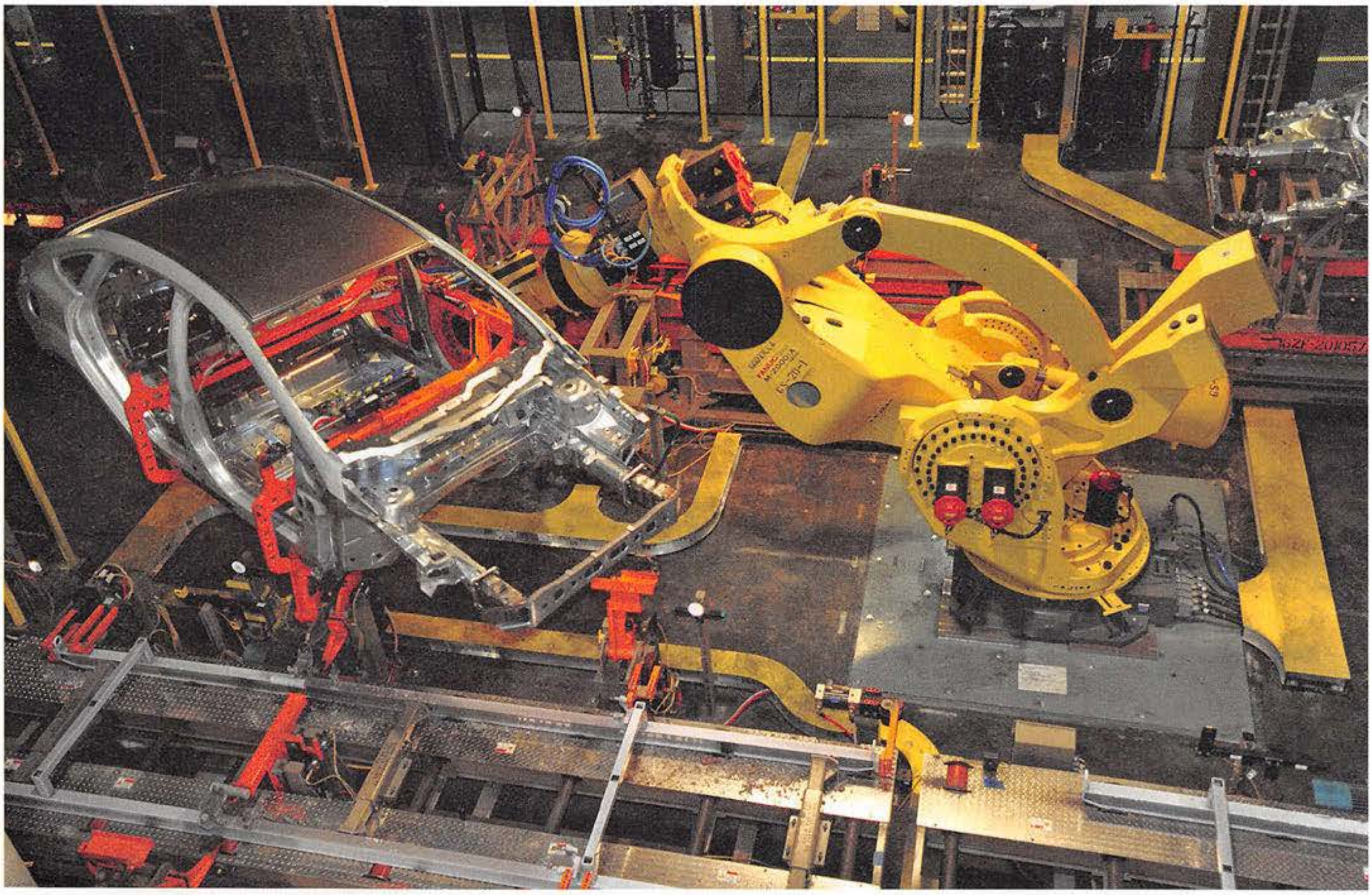
Fanuc Robotics North America, Inc.

Figure 25-9. A robot ladling molten aluminum into a wheel mold.



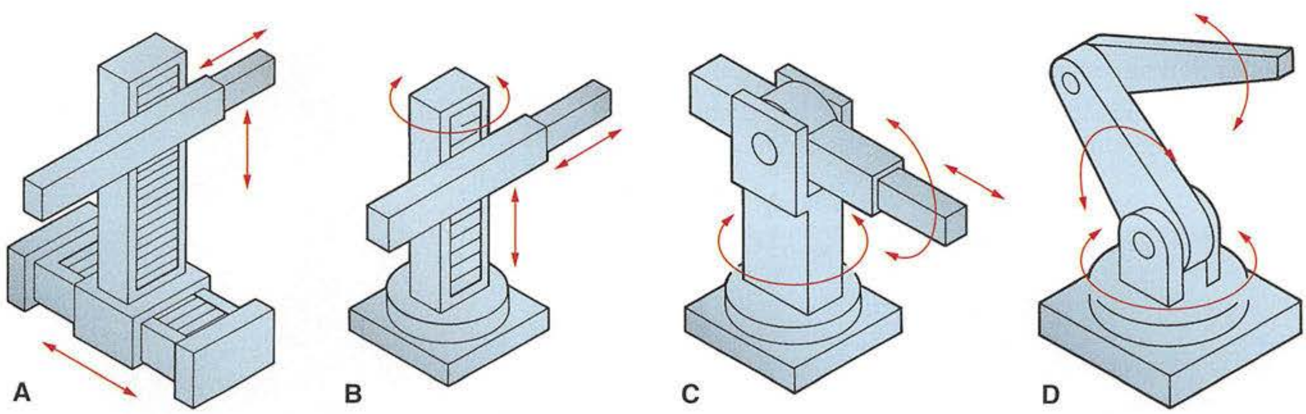
Fanuc Robotics North America, Inc.

Figure 25-10. This robot is being used to load and unload castings from a drilling machine.



Ford Motor Co.

Figure 25-11. This robot lifts an entire vehicle chassis and transfers it from one assembly line to another.



Goodheart-Willcox Publisher

Figure 25-12. Basic geometric configurations of robots. All provide three articulations (specific arm movements). A—Cartesian coordinates. B—Cylindrical coordinates. C—Polar coordinates. D—Revolute coordinates.

- Carefully check tool clearance to be sure the cutter will clear the workpiece, work-holding devices, clamps, and any other potential obstacles when manually positioning the work and during rapid traverse. Make a "dry run" for a safety check of tool positioning for each machining operation.
- Never enter the work envelope of a robot until you are sure power is turned off.
- Establish that the machine tool and control unit are functioning correctly, and continually monitor machining operations to be sure each tool is cutting

properly. Know how to safely stop machining operations in case of an emergency.

- To avoid injury, remove burrs and sharp edges before inspecting the accuracy or the finish of machined surfaces on finished parts.

25.4 Rapid Prototyping Techniques

A number of techniques allow designers to quickly generate three-dimensional models or prototypes of parts in relatively inexpensive materials. These techniques help to improve design and identify potential machining problems. They include laminated object manufacturing, stereolithography, fused deposition modeling, and direct shell production casting, among others.

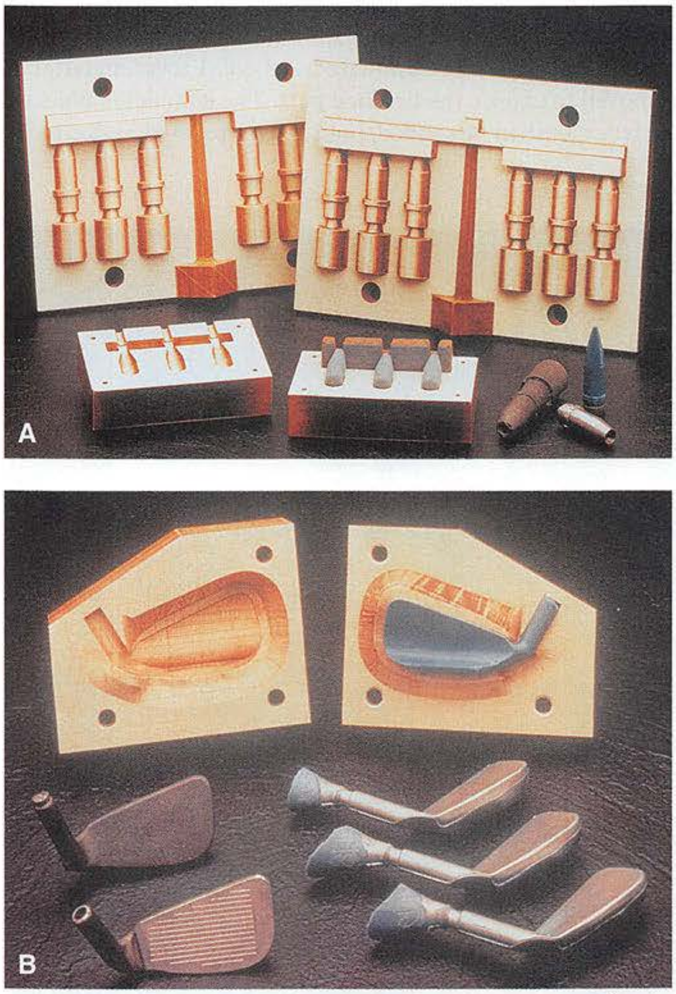
Rapid prototyping, or three-dimensional modeling, is a process that allows designers to create physical models from computerized designs. Some people try to distinguish between the terms *rapid prototyping, 3D modeling,* and *additive manufacturing.* In truth, there is very little difference. 3D modeling and rapid prototyping are often used interchangeably, but 3D modeling can be applied to the development of three-dimensional models inside a parametric modeling software, such as Autodesk® Inventor, SOLIDWORKS®, or Pro/ENGINEER®. Many of these software packages provide the ability to test designs through computer simulation to help designers and engineers determine the optimum design(s) before they invest time and money into the tooling and materials needed for prototypes.

Rapid prototyping got its start from the need to produce a physical part from a computerized 3D model. This provided designers and engineers with the ability to create multiple parts that could be tested for fit, form, and function, again before laying out large investments to make functioning prototypes. This form of rapid prototyping was a step beyond the computerized model in that it improved the ability to visualize the components more effectively than by simply viewing the CAD display. Over the years, the technology for creating prototypes has advanced, along with the materials that can be used to create the physical models. Today, some prototypes can be made to withstand physical, thermal, and chemical environments, allowing the prototypes to be thoroughly tested before finalizing the part designs.

Rapid prototyping processes have developed beyond making plastic models of parts to making the parts themselves. This process, called ***additive manufacturing,*** uses fused deposition technologies to build parts in layers just as in rapid prototyping processes. This process is cost-prohibitive for many high-volume processes in which current technologies can produce more parts per time period. Also, additive manufacturing cannot yet deliver goods that match the physical qualities of similar goods created using current technologies. However, as newer technologies become capable of higher levels of precision, and capable of applying the technology to more advanced materials, the process of additive manufacturing will continue to advance.

25.4.1 Laminated Object Manufacturing (LOM)

Laminated object manufacturing (LOM) is a process used to rapidly produce models, prototypes, accurate patterns, and molds. See **Figure 25-13**. This three-dimensional modeling method was developed in 1991 by Helisys, Inc. Since then, many automotive, aerospace, appliance, and medical product manufacturers have adopted the process. The equipment is compact and can operate 24 hours a day.



Helisys, Inc.

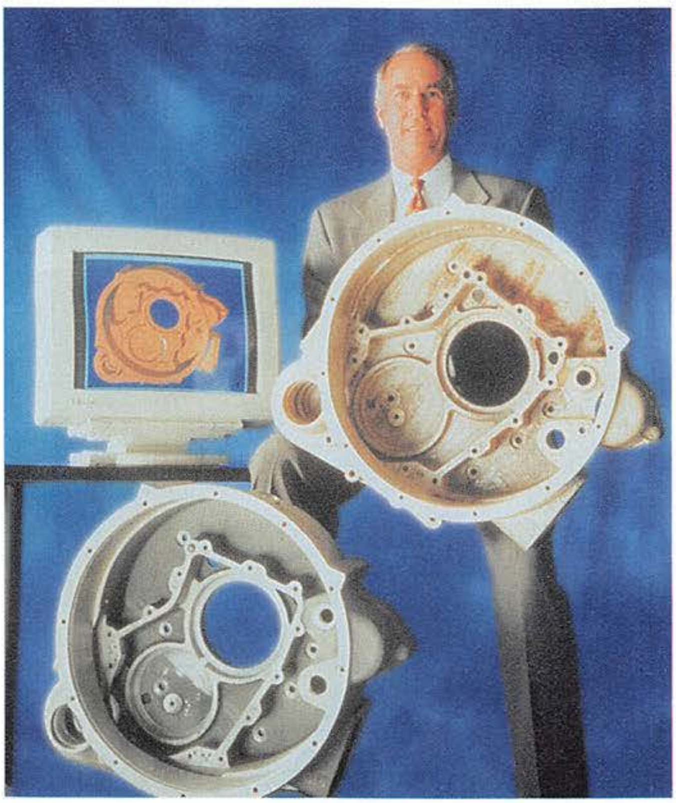
Figure 25-13. Laminated object manufacturing. A—LOM was used to produce these extremely accurate, full-size matchplates (patterns) and core box to be used in casting prototype metal projectiles. B—Rapid prototyping of the molds to make the wax patterns for the investment casting of oversized golf club irons was done with LOM. Design changes could be carried out before making the expensive molds necessary for quantity production.

Uses for LOM Parts

LOM uses inexpensive solid sheet material, such as paper, plastic, and composites, to form the desired designs. Molds made from the composite materials can be used in direct tooling applications for processes such as injection molding, vacuum forming, and sand casting. See **Figure 25-14**. LOM parts are accurate enough for form-and-fit verification applications. The finished part has a composition similar to wood and can be easily machined or modified to obtain the exact fit required. Modifications and corrections can be incorporated into the CAD design before manufacturing.

Operation of the LOM System

CAD data is programmed into the LOM's process controller. A cross-sectional slice is generated, and a laser cuts the outline of the cross section, then crosshatches the excess material for later removal, **Figure 25-15**. A new layer of material is bonded to the top of the previously cut layer. The next cross section is prepared and cut. This automatic process continues until all layers are laminated and cut. Excess material is removed to expose the finished part. The completed object's surface can then be sanded, polished, or painted as desired.



Helisys, Inc.

Figure 25-14. The CAD drawing on-screen was used to produce the LOM model (being held). That model was then used to produce a mold. The mold was used to produce an aluminum sand casting of the prototype rear-engine power takeoff housing for a diesel truck engine. This technique cut development time in half.

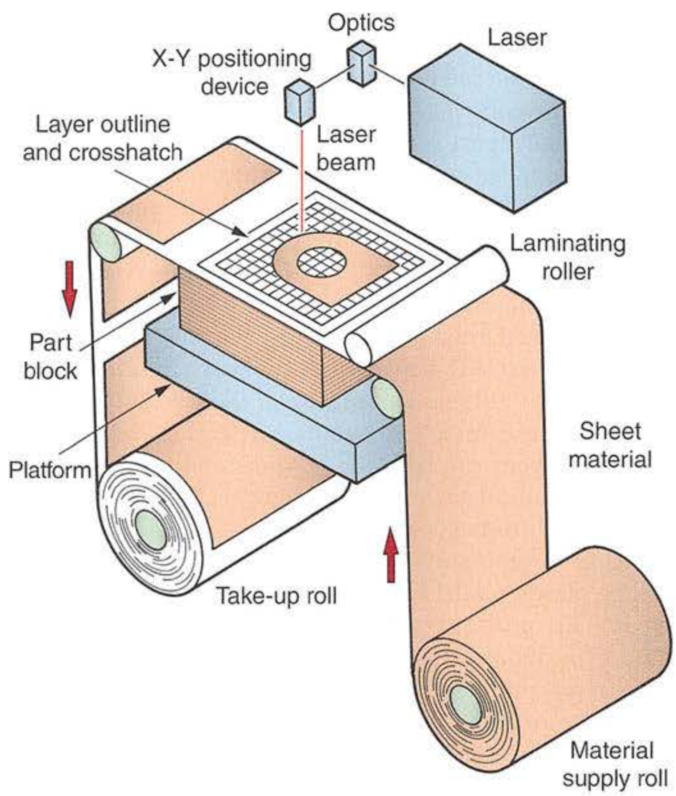
25.4.2 Stereolithography

Another rapid prototyping technology, called ***stereolithography***, can produce complex design prototypes of castings and other objects in hours, instead of days or weeks, **Figure 25-16**. The three-dimensional hard plastic models produced by the stereolithography process can be studied to determine whether they are the best solution to a design problem. Since new models can be made quickly, design changes and modifications can be evaluated without the expense of making new patterns or molds.

The stereolithography process uses a computer-guided low-power laser beam to harden a liquid photocurable polymer plastic into the programmed shape. The process starts by creating the required design on a CAD system and orienting it in three-dimensional space. A support structure is added to hold the various elements in place while the model is built up.

The design data is downloaded into the stereolithography machine, which operates somewhat like a CNC machine tool. A program within the stereolithography machine's controller slices the design into cross sections of 0.005″ to 0.020″ (0.12 mm to 0.50 mm) in thickness. The machine's control unit guides a fine laser beam onto the surface of a vat containing the liquid photocurable polymer. The liquid solidifies wherever it is struck by the laser beam, **Figure 25-17**.

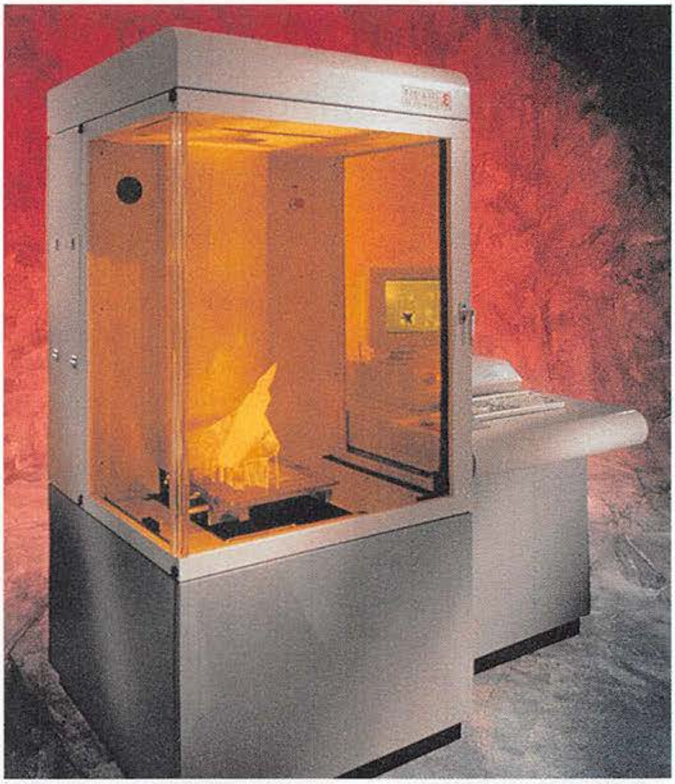
The model is created from the bottom up, on a platform located just below the surface of the liquid plastic. After each



Goodheart-Willcox Publisher

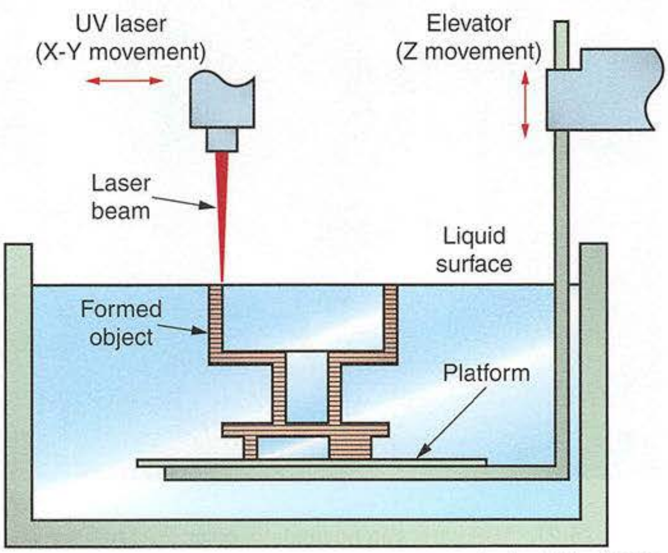
Figure 25-15. The Laminated Object Manufacturing process.

"slice" is formed, the platform drops a programmed distance. The sequence is repeated until the entire model is formed, **Figure 25-18**. The model is removed from the platform and rinsed in a chemical bath to remove excess resin. The model is then placed in an ultraviolet oven for curing. When the model has cured, the support structure is clipped away. The model can then be finished by filing, sanding, and polishing. Paint or dye can be applied. The entire stereolithography process is shown in **Figure 25-19**.



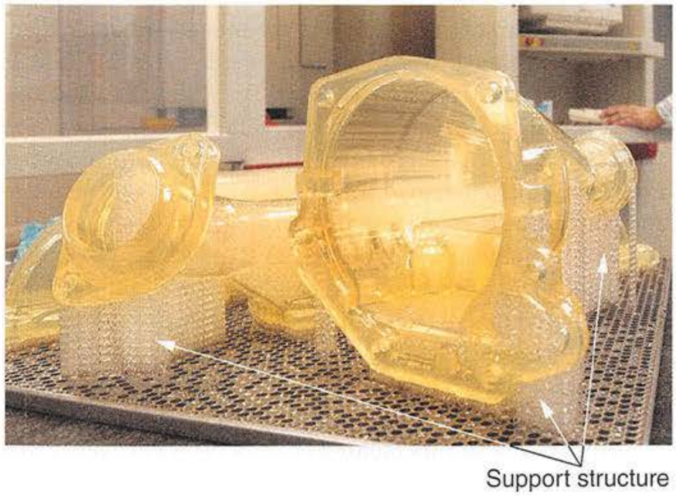
3D Systems

Figure 25-16. Stereolithography equipment is used to produce durable, finely detailed patterns that are extensively used in modeling, functional prototyping, and tooling applications.



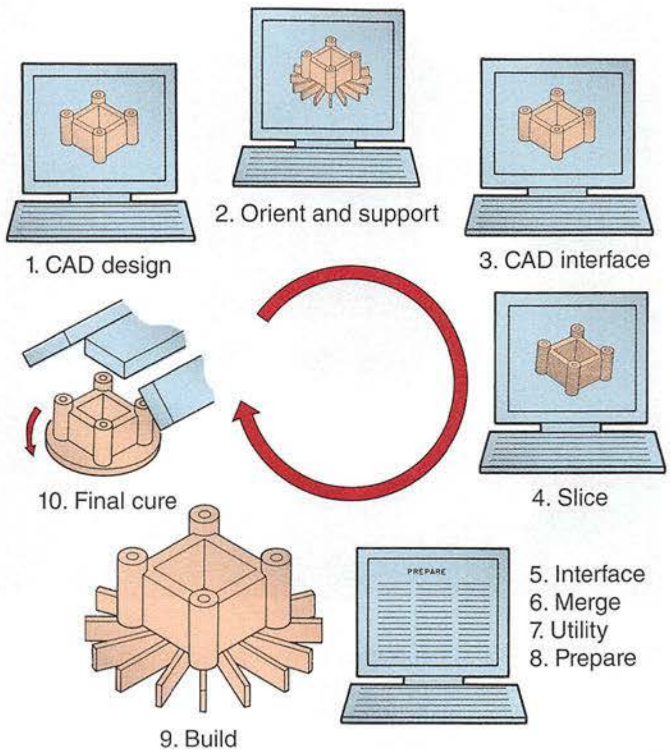
Goodheart-Willcox Publisher

Figure 25-17. Basic diagram of the stereolithography process.



Ford Motor Co.

Figure 25-18. A model created using the stereolithography process. Note the support structures on the model. Once the model is cured and the support structures are removed, this model will be used to train assembly robots. By training the robots with accurate prototypes rather than waiting for production tooling, the plant can greatly reduce setup time.



Goodheart-Willcox Publisher

Figure 25-19. The stereolithography process.

25.4.3 Other Rapid Prototyping Techniques

Rapid prototyping techniques that operate on a system similar to stereolithography can produce fully functional prototypes made of ABS (acrylonitrile butadiene styrene), medical ABS, or investment casting wax. When made of investment casting wax, the three-dimensional models can be used as the master for a cast part.

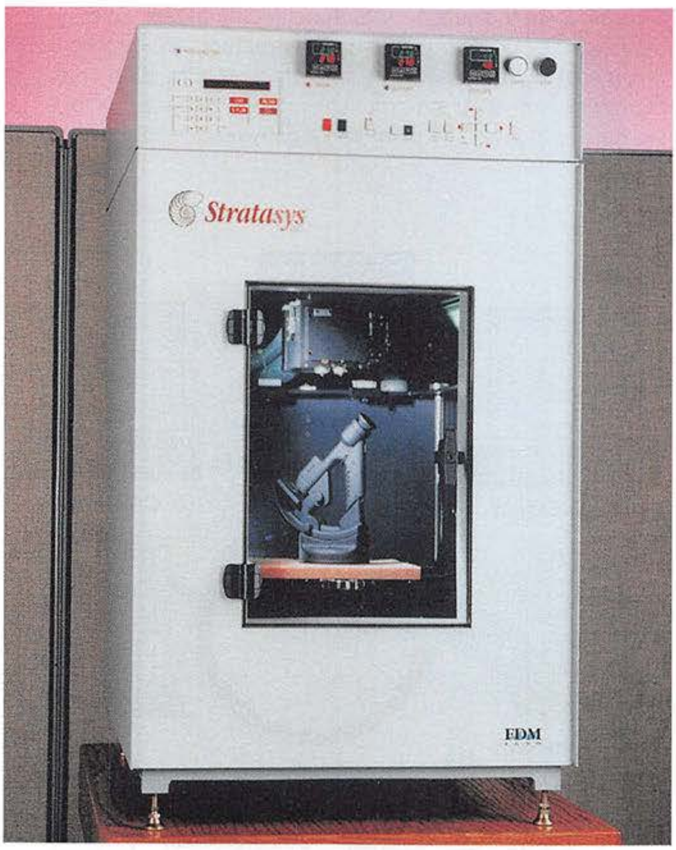
A rapid prototyping process called ***fused deposition modeling (FDM)*** can produce fully functional prototypes, **Figure 25-20**. When made of ABS, the parts can be installed and run for the best proof that a design works. The ABS parts can also be made in color, **Figure 25-21**.

The FDM process can produce three-dimensional objects based on CAD-generated solid or surface models. A temperature-controlled head extrudes thermoplastic material layer by layer. The designed object emerges as a three-dimensional part without tooling.

A variation of this technology has been developed that produces parts made of sand or ceramic material instead of plastic. Layers of material are built up, using an inkjet-style printer to spray a quick-hardening binder to solidify each layer. The technology has been used to quickly produce shell molds for casting metals, such as Inconel® and aluminum. The mold-making process is called ***direct shell production casting (DSPC)***. See **Figure 25-22**.

25.5 The Future of Automated Manufacturing

With technological advances being made at such a rapid pace, only time will tell what the full effect of computers and automated manufacturing techniques will be on our society. Some workers, mostly unskilled and semiskilled, have lost their jobs, much like the home artisans did during the Industrial Revolution. The same thing happened to carriage makers, blacksmiths, buggy whip manufacturers, and feed dealers when Henry Ford started to mass-produce

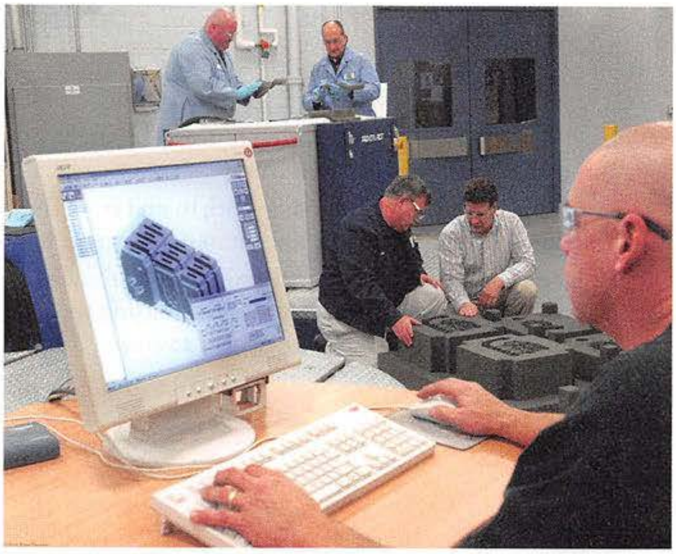


Stratasys, Inc.

Figure 25-20. Fused deposition modeling equipment can produce prototypes from ABS, medical ABS, or investment casting wax. When made from ABS, the parts can be installed and run to proof the design.

Stratasys, Inc.

Figure 25-21. ABS parts can be made in color.

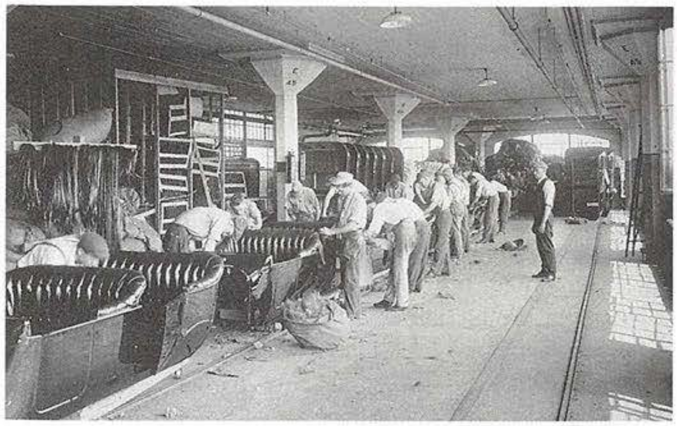


Ford Motor Co.

Figure 25-22. 3D sand printing equipment being used to produce sand-cast molds for production purposes.

the automobile, **Figure 25-23**. However, as these unskilled and semiskilled jobs have vanished, more jobs have become available in other areas. Many of these jobs require a higher level of technical training.

Advances in manufacturing technology displaced some skilled workers in the past, just as automated manufacturing and robots have displaced certain workers in the modern economy. However, machines eventually helped to employ, directly or indirectly, many more people than the number they originally displaced. Better jobs, with higher pay and improved working conditions, were created. These changes also demanded that workers and technicians become better educated. The key to the future, then, will be men and women who are well versed in the various areas of industrial technology.



A

B

Ford Motor Co.

Figure 25-23. Then and now in the manufacture of automobiles. A—When Henry Ford opened the first production line, he paid the workers $5 per day, a very good salary for the time. B—On a modern production line, robots perform many of the jobs. As a result, both productivity and quality have increased dramatically, while production costs have decreased.

Chapter Review

Summary

- Automation is a system for the continuous, automatic production of a product.
- Flexible manufacturing systems are adaptable, allowing a single production line to produce different products simultaneously.
- A robot is a programmable manipulator, capable of moving parts or tools to any location within its work envelope with a high degree of precision and repeatability.
- Safety concerns in an automated manufacturing environment include all the safety concerns encountered in traditional machining, plus the safety concerns directly related to the automated equipment.
- Rapid prototyping techniques can quickly and inexpensively produce accurate models that can be used to improve design and identify potential machining problems.
- Common rapid prototyping techniques include laminated object manufacturing, stereolithography, and fused deposition modeling.

Review Questions

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

1. What is automation?
2. How are automated machines actuated?
3. List the five basic manufacturing processes that automated machines can perform.
4. List three means of transferring work to and from the individual machines in a flexible manufacturing cell.
5. What do the following automation-related acronyms stand for? (An acronym is a word formed using the initial letters of words in a phrase. For example, *RPM* stands for "revolutions per minute.")
   A. FMS.
   B. CIM.
   C. FMC.
   D. CNC.
   E. JIT.
6. Explain the following terms:
   A. *Smart tooling.*
   B. *JIT inventory.*
   C. *Robot.*
7. In robotics, what is meant by the term *work envelope*?
8. List three general uses for robots.
9. How do the safety precautions for automated manufacturing compare with those for using traditional manual machine tools?
10. List three techniques that are available for the rapid prototyping of a CAD design.
11. In stereolithography, a(n) _____ is used to harden a thin layer of material as the object is built up.

CHAPTER **26**

Quality Control

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Summarize the history of quality control.
- Explain the difference between destructive and nondestructive testing.
- Describe the various nondestructive testing methods.
- Explain the advantage of using computers in quality control methods.
- Describe the importance of calibration and traceability in quality control.

Technical Terms

accuracy
calibration
coordinate measuring machine (CMM)
destructive testing
dye penetrant inspection
eddy-current inspection
magnetic particle inspection
megahertz (MHz)
nondestructive testing
optical comparator
precision
quality control (QC)
radiographic inspection
statistical process control (SPC)
traceability
ultrasonic testing

Quality control (QC) is a process that identifies and prevents potential product defects in the manufacturing process before they can cause injuries or damage or result in substandard products. The ultimate goal of quality control is not to detect imperfect products after they are manufactured, but rather to prevent them from ever being made. Quality control plays a vital role in improving the competitive position of a manufacturer.

26.1 The History of Quality Control

The development of quality control closely parallels the development of the airplane. Early aircraft were made with little concern for quality control, **Figure 26-1**. A full-size drawing was usually drawn in chalk on the shop floor. The plane was built over the chalk drawing. If a wood strip was straight with no knots and looked strong enough, it was used. The design was adjusted as needed to make parts fit. As a result, there were a lot of variations from plane to plane.

As the theory of flight became more refined, more care was taken in selecting materials that went into the planes, **Figure 26-2**. Tests were made to determine the strength of materials. Engines were made to specific standards and were checked many times during their manufacture. As a result, aircraft dependability increased greatly. In the automotive field, Henry Ford began the mass production of the Model-T using similar quality control methods.

The large-scale production of all-metal aircraft in the early 1930s introduced many new quality control techniques, **Figure 26-3**. Jigs and fixtures, which held the parts while they were machined or fabricated, were aligned with optical tools. The use of iron filings and a magnetic current, called *magnetic particle inspection*, helped to find defects and flaws in ferrous metals. Tolerances measured in ten-thousandths of an inch became common for engine components. Inspectors made up a larger percentage of the workforce than ever before. Because of the success of quality control programs in aeronautics, many other industries established quality control programs as well.

Goodheart-Willcox Publisher

Figure 26-1. There was very little quality control in the manufacture of early aircraft, like this one on display at an aeronautical museum.

Bill Hannan

Figure 26-2. This Fokker DRI triplane, used by the Germans in the First World War, was constructed to specifications. Some of the materials were inspected and tested before use. However, some of its components had tolerances as large as ±1″ (±25 mm).

John Winter

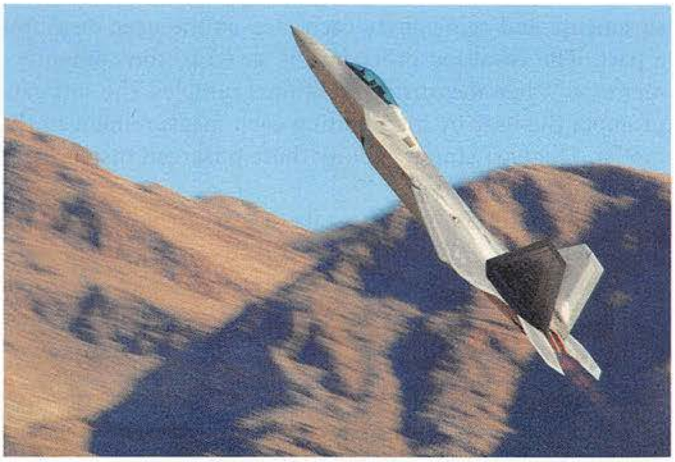
Figure 26-3. The manufacturing of all-metal aircraft during the 1930s brought about the development of quality control techniques that are still in use. This PBY Catalina amphibious aircraft was built in 1935 and is still in flying condition.

The need for thousands of high-performance aircraft during World War II, **Figure 26-4**, led to the introduction of many of the quality control techniques in use today. The inspector became a vital part of the manufacturing team.

Advanced vehicles, such as supersonic jets and spacecraft, **Figure 26-5**, require quality control programs of great scope and magnitude. Because these vehicles are subjected to extreme pressures, stresses, and temperatures, a

Bob Walker

Figure 26-4. The B-17 Flying Fortress was the backbone of the Air Force during the 1940s. Mass production was possible only because of modern quality control procedures.



Christopher Parypa/Shutterstock.com

Figure 26-5. F-22 Raptor in a steep climb. Even the smallest defect in such a high-performance plane could be catastrophic. A rigorous quality control program must be followed during its construction and maintenance.

breakdown or malfunction of any of the thousands of critical parts would be disastrous.

Following the lead of the aerospace industry, other industries began placing increasing emphasis on quality control techniques. Today, as products become more complex and demand for reliability continues to grow, an increasing percentage of industry's budget is being spent on quality control.

26.2 Types of Quality Control

Quality control techniques fall into two basic classifications:

- ***Destructive testing*** destroys or breaks the part during the quality control testing program.
- ***Nondestructive testing*** is done in such a manner that the usefulness of the product is not impaired.

26.2.1 Destructive Testing

Destructive tests range from the simple "break-it-and-look" test to costly and time-consuming quality control procedures involving various techniques, chemicals, and equipment, **Figure 26-6**. A specimen to use for testing is selected either at random or at intervals from a predetermined number of pieces. At a statistical level, destructive testing indicates the characteristics of the untested (and undestroyed) remaining pieces. However, it cannot ensure that 100% of the parts will be quality-compliant. In fact, any number of the untested parts could be defective and unwittingly be used in the manufacture of a product.

26.2.2 Nondestructive Testing

Nondestructive testing is a basic tool of industry. It is well-suited for testing electronic and aerospace products in which the performance of each part is critical. Each piece can be tested individually and as part of a completed assembly.

26.3 Nondestructive Testing Techniques

You are familiar with several methods of nondestructive testing: measuring, weighing, and visual inspection. While satisfactory for many products, these methods leave much to be desired for other products. To ensure effective quality control in all areas of manufacturing, industry has developed more sophisticated testing techniques.

Ford Motor Company

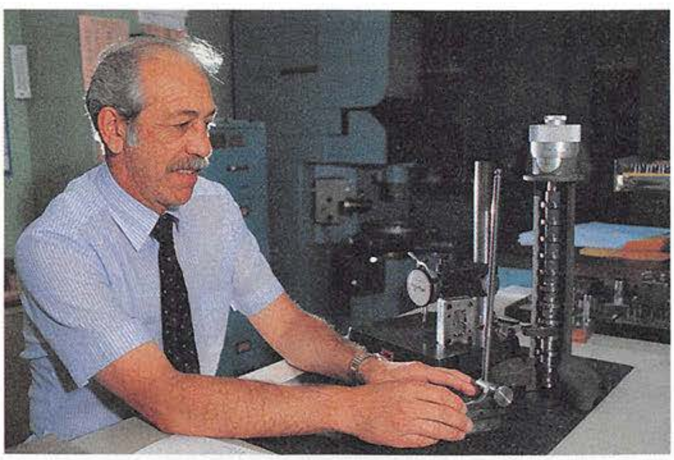
Figure 26-6. In this destructive test, a known weight is thrown against the vehicle door at a known velocity. The test is used to check the sensitivity of the side-impact airbag sensors.

26.3.1 Measuring

The use of micrometers, vernier tools, dial indicators, gages, and similar devices falls into the dimensional measuring category of quality control testing. See **Figure 26-7**. To ensure accuracy, these tools must be used properly and frequently calibrated using known standards, such as those established by the National Institute of Standards and Technology (NIST). The calibration is done in a precision tool calibration laboratory, **Figure 26-8**.

Aumm graphixphoto/Shutterstock.com

Figure 26-7. The use of basic measuring tools is still an important aspect of quality control systems. Many feature digital displays, eliminating human error that may occur when visually reading measurements.



Master Lock Co.

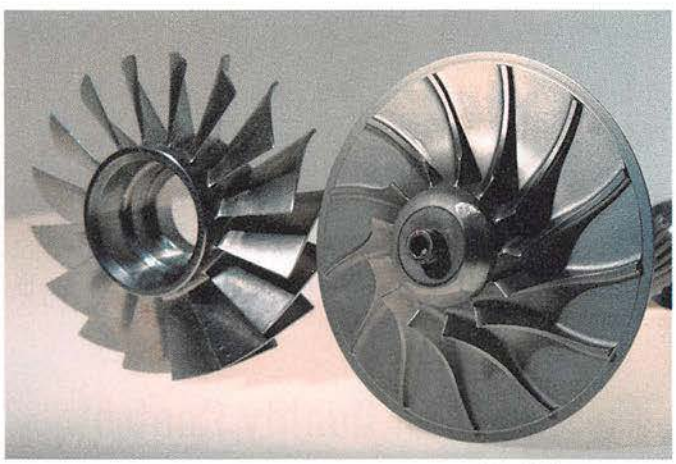
Figure 26-8. Personnel in the precision tool calibration laboratory test and keep an accurate set of records on all production measuring tools to ensure their accuracy. This technician is using a dial indicator to check the accuracy of a plate lamination gage used by automatic press operators at a production site.

The shape of some products, **Figure 26-9**, prevents accurate measurements from being made by conventional measuring tools. A ***coordinate measuring machine (CMM)***, **Figure 26-10**, is an instrument that makes precise measurements electronically. A CMM can be used manually or programmed to check any number of individual reference points on an object against specifications.

Measuring can be done visually on an ***optical comparator***, which is a gaging system for inspection and precise measurement of small parts and small sections of larger parts. See **Figure 26-11**. A part can be magnified up to 500× without distortion to permit accurate measurement. An enlarged image of the part being inspected is projected onto a screen, where it is superimposed on a grid or an accurate drawing overlay of the part. At maximum magnification, variations as small as 0.00001″ (0.00025 mm) can be noted by a skilled operator, **Figure 26-12**.

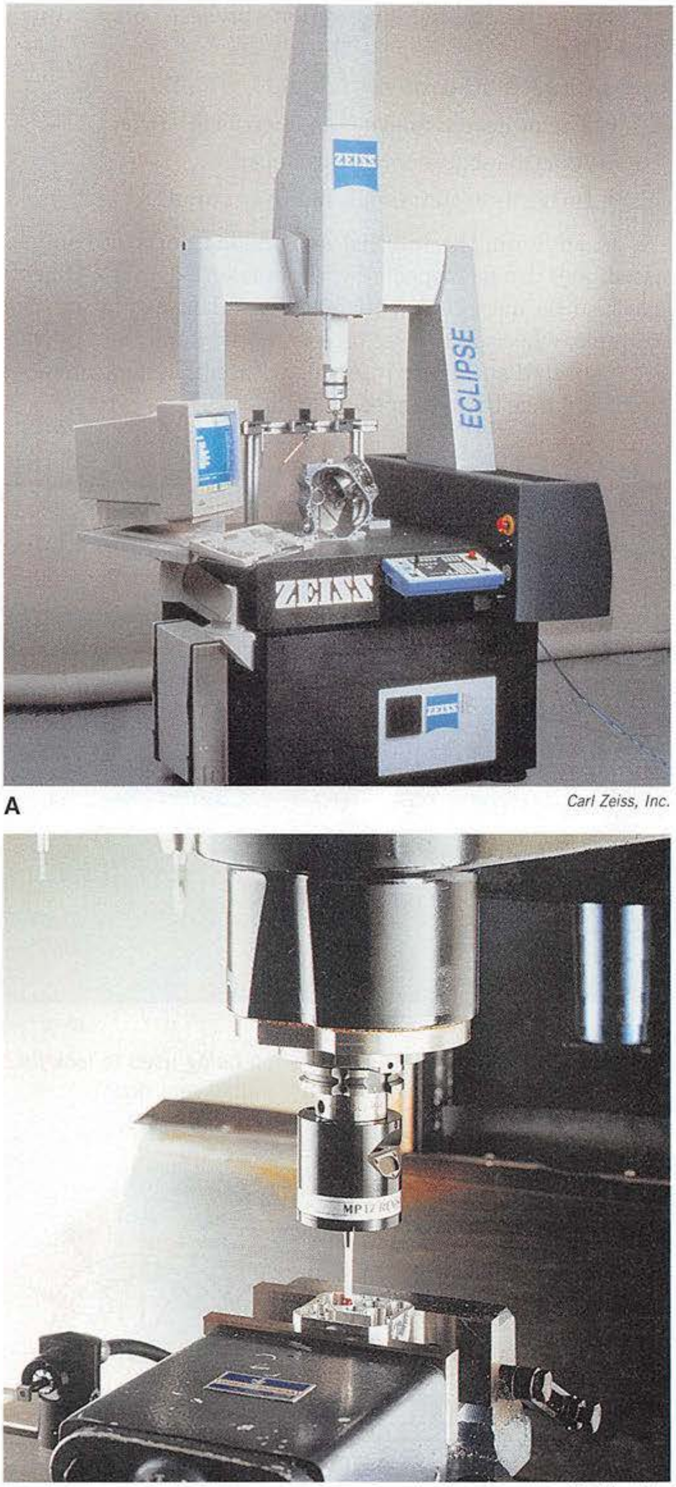
An *optical gaging system* automates the process of optical comparator inspection by incorporating a computer. The computer's software builds a graphical model of the measurements and remembers each step as the user measures a part. The resulting procedure is saved to the computer's memory. When measuring additional samples, the software prompts the user by highlighting each measurement in the model. The user simply follows these onscreen instructions to finish the inspection.

Manufacturers can also take a statistical approach to quality control. This is carefully worked out in a mathematical approach to quality measurement known as ***statistical process control (SPC)***. In statistical process control, a certain percentage of parts made in a production run are measured. If a wide variance is found among the inspected parts, adjustments are made and the inspection rate is increased to include more or all of the units manufactured. Through statistical analysis of the variations found in the manufacturing process, the manufacturer can predict factors such as tool



Mark Yuill/Shutterstock.com

Figure 26-9. Complex parts, such as these turbine wheels, cannot be checked for accuracy using conventional measuring tools. Why do you think those measuring tools cannot be used?



A Carl Zeiss, Inc.

B Renishaw, Inc.

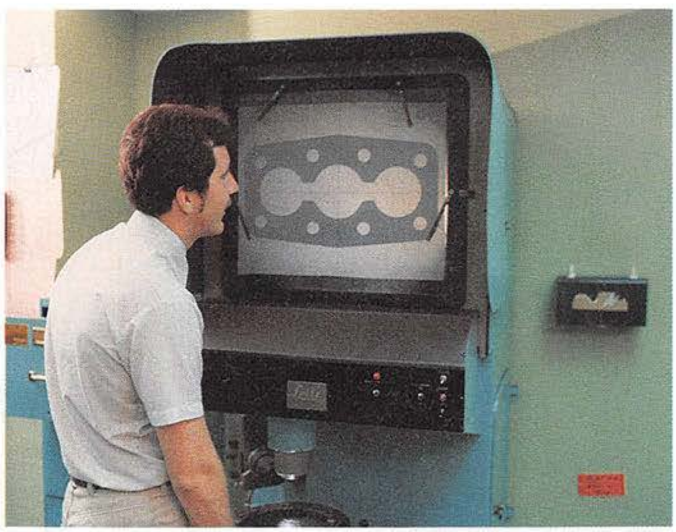
Figure 26-10. Coordinate measuring machines. A—This CNC 3-axis coordinate measuring machine, with a weight capacity of 1000 lb, can measure parts with an accuracy of 0.0002″ (0.005 mm). Because of ceramic material used in parts of its construction, accuracy is guaranteed between 64°F and 78°F. The probe system can scan the work at a velocity of 17″ (425 mm) per second. B—This probe is being used to verify the accuracy of the holes machined into a complex part. A CMM is more accurate than any manual method for measuring complex contours and surfaces.

and equipment wear and correct the problems before they result in unacceptable products.

Special gaging and inspection tools can be designed for almost any application, **Figure 26-13**. Combining precision tools with electronic devices permits accurate inspection to be made by semiskilled workers.

Photo courtesy of Quality Vision International, Inc. (QVI®) © 2016

Figure 26-11. Optical comparator inspection is performed automatically on some machines.



Master Lock Co.

Figure 26-12. A quality control engineer is checking hole location on a plate lamination of an automatic lock body assembly, using an optical comparator with a part overlay transparency.

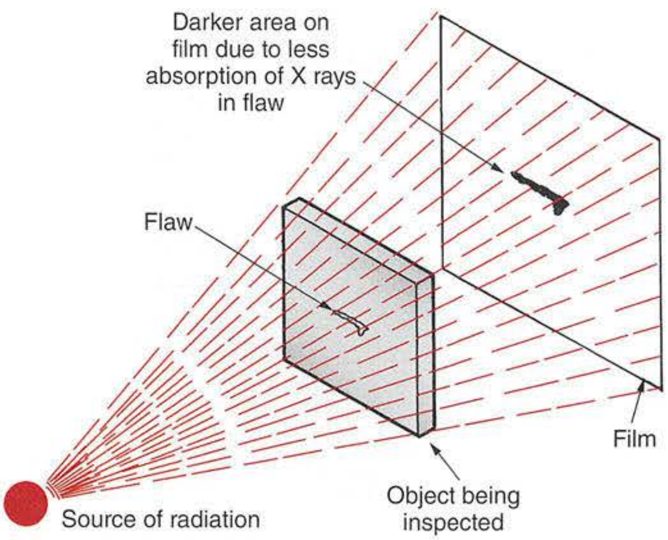
Aumm graphixphoto/Shutterstock.com

Figure 26-13. High accuracy optical microscope inspects a tool.

26.3.2 Radiographic (X-ray) Inspection

Radiographic inspection uses X rays and gamma radiation to detect flaws, such as cracks or voids in an object. This inspection method uses radiographic inspection equipment to project X rays and gamma radiation (highly energetic, penetrating radiation found in certain radioactive elements) through the object under inspection and onto a section of photographic film, **Figure 26-14**. The developed film shows an image of the internal structure of the part or assembly, **Figure 26-15**. Radiographic inspection has become routine in the inspection of critical parts and materials.

Many kinds of *peripheral* (outside circumference) inspections can be made because of the *omnidirectional* (all directions) characteristics of the X rays, **Figure 26-16**.



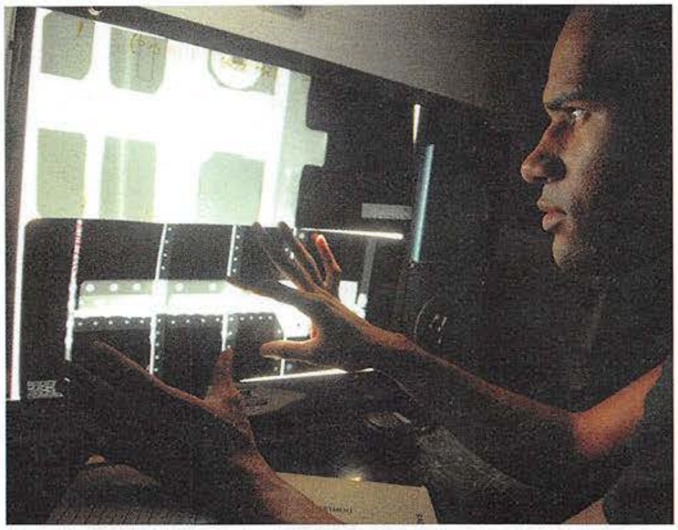
Goodheart-Willcox Publisher

Figure 26-14. The radiographic inspection process.

The radiographic inspection process offers many advantages:

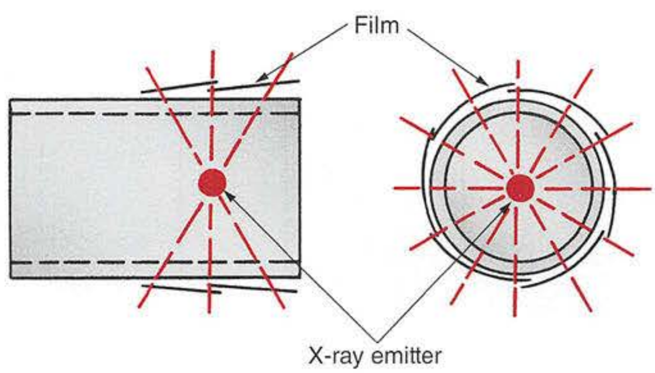
- Inspection sensitivity is high.
- The projected image is geometrically correct.
- A permanent record is produced.
- Image interpretation is highly accurate.

In addition, the internal and hidden parts of complex assemblies can be inspected without fear of damage. Objects can also be inspected without being taken out of use. For example, wheels and axles on locomotives and railroad cars are inspected at regular intervals without having to remove them and take them to special facilities.



US Air Force

Figure 26-15. Radiographic examination being used to look for cracks and other flaws in an aircraft landing gear door.



Goodheart-Willcox Publisher

Figure 26-16. Technique used to perform radiographic inspection on cylindrical objects. The X-ray emitter is placed in the center of the object and X-ray film is placed all around the circumference of the object. A flaw causes more exposure of the film, so an image of the flaw is shown on the film when it is developed.

> **SAFETY NOTE**
>
> If you perform radiographic inspections, you must observe strict safety precautions when handling radioactive materials. You must also avoid exposure to X-ray radiation during testing. Exposure to X rays beyond established limits can be harmful.

26.3.3 Magnetic Particle Inspection

Magnetic particle inspection, commonly known as *magnafluxing* (a trademark of Magnaflux Corp.) is a nondestructive testing technique that uses iron particles and induced magnetic fields to detect flaws on or near the surface of ferromagnetic (iron-based) materials. The technique is rapid, but it shows only serious defects. It does not show scratches or minor visual defects.

The magnetic particle inspection technique is based on the theory that every conductor of electricity is surrounded by a circular magnetic field. If the part is made of ferromagnetic material, these lines of force are, to a large extent, contained within the piece. A circular magnetic field, if not interrupted, has no poles. However, because a flaw or other imperfection cuts through these magnetic lines of force, poles form at each edge of the flaw. These poles hold the finely divided iron particles, thus outlining the flaw. However, if the flaw is parallel to the lines of magnetic force, the flaw will not interrupt the force, so no indication of it will appear when iron particles are applied, **Figure 26-17**.

In practice, a magnetic field is induced into the part and fine particles of iron are blown (dry method) or poured in liquid suspension (wet method) over it, **Figure 26-18**. Flaws

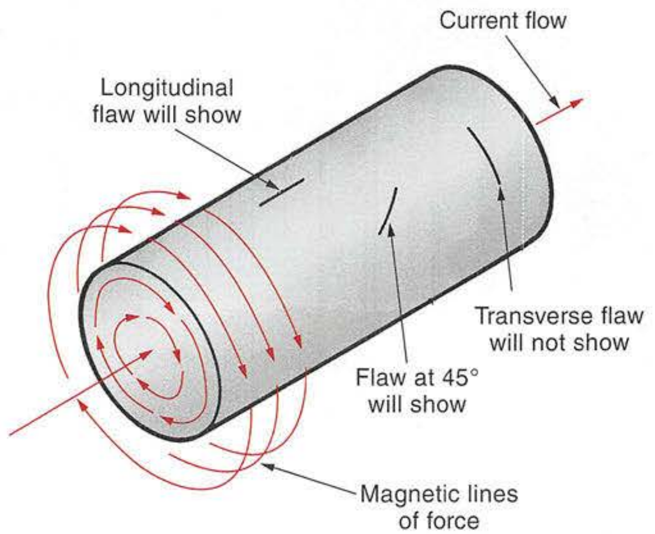


Figure 26-17. Theory, scope, and limitations of the magnetic particle inspection technique.

CAREER CONNECTION

Quality Control Inspector

What does a quality control inspector do?

Quality control inspectors test finished products for defects or errors from the specifications set for the product. There are two types of inspectors:

- Materials inspectors check products for flaws, such as missing pieces, scratches, and seams that are off specifications.
- Mechanical inspectors confirm that a product's parts fit together and operate correctly. They may also conduct test runs and other trials to ensure proper operation.

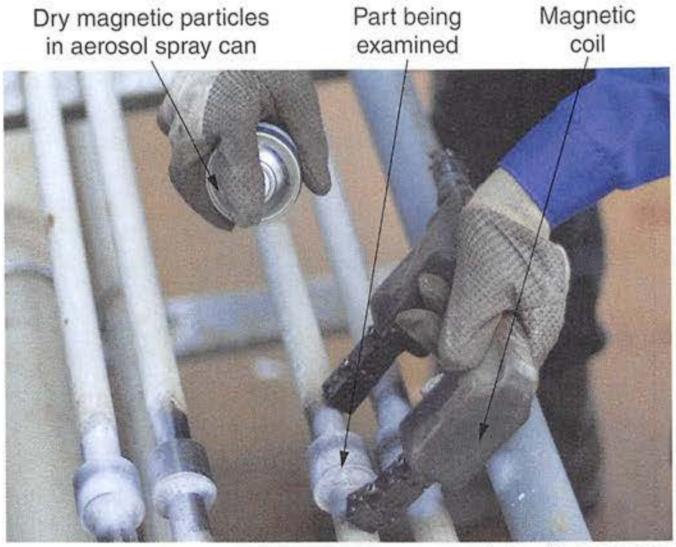
What education and skills are needed to be a quality control inspector?

A high school diploma is required for most positions, but much of this skill set is learned through on-the-job training. The length of training varies by employer and can last from a month to a year. Interested students should choose coursework in industrial trades and sciences and take advantage of any opportunities to work in laboratory settings.

What is it like to be a quality control inspector?

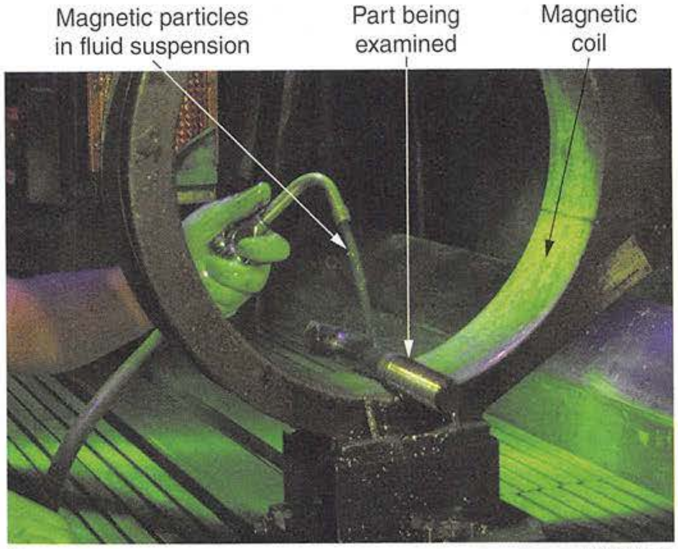
The environment a quality control inspector works in is decided by the needs of the product being tested. Inspectors in manufacturing can work in environments carefully controlled for testing or directly in manufacturing areas, where exposure to loud noise and other hazards is possible. Medical and pharmaceutical inspectors typically work in laboratories.

The *Occupational Outlook Handbook* reports that quality control inspectors average $36,780 per year. Wages vary by industry, and the highest paid workers can make more than $62,400 annually.



A

karlstury/Shutterstock.com



B

US Air Force

Figure 26-18. Variations of magnetic particle inspection. A—Dry method. In this test, fine iron particles are sprayed over the test area using an aerosol spray can. A handheld magnetic coil is placed on the test site. B—In the wet method, iron particles suspended in a fluid are poured over the part being inspected. The iron particles have been colored with fluorescent dye to improve their visibility under a black light.

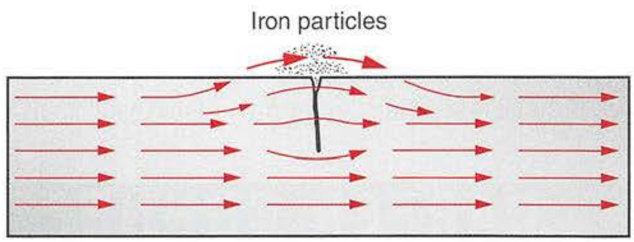
disturb or distort the magnetic field, giving it magnetic properties different from those of the surrounding metal. Many of the iron particles are attracted to the area and show the exact location, shape, and extent of the flaw, **Figure 26-19.**

26.3.4 Dye Penetrant Inspection

Because magnetic particle inspection is useful only on steel and iron, a different inspection method is required for nonmagnetic materials. ***Dye penetrant inspection*** relies on capillary action to show flaws in parts and can therefore be used on nonmagnetic metals. In dye penetrant inspection, a fluorescent dye solution is applied to the part's surface by dipping, spraying, or brushing, **Figure 26-20.** Capillary action draws the dye solution into the defect. The surface is rinsed clean, and a developer is applied. The developer acts as a blotter, drawing the dye penetrant back to the surface.

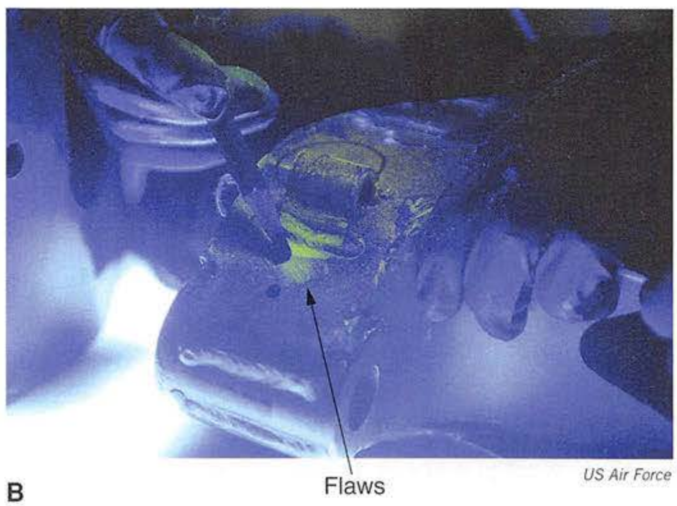
When the part is inspected under ultraviolet light (black light), any defects will glow with fluorescent brilliance, making them easily visible, **Figure 26-21.** This is because dye penetrant has flowed into and remains in the flaw.

Visible dye penetrant inspections use a dye that makes flaws visible under normal light. This type of dye penetrant test is easy to use, accurate, economical, and does not require a black light. Application is similar to that described for the fluorescent dye penetrant. The specimen is coated with a red liquid dye that soaks into the surface crack or flaw. The liquid is then washed off and the part is dried. A developer is dusted or sprayed on the part, and any flaws or cracks show up red against the white background of the developer.



A

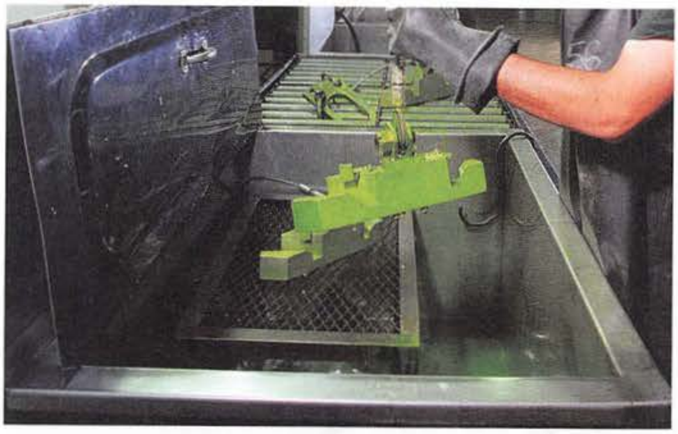
Goodheart-Willcox Publisher



B

US Air Force

Figure 26-19. Magnetic particles reveal cracks in a part or assembly. A—Cracks in the part generate a magnetic field on the surface of the part, which holds the iron particles in place. B—A buildup of iron particles makes even tiny flaws visible.



US Air Force

Figure 26-20. Parts being dipped in fluorescent dye penetrant. The excess dye will be rinsed off and a developer will be applied to the parts before they are inspected.

US Air Force

Figure 26-21. A part treated with dye penetrant is being examined under ultraviolet light. The areas of the part that are glowing brightly are potential flaws.

26.3.5 Ultrasonic Inspection

Ultrasonic testing techniques make use of sound waves above the audible range. They can be used to detect cracks and flaws in almost any kind of material that is capable of conducting sound. Sound waves can also be used to measure the thickness of the same materials from one side. Ultrasonic test equipment is shown in **Figure 26-22**.

The human ear can hear sound waves whose frequencies range from about 20 to 20,000 hertz (cycles per second, abbreviated Hz). Sound waves that oscillate (vibrate) with a frequency greater than 20,000 Hz are inaudible (cannot be heard) and are known as ultrasound. Ultrasonic testing equipment produces sound waves whose frequency is measured in millions of cycles per second, or ***megahertz (MHz)***.

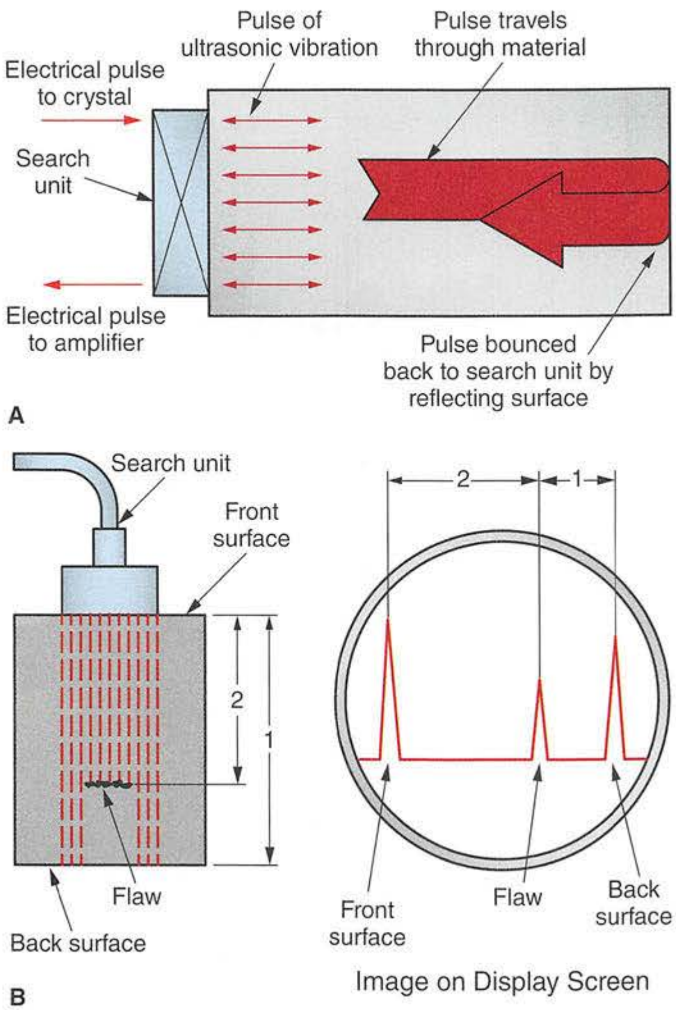
Sound waves are used to obtain information about the internal structure of a material. By observing the echoes that are reflected from within the material, it is possible to judge distances by the length of time required to receive an echo from an obstruction (flaw). See **Figure 26-23**. The ultrasonic testing equipment includes a timing device for measuring the relative length of elapsed time between the sending of the sound waves and the return of the various echoes.

US Navy

Figure 26-22. An ultrasonic inspection is being performed with portable, contact-type ultrasonic testing equipment.

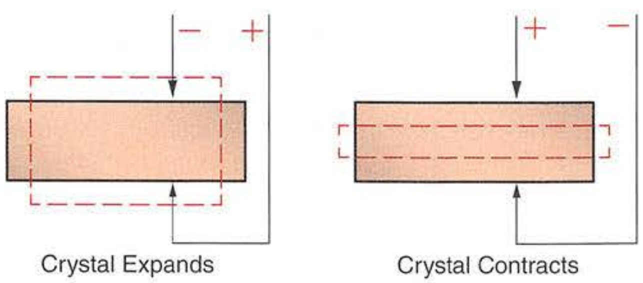
The search unit contains a piezoelectric transducer (crystal) that is electrically pulsed, which causes it to vibrate at its own natural frequency to produce the ultrasonic waves, **Figure 26-24**. A liquid coupling transmits the sound waves from the search unit to the metal and returns the echoes to the search unit. The liquid coupling is usually a film of oil, glycerin, or water between the transducer and the test piece. The same results can be achieved by immersing both the test piece and the search unit in water, **Figure 26-25**. Immersion testing is ideal for production testing, since there is no contact between the search unit and the work. The lack of contact prevents the transducer from wearing out.

The search unit's transducer vibrates for about two-millionths of a second. The result is a very short burst of sound waves that travel through the liquid to the surface of the test material. A portion of the sound is immediately reflected from the surface of the metal as a very large echo. Part of the sound will not be reflected and will continue into the test material. If this portion of the sound encounters no interference, such as a discontinuity (flaw) in the material, it continues until it is partially reflected from the back surface as a second echo or "back reflection." If there is a flaw in the interior, a portion of the sound will be reflected from the flaw and will return to the transducer as a separate echo between the echoes received from the front and back surfaces.



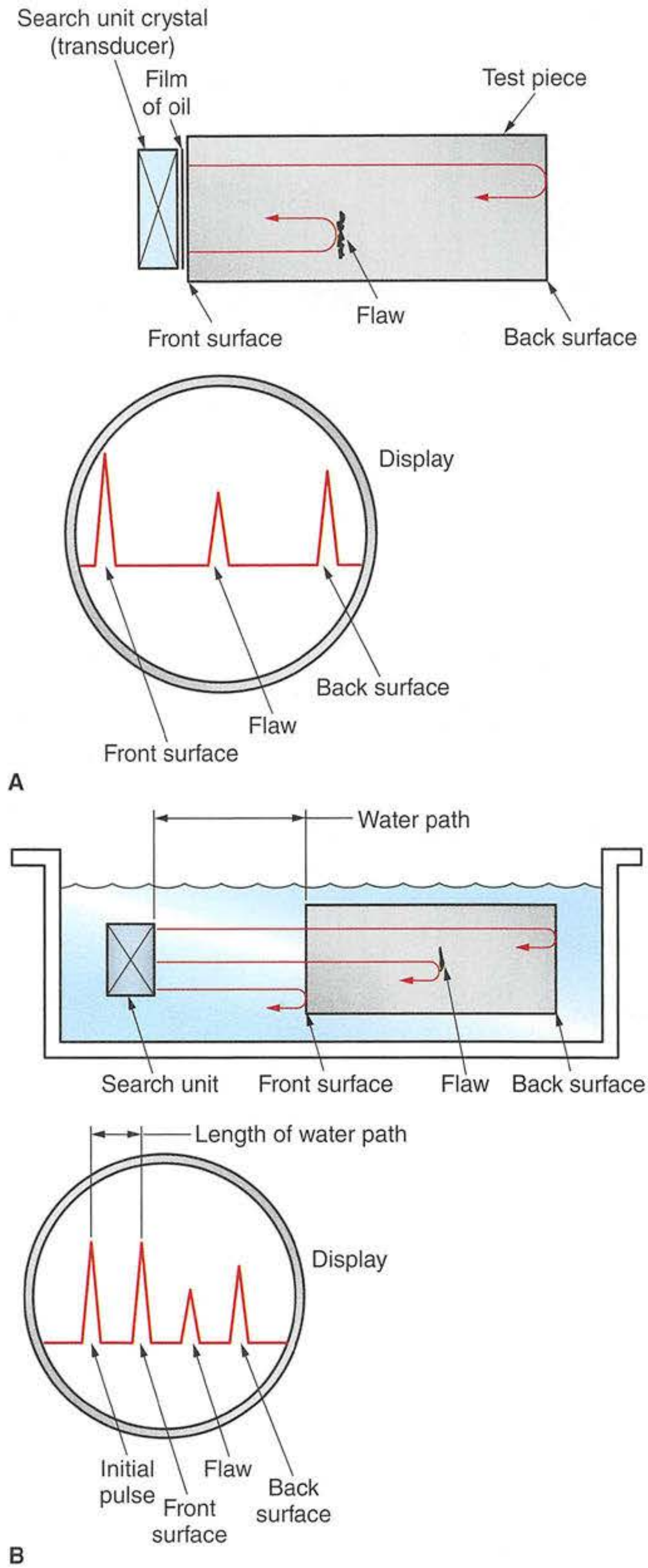
Goodheart-Willcox Publisher

Figure 26-23. Ultrasonic testing. A—Sound waves are produced by the search unit and pass through the material being inspected. When they strike the back surface of the material, they are reflected back to the search unit. B—When the search unit passes over a flaw, the length of time it takes the ultrasonic sound waves to be reflected back to the search unit is reduced.



Goodheart-Willcox Publisher

Figure 26-24. A search unit's piezoelectric transducer is a device containing a crystal that changes shape based on electric polarity. As the X-cut crystal is exposed to alternating polarity, the crystal's shape fluctuates, producing ultrasonic sound waves.



Goodheart-Willcox Publisher

Figure 26-25. Liquid coupling. A—Action of a contact ultrasonic inspection device. A film of oil, water, or glycerin is used to make a positive contact between the transducer and the test piece. B—Immersion ultrasonic testing. Note the extra spike on the display, indicating the path through water.

After the transducer has given off its short burst of sound waves, it stops vibrating and "listens" for the returning echoes. When the echoes are received, they cause the transducer to vibrate, generating an electric current that can be displayed as a graph, **Figure 26-26**. The graph displayed onscreen can be expanded or condensed to improve its readability.

There are two basic categories of ultrasonic testing, as shown in **Figure 26-27**:

- **Pulse echo.** Uses sound waves that travel through the test piece until they are reflected by flaws or the back of the test piece. The relative positions of the echoes on a graph identify the location of flaws. The same search unit is used both to transmit the sound and to receive the echoes.
- **Through inspection.** Uses one search unit to transmit ultrasonic waves through the piece and another search unit to pick up the signal on the opposite side of the piece. If the beam is partially blocked by a flaw, the reduced intensity of the signal activates a warning device (such as a flashing light or ringing bell) to alert the operator to the flaw. The through inspection technique is frequently used to check the integrity of helicopter rotor blades, for example.

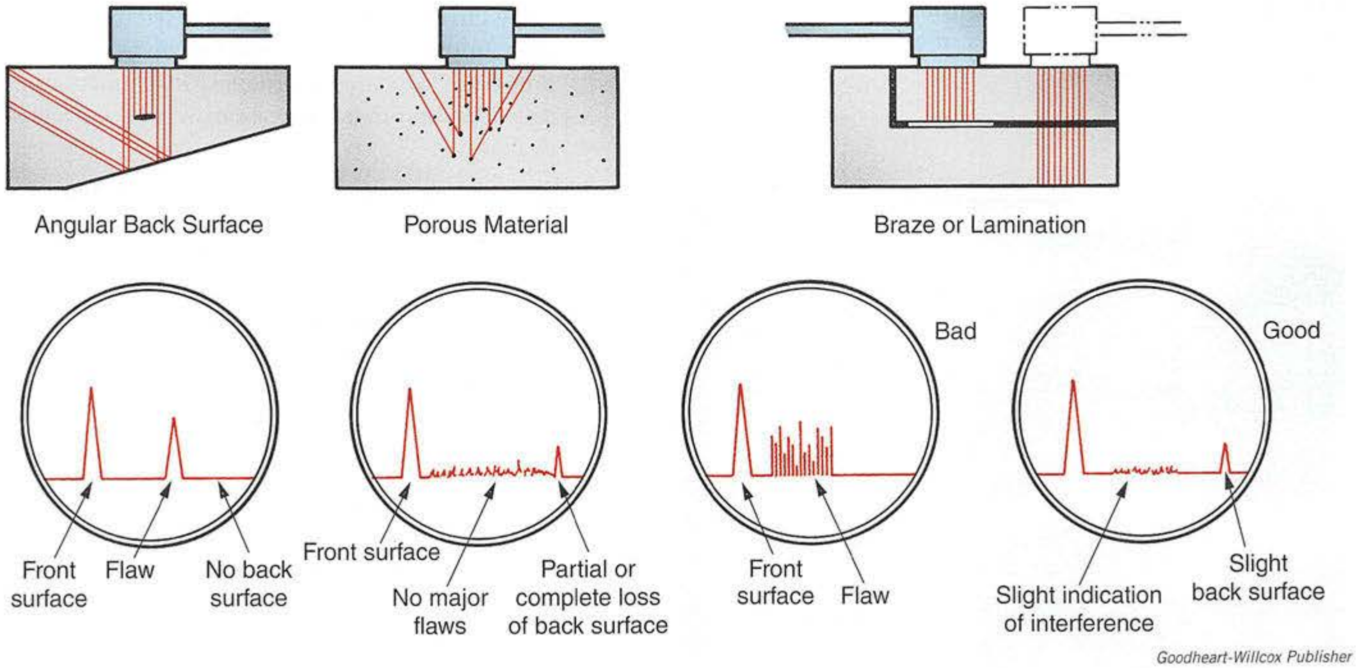


Figure 26-26. Various flaws as they appear on a video display.

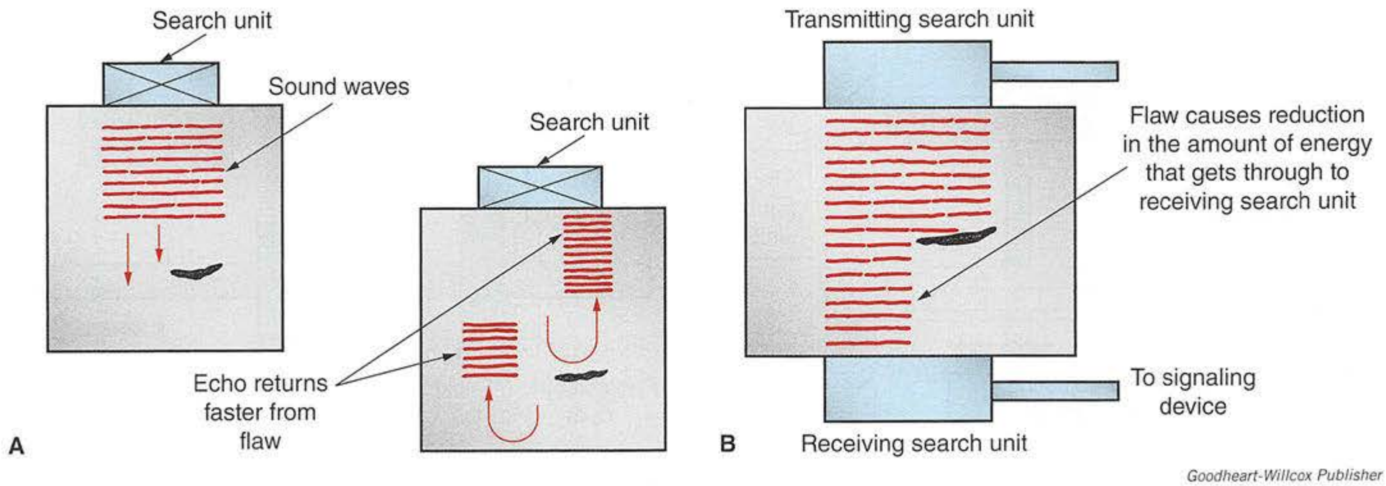
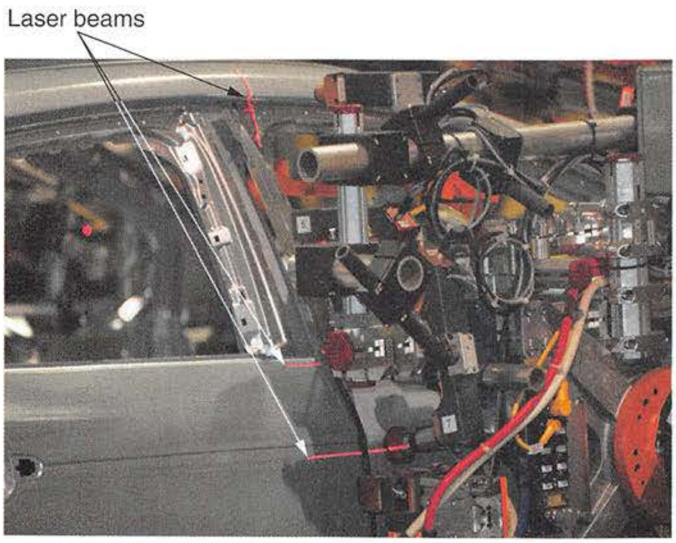


Figure 26-27. Types of ultrasonic inspection. A—With pulse echo ultrasonic inspection, the reflected waves return sooner when they bounce off a flaw. B—Through inspection uses one search unit to generate the ultrasonic waves and a separate search unit to detect the ultrasonic waves when they reach the opposite side of the piece.

26.3.6 Laser Inspection

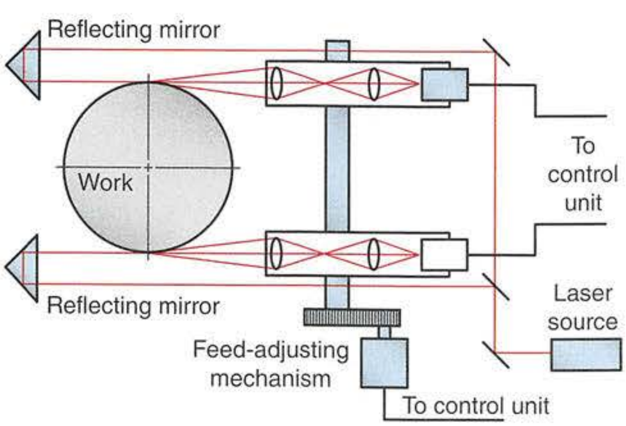
Lasers have been adapted for quality control, **Figure 26-28.** In addition to being able to check the fit of components on an assembly line, a computer-controlled laser can be made sensitive enough to detect tool wear and automatically compensate for it, **Figure 26-29.** This helps prevent imperfect parts from being made. Laser measuring devices can also be used to evaluate the condition of products that are in use.

Some laser inspection devices include the ability to scan products in grid patterns, much like optical and CMM units, and covert the scanned data into 3D models that can be compared to standard models using computer systems. The key advantage of precision laser scanning systems is the speed with which the scans can be performed. They can provide data for quality control decisions in a fraction of the time required by other methods.



Ford Motor Company

Figure 26-28. Robotic laser inspection being used to check the fit of a door as it is installed on a vehicle.



Goodheart-Willcox Publisher

Figure 26-29. One type of laser inspection device that continuously monitors tool wear. The laser beams can detect work that is oversize or undersize.

26.3.7 Eddy-Current Inspection

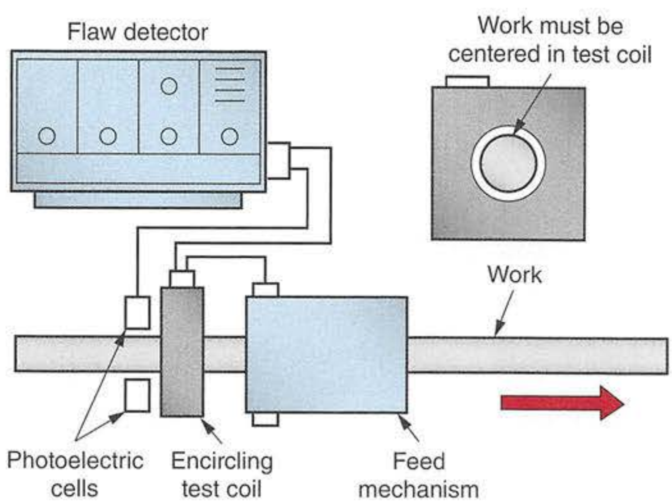
Eddy-current inspection is based on the fact that flaws in a metal product cause impedance (resistance) changes in a coil brought near the part. A metal part with flaws produces a different eddy current than one without flaws when a test coil is placed next to it.

Eddy-current inspection methods can be divided into two general categories:

- **Eddy-current differential system.** Used to detect cracks, seams, holes, or other flaws in metal parts (such as wire, tubing, and bar stock) as they move off the production line. The test equipment must be sensitive to rapid change.
- **Eddy-current absolute system.** Used to detect variations in dimension, composition, and other physical properties of a metal product. Instrumentation for this operation must be responsive to relatively small changes.

Eddy-current inspection equipment is designed to detect changes in impedance and convert them into a form that can be monitored by the operator. See **Figure 26-30.** The material being tested can be passed through encircling detection coils at speeds up to 500 fpm (152 mpm). When a flaw in the test piece is detected, the eddy-current test equipment does one or more of the following:

- Flashes a warning light to alert the operator.
- Sounds a tone alarm.
- Activates a rejection device to eject the part that does not meet standards.
- Marks the section that contains the flaw so it can be removed.
- Provides an electronic record of the test.



Goodheart-Willcox Publisher

Figure 26-30. Eddy-current inspection system. Photoelectric cells turn off the alarm system when the end of a test piece passes into the test coil. Any flaw would cause small current changes in the test coil.

26.4 Other Quality Control Techniques

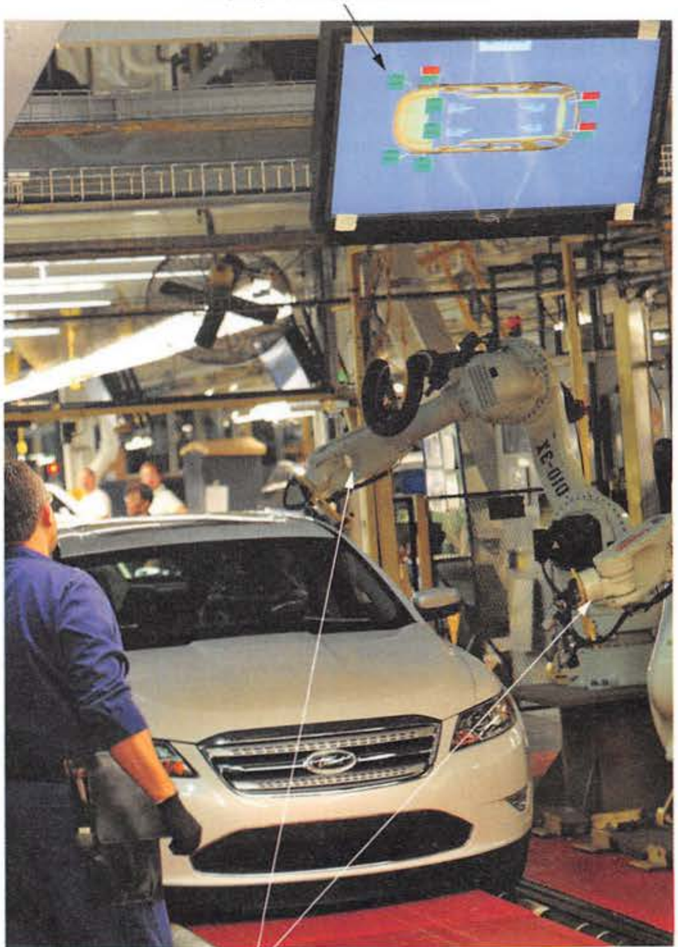
In addition to the quality control techniques described in this chapter, industry makes extensive use of other testing devices. For example, the quality of a machined surface may be critical. The surface finish can be inspected visually with a microfinish surface comparator or electronically with a profilometer. See **Figure 26-31**.

Because of the versatility of the computer, many new quality control techniques are being developed as the need arises. The use of computers makes complete inspection of all parts or products much faster, **Figure 26-32**.

26.5 Calibration and Traceability

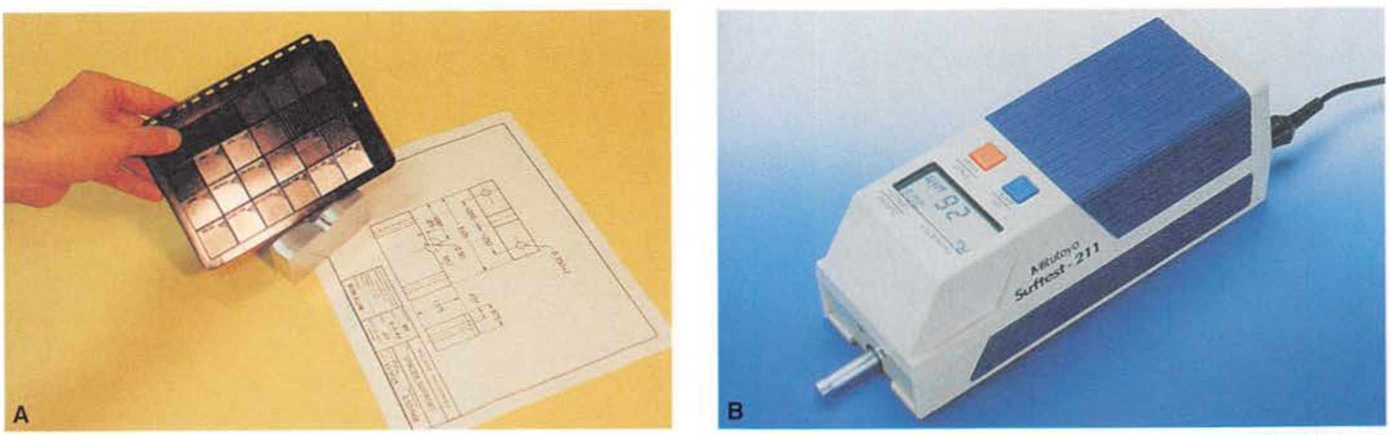
In a manufacturing setting, the ability to measure manufactured parts accurately and precisely is critical. ***Accuracy*** describes how close the value of the measurement is to the true value, and ***precision*** describes how close two or more results are to each other, or the repeatability of the measurement. For example, if a reference part is weighed four times on the same scale, and the readings are 3.212 lb, 3.211 lb, 3.211 lb, and 3.212, the scale is precise, because all of the values are very close to each other. If the actual weight of the part is 3.211 lb, then the scale is accurate as well. However, if the actual weight of the object is 3.415 lb, then the scale is not accurate, even though it is precise.

Accurate measurements are needed to set up machines, monitor manufacturing processes, and separate good parts from bad parts. To operate at the level of accuracy and precision for which they were made, measurement tools must be inspected using approved inspection procedures and equipment that has been inspected by an outside source. Most



Ford Motor Company

Figure 26-32. New quality control devices make 100% inspection possible. Lasers scan the front end, doors, and other assemblies to make sure the vehicle's fit and finish are acceptable.



Mitutoyo/MTI Corp.

Figure 26-31. Methods of checking surface finishes. A—Machinist is using a microfinish surface comparator. B—Surface roughness can also be checked electronically with a profilometer.

manufacturers either hire an outside company to perform these inspections or set up their own metrology laboratory.

If the inspections are performed internally, they are performed using sets of gage blocks that have been inspected by an outside laboratory and are compliant with the standards specified by the National Institute of Standards and Technology (NIST), **Figure 26-33**. Such an inspection is called a ***calibration***. If performed properly, all calibrations can be validated by tracing the equipment and procedures back to the NIST standards. This is known as ***traceability***. Larger measurement systems, such as coordinate measuring machines, laser scanners, and optical gaging systems, require specialty equipment for calibrations. Outside vendors are commonly used for this type of equipment.

alterfalter/Shutterstock.com

Figure 26-33. A set of gage blocks compliant with NIST standards can be used to calibrate measurement devices.

Chapter Review

Summary

- The preferred goal of quality control is not to detect and discard imperfect parts, but to prevent the defective parts from being manufactured in the first place.
- The advancement of quality control techniques has paralleled advances in the aeronautical field; as aircraft began to require better quality control methods, suitable methods were developed.
- Quality control measures can be divided into two categories: destructive testing and nondestructive testing.
- In destructive testing, a sample part is selected from a lot and tested. The testing process destroys the specimen.
- In nondestructive testing, the testing process does not damage the specimen. As a result, every part can be tested during the manufacturing process.
- Nondestructive testing methods include measuring, radiographic (X-ray) inspection, magnetic particle inspection, dye penetrant inspection, ultrasonic inspection, laser inspection, and eddy-current inspection.
- The use of computers in quality control devices makes product inspection faster and more accurate.
- The measuring tools used to inspect parts for quality control must be calibrated using equipment that is compliant with the National Institute of Standards and Technology standards.

Review Questions

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

1. Which of the following best describes the ultimate goal of a quality control program?
   A. Identify imperfect products at the end of the production process.
   B. Prevent imperfect products from being made.
   C. Reduce the cost of manufacturing.
   D. None of the above.
2. Briefly explain why the development of quality control measures has paralleled the development of aircraft.
3. Quality control falls into two basic classifications. Name and explain each.
4. Precision measuring tools, such as micrometers, vernier tools, or dial indicators, must be inspected and _____ frequently to ensure their accuracy.
5. The _____ is an instrument that projects the magnified image of a small part on a screen so it can be measured.
6. Which of the following is an advantage of statistical process control?
   A. It allows parts to be measured more accurately.
   B. It predicts problems so they can be corrected before they result in unsatisfactory products.
   C. It relies entirely on visual inspection and does not require any measuring or testing instruments.
   D. All of the above.
7. Radiographic inspection uses _____ and gamma radiation to detect flaws in an object.
8. List four advantages of the radiographic inspection process.
9. Magnetic particle inspection is commonly known as _____.
10. Only _____ metals can be inspected by the magnetic particle technique.
11. Describe the magnetic particle inspection process.
12. Which of the following explains why dye penetrant inspection is more appropriate than magnetic particle inspection in some situations?
    A. Magnetic particle inspection requires much more training.
    B. Magnetic particle inspection is more expensive than dye penetrant inspection.
    C. Dye penetrant inspection can be used on nonmagnetic materials.
    D. Dye penetrant inspection causes less damage to the material than magnetic particle inspection.
13. Describe the dye penetrant inspection process.
14. Ultrasonic inspection makes use of _____.
    A. accurately made measuring fixtures
    B. high-frequency sound waves
    C. X rays
    D. All of the above.
15. List the two basic categories of ultrasonic testing. Briefly describe each.
16. Eddy-current inspection relies on changes in _____ in a test coil to identify flaws in a part.

CHAPTER 27

Geometric Dimensioning and Tolerancing

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

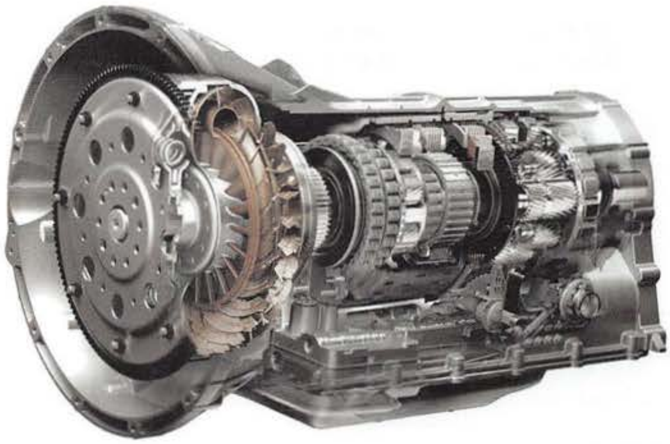
- Explain the basics of geometric dimensioning and tolerancing.
- Explain the differences between conventional and geometric dimensioning and tolerancing.
- Identify each geometric dimensioning and tolerancing symbol.
- Describe how each geometric tolerance is applied.
- Explain the concept of bonus tolerancing.

Technical Terms

actual size
angularity
bonus tolerancing
circular runout
circularity
concentricity
cylindricity
datum
feature
feature control frame
flatness
form tolerances
geometric dimensioning and tolerancing (GD&T)
geometric tolerance
least material condition (LMC)
location tolerances
maximum material condition (MMC)
orientation tolerances
parallelism
perpendicularity
position tolerance
profile of a line tolerance
profile of a surface tolerance
profile tolerance
regardless of feature size (RFS)
runout tolerance
straightness
symmetry
total runout

While conventional tolerancing is appropriate for many products, the tighter tolerances (reduced variation) required in today's manufacturing processes require more stringent controls, especially when the form (shape and size) and position (location) of mating features on parts determine how components fit and function together as an assembled unit. To manufacture components to tighter tolerances and specifications, processes with greater levels of precision are needed. See **Figure 27-1**. More precise processes, along with more stringent controls, provide the foundation for more efficient and economical manufacturing operations through geometric dimensioning and tolerancing.

Geometric dimensioning and tolerancing (GD&T) 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 chapter. Geometric dimensioning and tolerancing is far more involved than described on the following pages. Detailed information can be found in the publication ASME Y14.5M. As you progress in machining technology, you should consider studying a copy of ASME Y14.5M, purchasing a text on the subject, or enrolling in a class on geometric dimensioning and tolerancing.



Ford Motor Co.

Figure 27-1. A high degree of precision is needed to produce the parts used in this transmission. The specified tolerances for the shape and location of features on the parts must not be exceeded.

27.1 Definitions

Geometric characteristic symbols are used to provide clarity and precision in communicating design specifications. See **Figure 27-2**. These symbols are standardized by the American Society of Mechanical Engineers (ASME). ***Geometric tolerance*** is a general term that refers to tolerances that 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 27-3**.

A ***datum*** is an exact point, axis, or plane. It is the origin from which the location or geometric characteristic of a feature of a part is established. It is identified by a solid triangle

WORKPLACE SKILLS

Respecting Diversity

As you have learned in previous chapters, teamwork is an important aspect of most working environments today. Your clients and coworkers will all have different backgrounds and cultures. Diversity includes all these differences among people and can specifically refer to gender, sexuality, religion, race, nationality, and many other distinctions.

Being a good team player involves accepting these differences and working together with your group members. Sometimes conflicts can arise because of different beliefs, but we do not all have to think the same way to be able to reach a goal together. Having team members from different backgrounds can be beneficial to a group. When brainstorming solutions for a problem, having diversity within a group can lead to a wide variety of ideas. Our backgrounds and our cultures affect our ways of thinking, so one person will not approach a problem the same way as someone else of a different culture and background would. Combining different ways of thinking can lead to much more creative solutions.

As you go to work or school each day, you may encounter others who categorize people using biased words and comments. Using age, gender, sexuality, race, disability, or ethnicity as a way to describe others is disrespectful, unethical, and sometimes even illegal. Use bias-free language in all your communications, whether verbal or printed, to show respect for those with whom you come in contact.

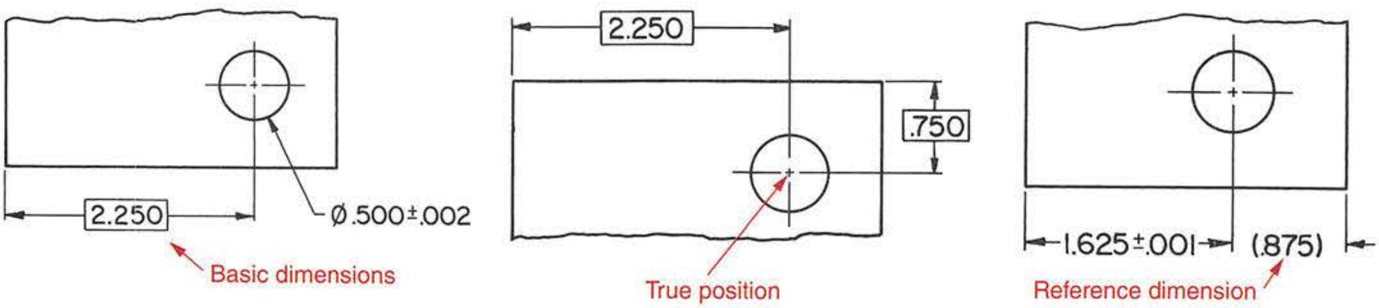
Type of Tolerance	Geometric Characteristic	Symbol
Form	Straightness	—
	Flatness	▱
	Circularity	○
	Cylindricity	⌭
Orientation	Angularity	∠
	Perpendicularity	⊥
	Parallelism	//
Location	Position	⌖
	Concentricity	◎
	Symmetry	⌯
Profile	Profile of a line	⌒
	Profile of a surface	⌓
Runout	Circular runout	↗ or ↗
	Total runout	⌰ or ⌰

A American Society of Mechanical Engineers

Geometric Tolerance Specification	Symbol
Maximum material condition (MMC)	Ⓜ
Least material condition (LMC)	Ⓛ
Regardless of feature size (RFS) (old)	Ⓢ
Diameter	Ø
Radius	R
Basic dimension	[30]
Reference dimension	(30)
Counterbore	⌴
Countersink	⌵
Spotface	SF
Depth/deep	↧
Dimension not to scale	25
Number of times/places	6X
Between	↔

B Goodheart-Willcox Publisher

Figure 27-2. Geometric dimensioning and tolerancing uses many symbols to define tolerances. A—The 14 geometric control characteristics are grouped into five categories: form, orientation, location, profile, and runout tolerances. B—Additional symbols used with the geometric control characteristic symbols to fully define features in GD&T.



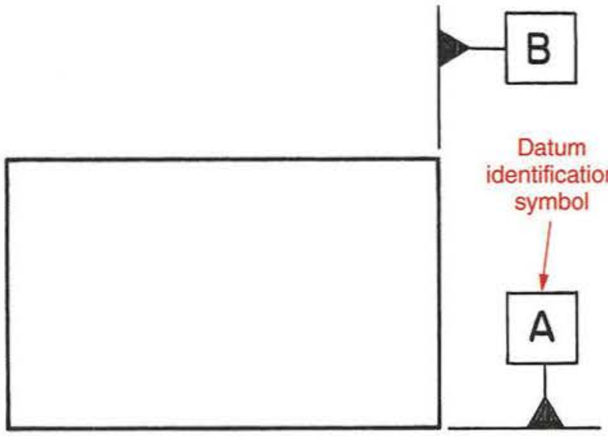
Goodheart-Willcox Publisher

Figure 27-3. Basic dimensions are usually enclosed in a rectangular frame. They are not toleranced. True position is the theoretical exact location of a 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.

with an identifying letter, **Figure 27-4.** ***Feature*** is a general term applied to a physical portion of a part, such as a surface, pin, hole, or slot. A datum feature is the actual feature of a part used to establish a datum. **Figure 27-5.**

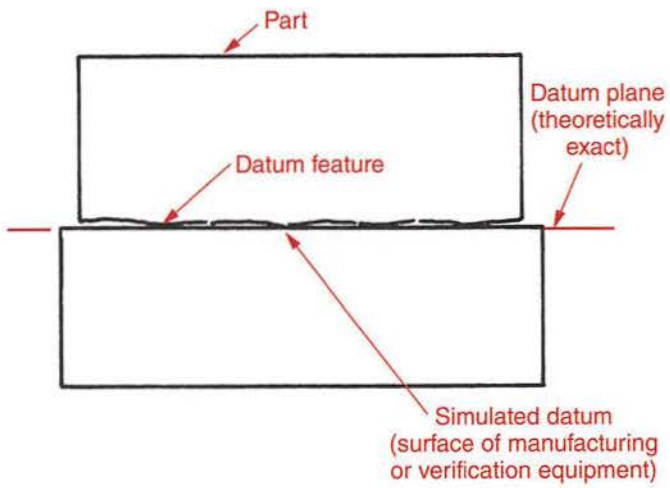
Maximum material condition (MMC) is the condition in which the size of a feature contains the maximum amount of material within the stated limits of size. Examples include a minimum hole diameter and maximum shaft diameter, both of which result in the greatest possible amount of material being used. See **Figure 27-6.** 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 27-7.** LMC is indicated by an *L* within a circle.



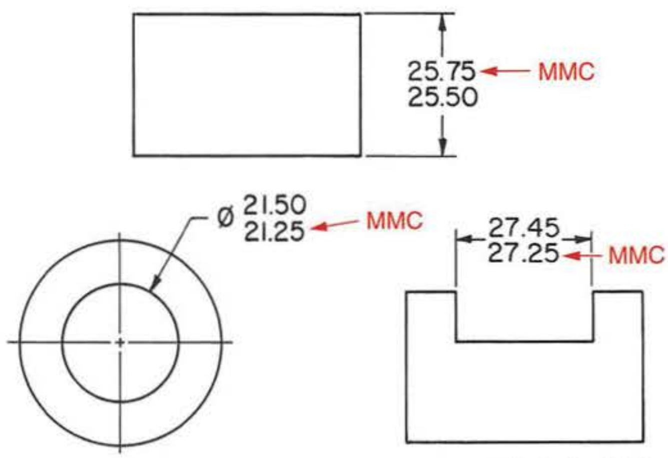
Goodheart-Willcox Publisher

Figure 27-4. Datums are exact points, axes, or planes from which features of a part are located.



Goodheart-Willcox Publisher

Figure 27-5. A datum feature is a physical feature on a part used to establish a datum.



Goodheart-Willcox Publisher

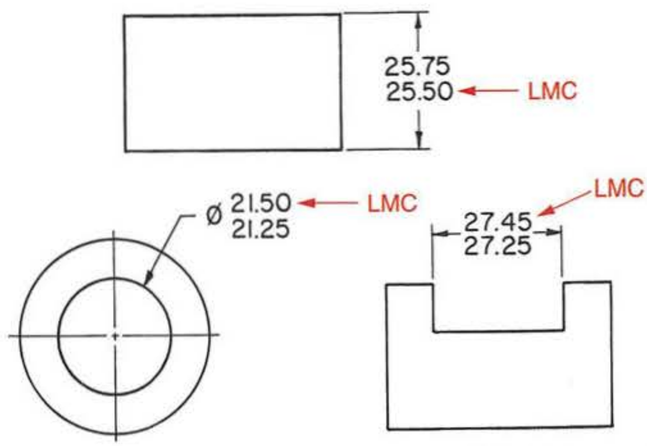
Figure 27-6. Maximum material condition (MMC) indicates that the size of a feature contains the maximum amount of material within the stated tolerance limits.

Regardless of feature size (RFS) specifies that the size of a feature tolerance must not be exceeded. Sometimes, RFS is indicated by an *S* within a circle. In 1994, the requirement to specify RFS with a symbol was removed from the ASME standard, but some older documents still have the symbol. Today, RFS is assumed for all geometric tolerances unless otherwise specified.

The maximum and minimum sizes of a feature are called the *limits of size*, **Figure 27-8**. The measured size of a part after it is manufactured is its ***actual size***.

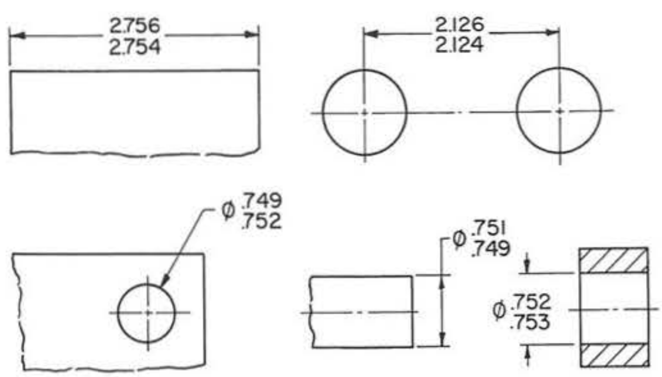
27.2 Application of Geometric Dimensioning and Tolerancing

A datum identification symbol, **Figure 27-9**, consists of a square frame that contains the datum reference letter. All letters except *I*, *O*, and *Q* may be used. A rectangular frame



Goodheart-Willcox Publisher

Figure 27-7. Least material condition (LMC) indicates that the size of a feature contains the least amount of material within the stated limits of size.

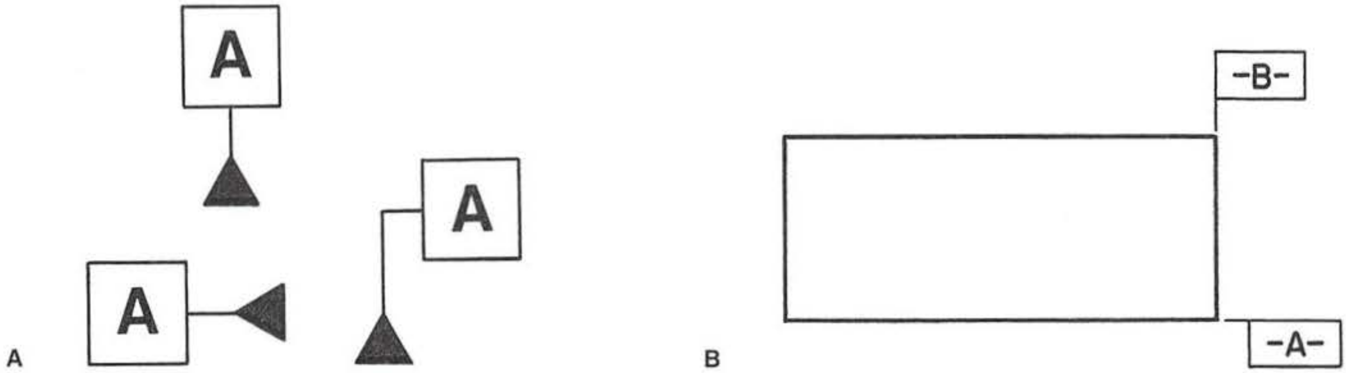


Goodheart-Willcox Publisher

Figure 27-8. Limits of size are the maximum and minimum sizes of a feature.

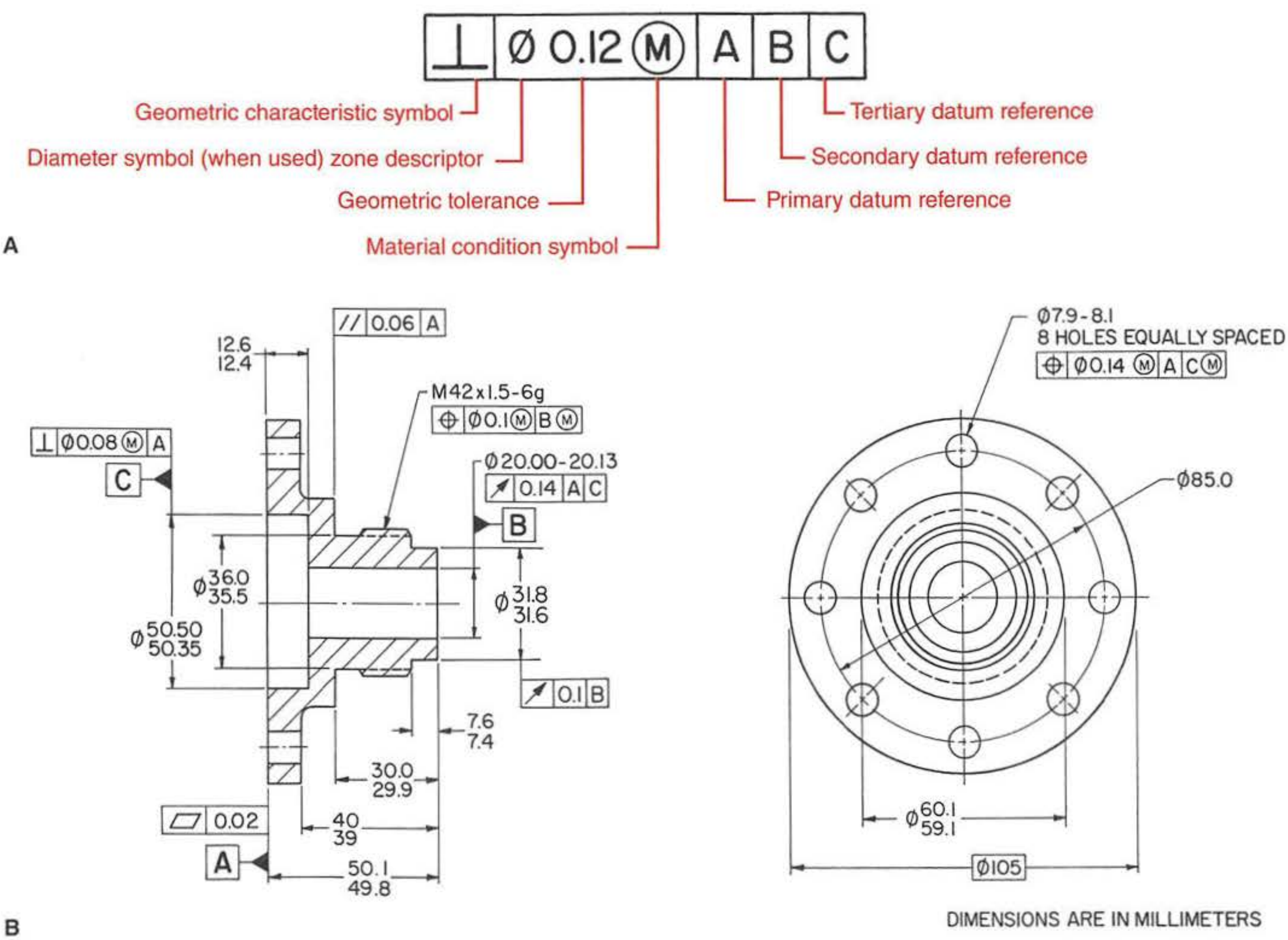
with the datum reference letter preceded and followed by a dash may be found on older drawings.

A ***feature control frame*** is used to define the geometric tolerancing characteristics of a feature that is related to a datum. It contains the geometric symbol, allowable tolerance, and 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 27-10**.



Goodheart-Willcox Publisher

Figure 27-9. 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 it is still found on old drawings.



Goodheart-Willcox Publisher

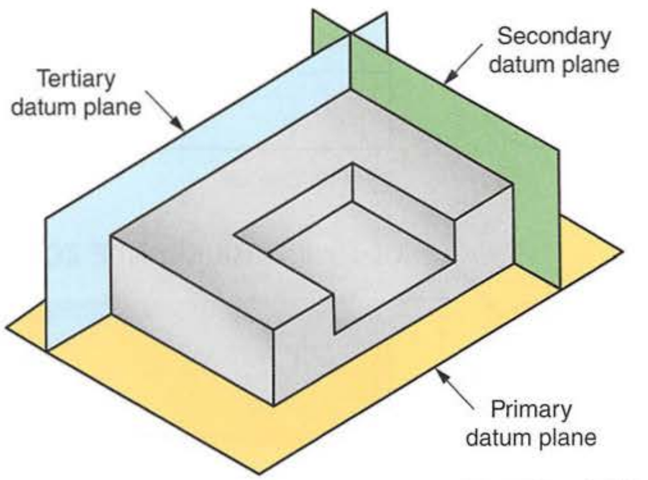
Figure 27-10. A feature control frame is used when a location or form tolerance is related to a datum. A—Components of a feature control frame. B—Feature control frames are used to specify tolerances on this drawing.

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 up to three mutually perpendicular planes, **Figure 27-11**.

27.3 Form Tolerances

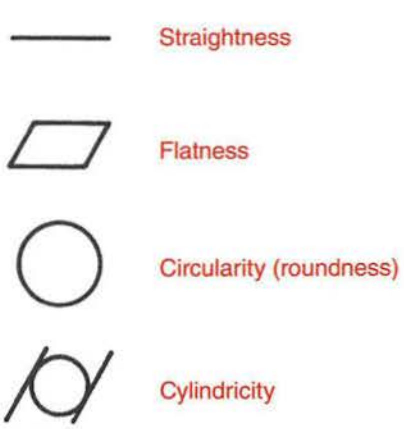
Form tolerances control flatness, straightness, circularity (roundness), and cylindricity. They are indicated by the symbols shown in **Figure 27-12**. Form tolerances control only the variation permitted on a single feature and are used when form variation is less than that permitted by size tolerance.

Flatness is a measure of the variation of a surface perpendicular to its plane. The flatness tolerance specifies the two parallel planes within which all points of a surface must lie, **Figure 27-13**.



Goodheart-Willcox Publisher

Figure 27-11. Datum references are perpendicular planes. The first datum referenced is the primary datum, followed by the secondary and tertiary datums.

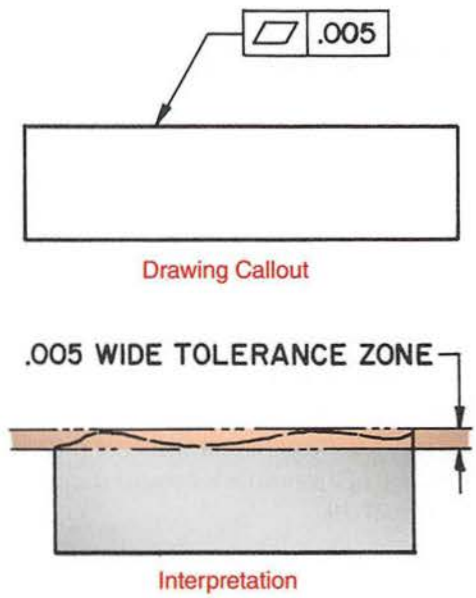


Goodheart-Willcox Publisher

Figure 27-12. Form geometric symbols.

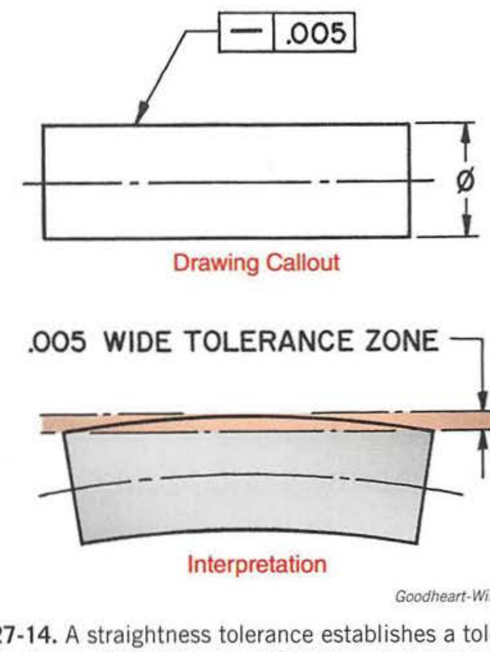
Straightness describes how close the surface of an object is to a perfectly straight line. A straightness tolerance establishes a tolerance zone of uniform width along a line. All elements of the surface must lie within this zone, **Figure 27-14**.

Circularity (roundness) 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 tolerance specifies a tolerance zone bounded by two concentric circles, indicated on a plane perpendicular to the



Goodheart-Willcox Publisher

Figure 27-13. The flatness tolerance specifies the two parallel planes within which a surface must lie.



Goodheart-Willcox Publisher

Figure 27-14. A straightness tolerance establishes a tolerance zone of uniform width along a straight line. All elements of the surface must lie within this zone.

axis of a cylinder or a cone, within which each circular element must lie. It is a single cross-sectional tolerance. See **Figure 27-15**.

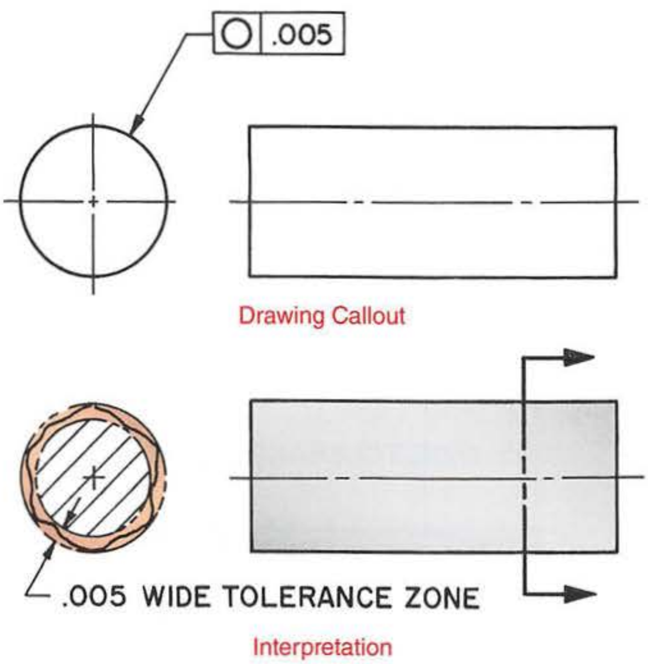
Cylindricity represents a surface in which all points are an equal distance from a common center. The cylindricity tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length. It consists of two concentric cylinders within which the actual surface must lie. This tolerance covers both the circular and longitudinal elements. See **Figure 27-16**.

27.4 Profile Tolerances

A ***profile 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 within which elements of the surface must be contained. The symbols used to indicate profile tolerances are shown in **Figure 27-17**.

A ***profile of a line tolerance*** is a two-dimensional (cross-sectional) tolerance zone extending along the length of the element. It is located using basic dimensions, **Figure 27-18**.

The ***profile of a surface 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 27-19**.



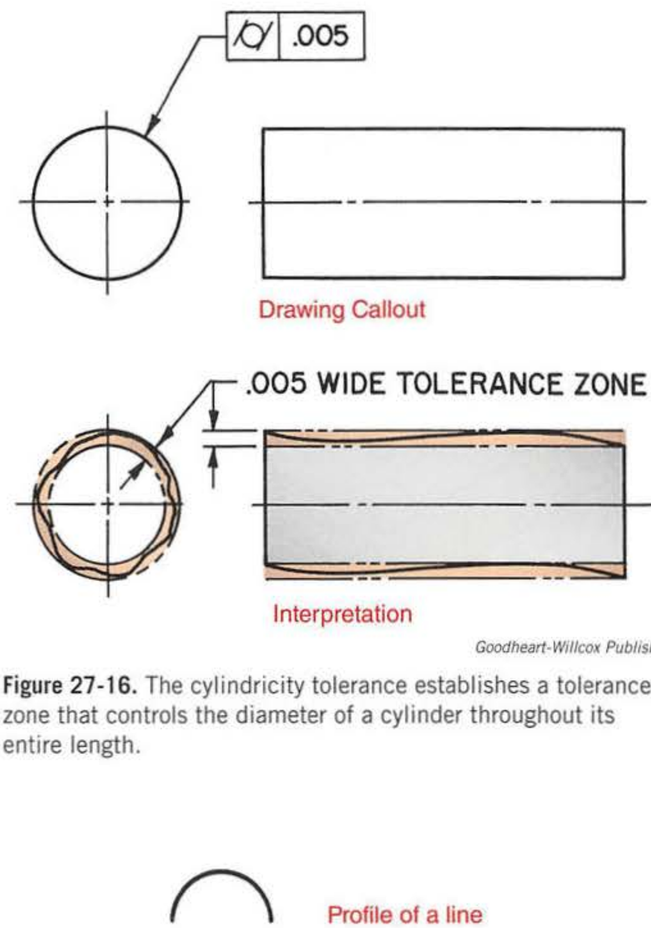
Goodheart-Willcox Publisher

Figure 27-15. A circularity 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.

27.5 Orientation Tolerances

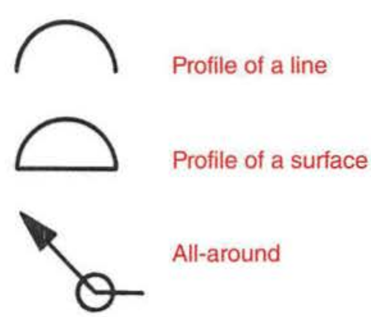
Orientation tolerances control the degree of parallelism, perpendicularity, or angularity of a feature with respect to one or more datums. There are three orientation tolerances, **Figure 27-20**.

Angularity locates a surface or axis at a specified angle to a datum plane or axis. The specified angle must be other than 90°. An angularity tolerance establishes a tolerance zone defined by two parallel lines or planes or by 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 27-21**.



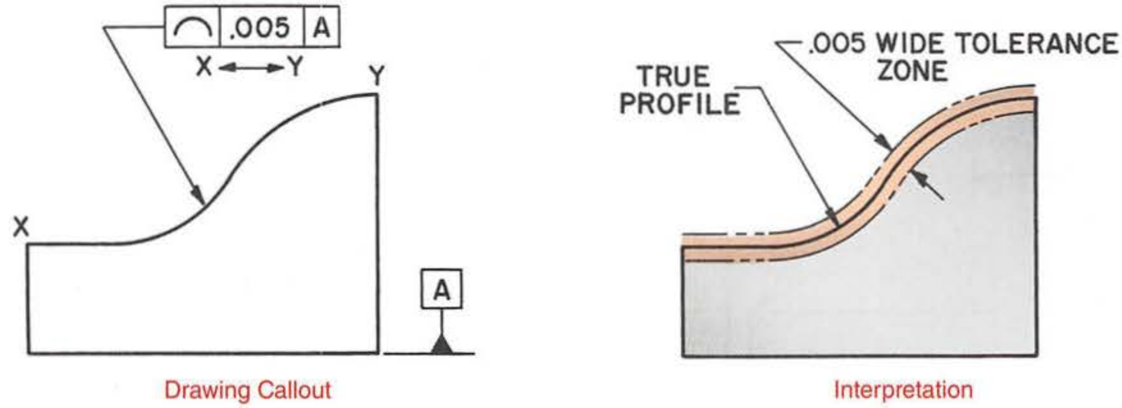
Goodheart-Willcox Publisher

Figure 27-16. The cylindricity tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length.



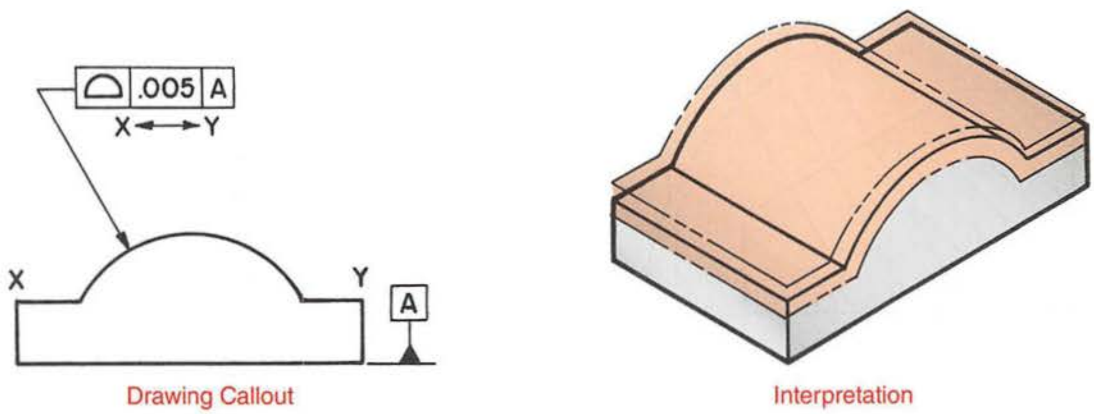
Goodheart-Willcox Publisher

Figure 27-17. Profile tolerance symbols. When a tolerance is specified for all sides of an object, the "all-around" symbol is used.



Goodheart-Willcox Publisher

Figure 27-18. A profile line tolerance is a two-dimensional tolerance zone extending along the length of the considered element.



Goodheart-Willcox Publisher

Figure 27-19. The profile surface tolerance is three-dimensional and extends along the length and width of the surface.

Goodheart-Willcox Publisher

Figure 27-20. Orientation tolerance symbols.

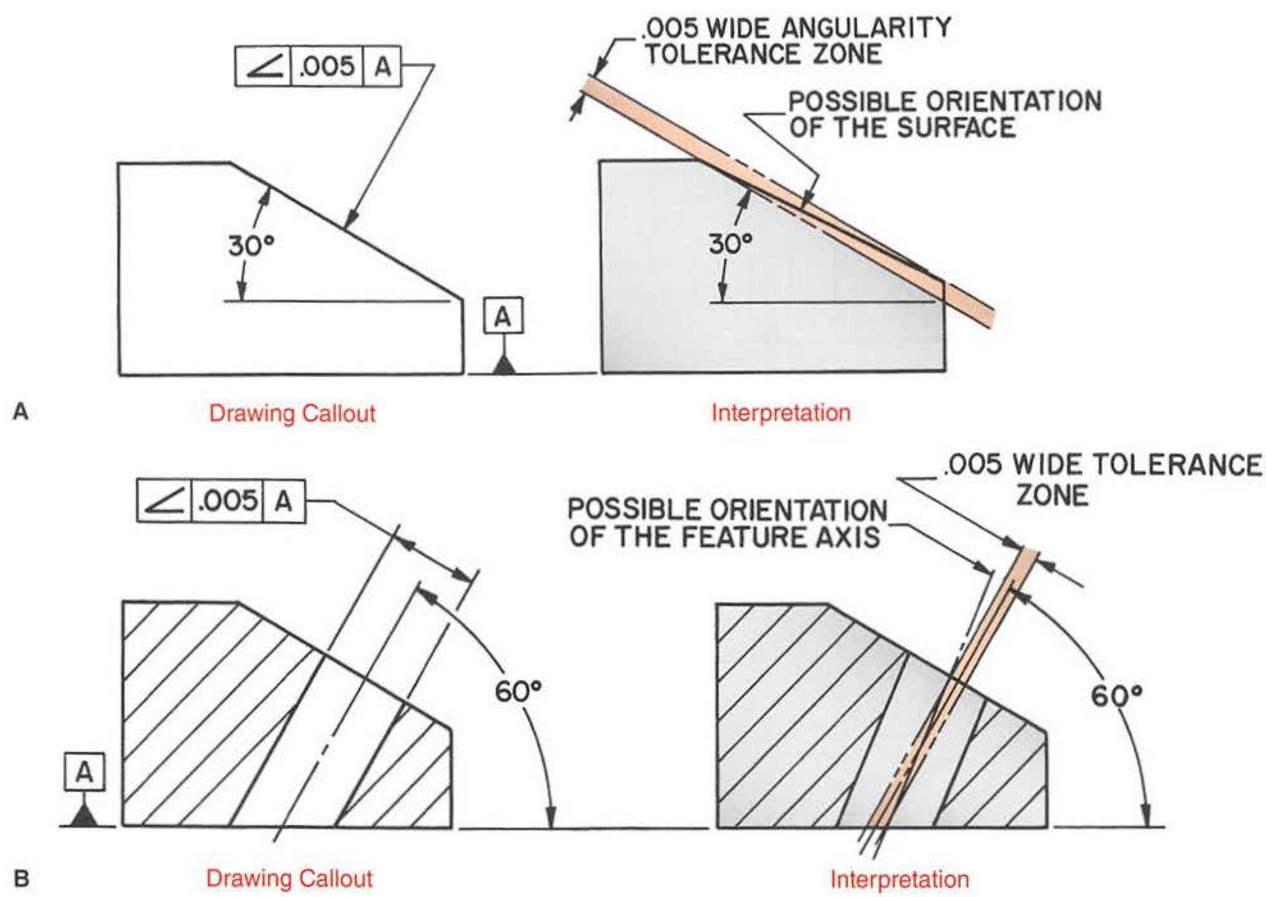
Perpendicularity describes how closely two lines or surfaces are to being perpendicular (at 90° to each other). A perpendicularity tolerance specifies a tolerance zone at right angles to a given datum or axis. It is described by two parallel lines or planes or by a cylindrical tolerance zone. The line, surface, or axis of the considered feature must lie within this zone, **Figure 27-22**.

Parallelism describes how close all elements of a line or surface are to being parallel (equidistant at all points) to a given datum plane or axis. A parallelism 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 27-23**.

27.6 Location Tolerances

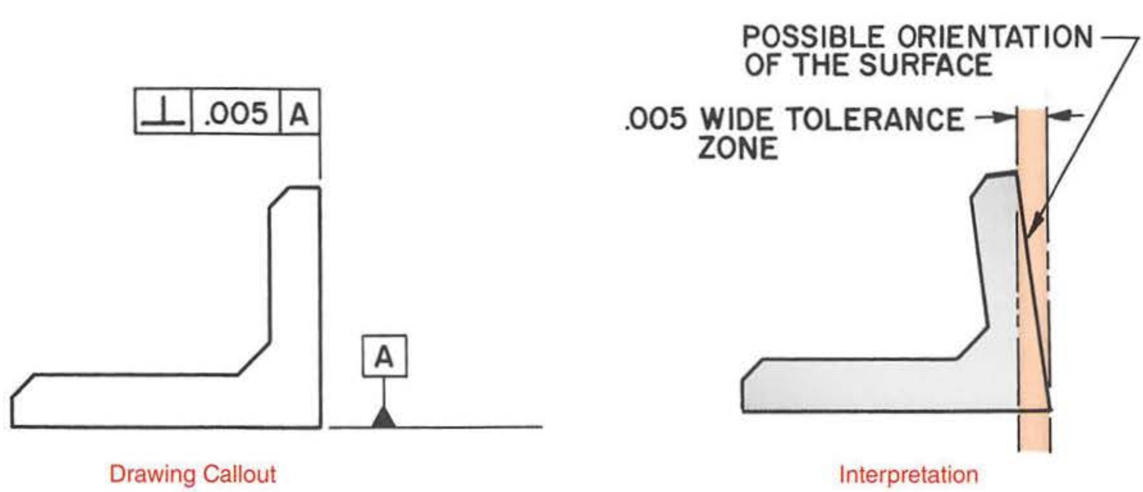
Location tolerances are used to establish the locations of features and datums. They define the zone within which the center, axis, or center plane of a feature may vary from a true (theoretically exact) position. Location tolerances are also known as *positional tolerances* and include position, concentricity, and symmetry. See **Figure 27-24**.

Basic dimensions establish the true position of a feature from specified datums and related features. A ***position tolerance*** establishes how far a feature may vary from its true position, **Figure 27-25**.



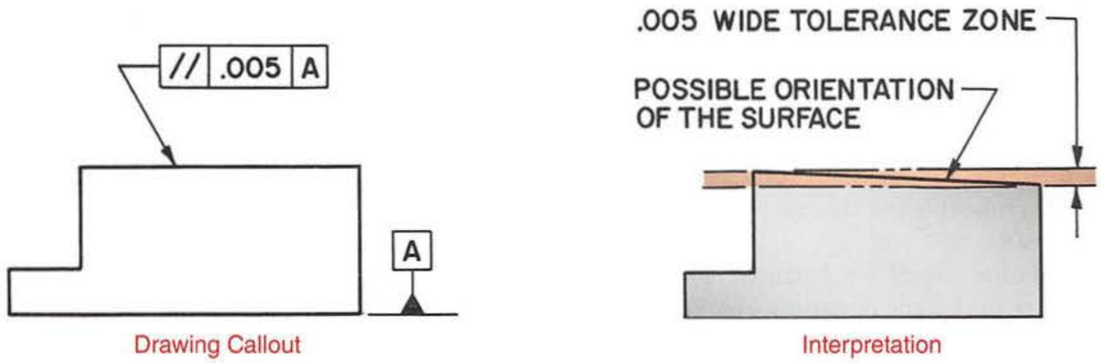
Goodheart-Willcox Publisher

Figure 27-21. An angularity tolerance establishes a tolerance zone defined by two parallel lines, two parallel planes, or a cylindrical zone at a specified basic angle other than 90°. A—Angularity of a surface. B—Angularity of an axis.



Goodheart-Willcox Publisher

Figure 27-22. The line, surface, or axis of a considered feature must lie within the perpendicularity geometric tolerance zone.

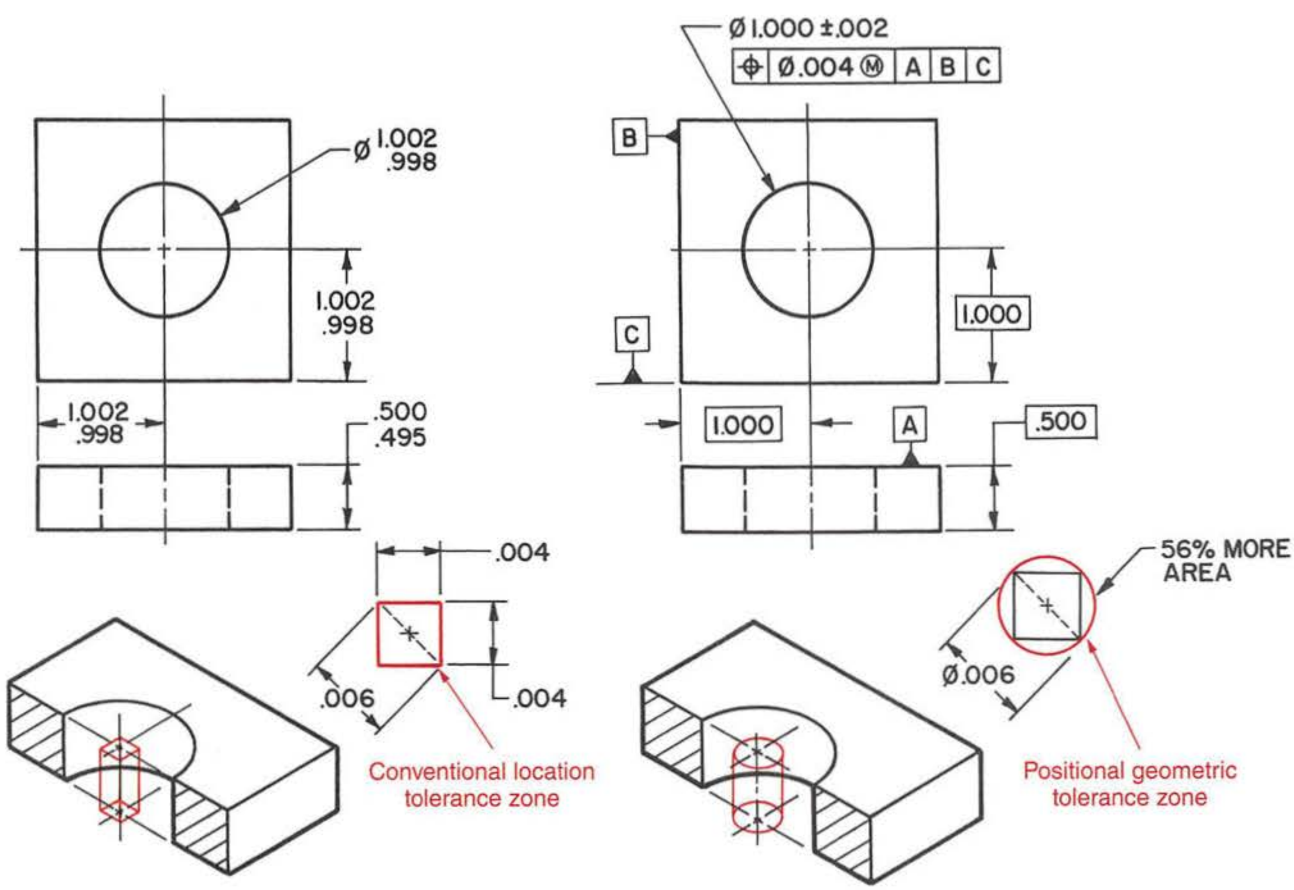


Goodheart-Willcox Publisher

Figure 27-23. A parallelism tolerance is a tolerance zone defined by two lines parallel to a datum within which the elements of a surface or axis must lie.

Goodheart-Willcox Publisher

Figure 27-24. Location or positional tolerance symbols.



Goodheart-Willcox Publisher

Figure 27-25. A position tolerance establishes how far a feature may vary from its true position.

Concentricity defines the relationship between the axes of two or more of an object's cylindrical features. A concentricity tolerance is expressed as a cylindrical tolerance zone. The axis or center point of this zone coincides with a datum axis, **Figure 27-26.**

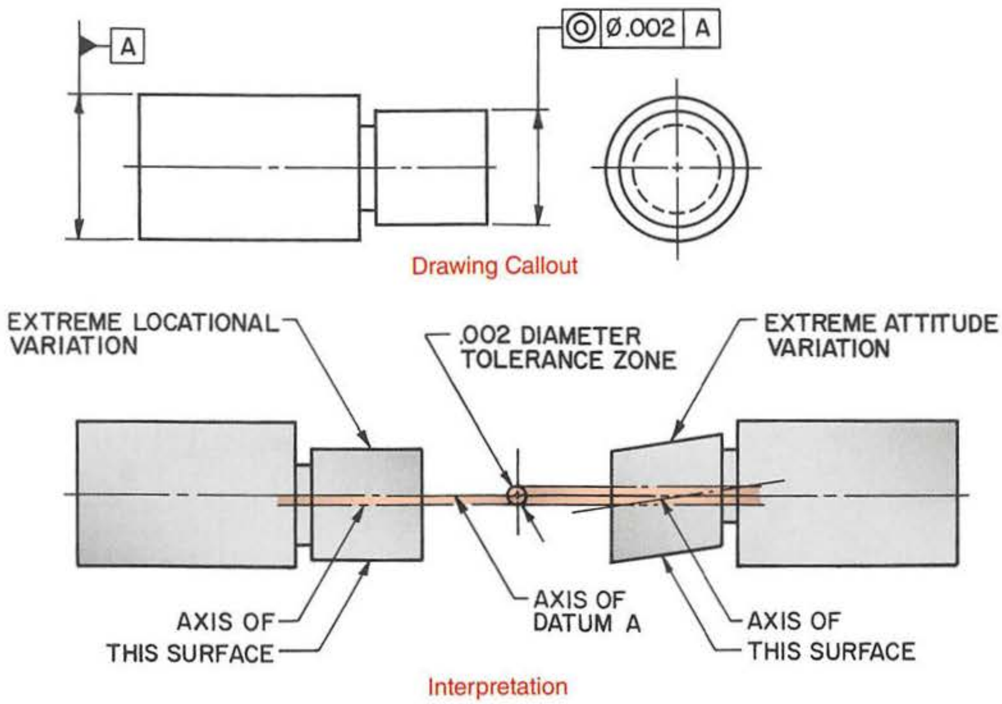
Since this tolerance is sometimes difficult and time-consuming to verify, runout or position geometric tolerances are often used instead.

Symmetry indicates equal or balanced proportions on either side of a central plane or datum, **Figure 27-27.** A symmetry tolerance is a zone within which the symmetrical surfaces align with the datum of a center plane or axis, **Figure 27-28.**

27.7 Runout Tolerances

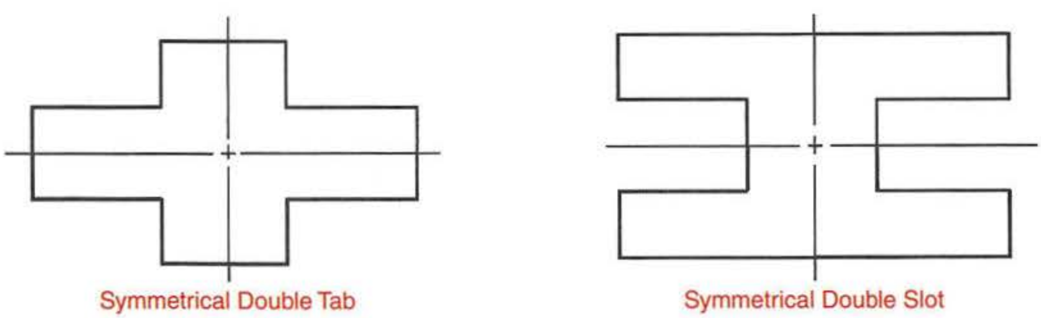
A ***runout tolerance*** is a combination of geometric tolerances used to control the relationship between one or more features of a part and an associated datum axis. There are two types of runout tolerances—total runout and circular runout. These tolerances are indicated by the symbols shown in **Figure 27-29.** Runout tolerances are used to control runout of surfaces around or perpendicular to a datum axis.

Total runout controls the circularity, straightness, angularity, and cylindricity of a part when applied to surfaces rotated around a datum axis, **Figure 27-30.** The entire surface must lie within the tolerance zone.



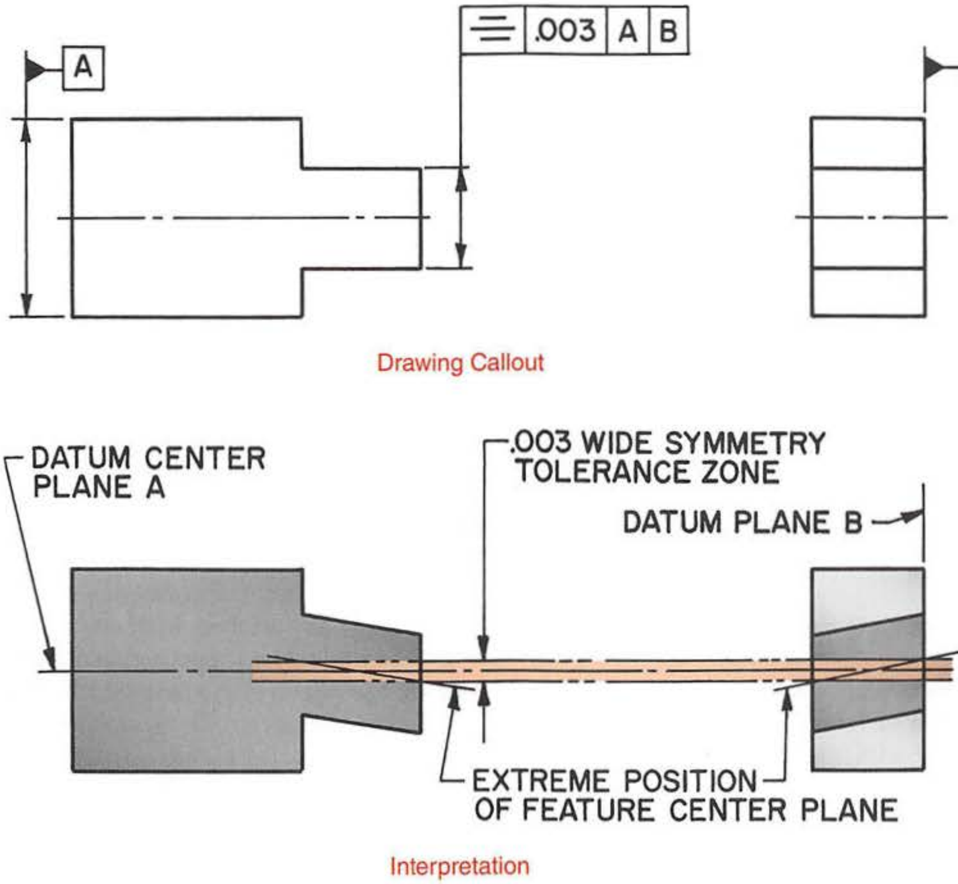
Goodheart-Willcox Publisher

Figure 27-26. A concentricity tolerance is expressed as a cylindrical tolerance zone. The axis or center point of this zone coincides with a datum axis.



Goodheart-Willcox Publisher

Figure 27-27. Symmetry indicates equal or balanced proportions on either side of a central plane.

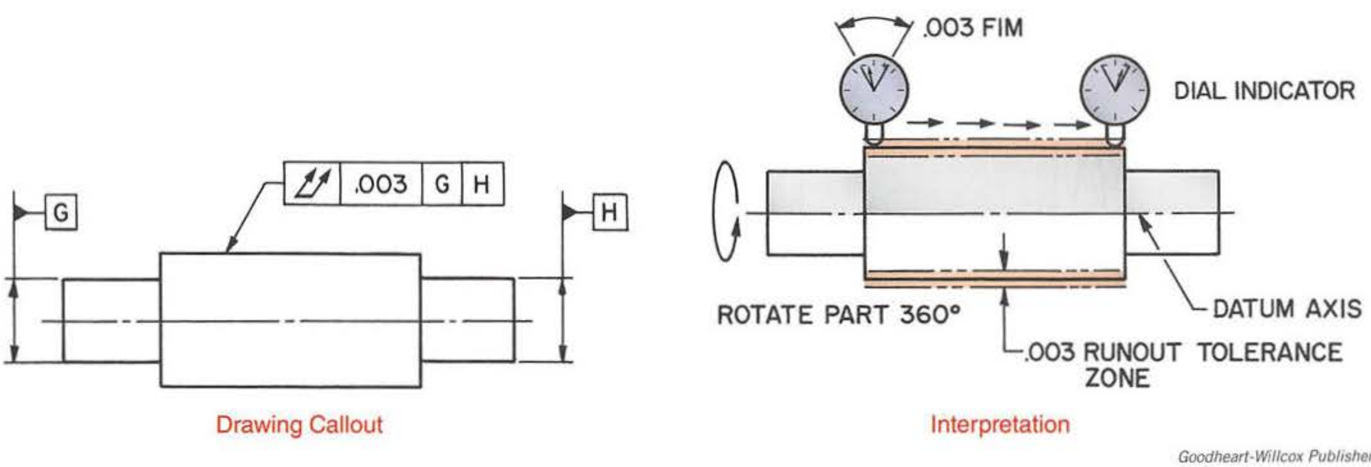


Goodheart-Willcox Publisher

Figure 27-28. A symmetry tolerance is a zone within which the symmetrical surfaces align with the datum of a center plane or axis.

Goodheart-Willcox Publisher

Figure 27-29. Runout tolerance symbols. Arrows may be filled or unfilled.



Goodheart-Willcox Publisher

Figure 27-30. Total runout controls the 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 27-31**. 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.

27.8 Bonus Tolerancing

Bonus tolerancing is a method of applying the unused tolerance of a part that is not at LMC or MMC to its mating part to provide a larger acceptable tolerance for the mating part. The concept of bonus tolerancing is perhaps the hardest of all the geometric dimensioning and tolerancing applications to grasp. While it is difficult, the application of bonus tolerancing allows manufacturers the opportunity to gain additional tolerance for true position based on the deviation in size of mating features.

For example, **Figure 27-32** shows a simple part with a pin in the center of the part. There is a tolerance for the pin diameter and a location tolerance for the center of the pin relative to the two datum surfaces shown. Note the feature control frame associated with the pin. The position symbol indicates that the tolerance information inside this feature control frame is related to the position of the boss in reference to datum surfaces A and B. The dimensions provided indicate that the center of the pin should be 1.000″ from the bottom of the part (Datum A) and 1.000″ from the left side of the part (Datum B).

Tolerances are not provided with the location dimensions. Rather, they are included within the feature control frame that has been placed below the diameter dimension for the pin. The targeted dimension for the pin is 0.750″. The pin's diameter can vary from this dimension by up to 0.002″, meaning that this pin can measure anywhere between 0.748″ and 0.752″ in diameter and still be acceptable.

To apply bonus tolerancing, consider a pin at less than MMC that still falls within the tolerance range. Because the actual pin is smaller than its maximum size, more location tolerance can be applied to its mating part using bonus tolerancing.

For example, if the pin on a single part measures 0.750″ in diameter, which is 0.002″ smaller than the MMC of 0.752″, then the diameter of the position tolerance can be increased from 0.002″ as specified in the feature control frame to 0.004″. In this case, the application of bonus tolerancing can be used to accept a part whose pin's location may be slightly outside the tolerance zone by applying the tolerance gained from the pin's smaller diameter to its location tolerance. The bonus tolerance is based on the pin diameter, as shown in the table in **Figure 27-33**.

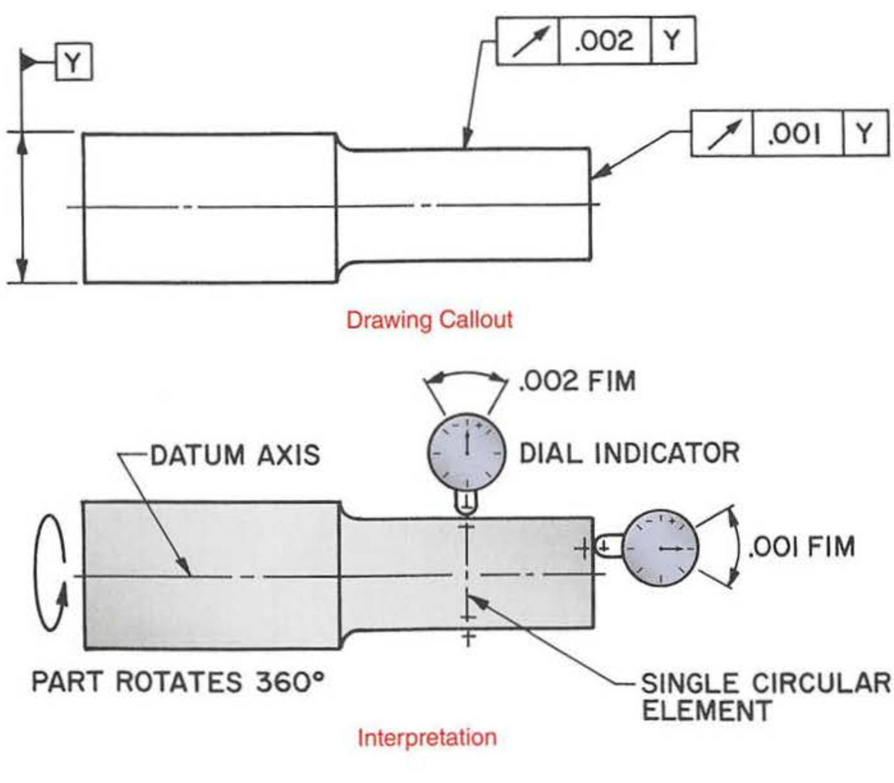
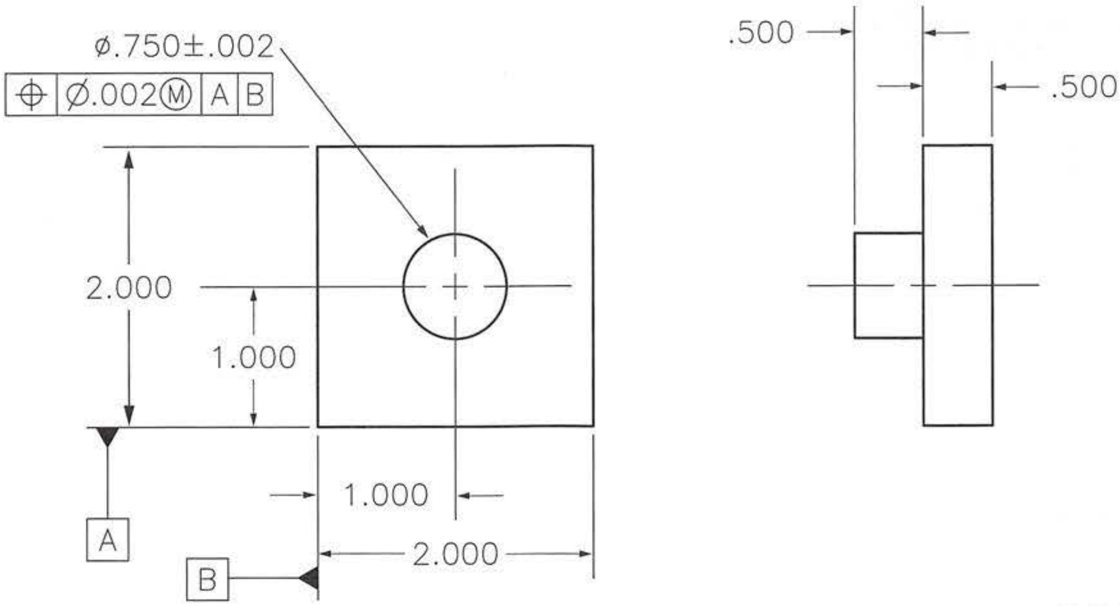


Figure 27-31. Circular runout controls circularity of a single circular cross section.



Goodheart-Willcox Publisher

Figure 27-32. Example drawing of a part with a pin in the center.

Bonus Tolerance Based on Pin Diameter	
Diameter Size Feature	**Diameter Position Tolerance Allowed**
0.752	0.002
0.751	0.003
0.750	0.004
0.749	0.005
0.748	0.006

Goodheart-Willcox Publisher

Figure 27-33. Bonus tolerance amounts for the pin example. Notice that the amount of location tolerance increases as the diameter of the pin decreases.

Bonus tolerancing cannot always be applied. It can be used only when a condition modifier is associated with the dimension of a feature (LMC or MMC). The application of bonus tolerancing often requires mating parts to be matched together and assembled at the manufacturer, which means that a replacement part may not fit properly with a part that was accepted using bonus tolerancing. While the application of bonus tolerancing has helped contain manufacturing costs, it has also resulted in requiring consumers to replace entire subassemblies instead of individual parts when repairs are needed. The decision to include bonus tolerancing in the design is usually made by the designer of the part, and machinists are responsible only for its application when inspecting parts for conformance.

Chapter Review

Summary

- Geometric dimensioning and tolerancing provides additional precision compared to conventional dimensioning.
- Geometric characteristic symbols standardized by the American Society of Mechanical Engineers provide clarity and precision in communicating design specifications.
- Feature control frames contain the geometric symbol, allowable tolerance, and datum reference letter(s) that control the characteristics of a feature.
- Form tolerances control flatness, straightness, circularity, and cylindricity.
- Profile tolerances control the outline or contour of an object and include specifications for profile of a line and profile of a surface.
- Orientation tolerances control the degree of parallelism, perpendicularity, or angularity of a feature with respect to one or more datums.
- Location tolerances establish the locations of features and datums.
- Runout tolerances are combinations of geometric tolerances that control the relationship between one or more features of a part and an associated datum axis.
- Bonus tolerancing can decrease manufacturing costs, but it also decreases the interchangeability of the parts.

Review Questions

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

1. A dimension providing the theoretical exact location of a feature is a(n) ______ dimension.
2. Dimensions placed between parentheses are ______ dimensions.
3. Define the term *maximum material condition (MMC).*
4. Define the term *least material condition (LMC).*
5. When is a feature control frame used?
6. List the four form tolerances and sketch the symbol used to represent each tolerance.
7. What are the two types of profile tolerances, and what is the major difference between them?
8. What is the purpose of an angularity tolerance?
9. Name the three types of location tolerances.
10. How is circular runout measured?
11. What is bonus tolerancing?
12. Explain the advantages and disadvantages of bonus tolerancing.

CHAPTER **28**

Metal Characteristics

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:
- Explain how metals are classified.
- Describe the characteristics of ferrous, nonferrous, high-temperature, and rare metals.
- Recognize the hazards that are posed when certain metals are machined.
- Explain the characteristics of some reinforced composite materials.

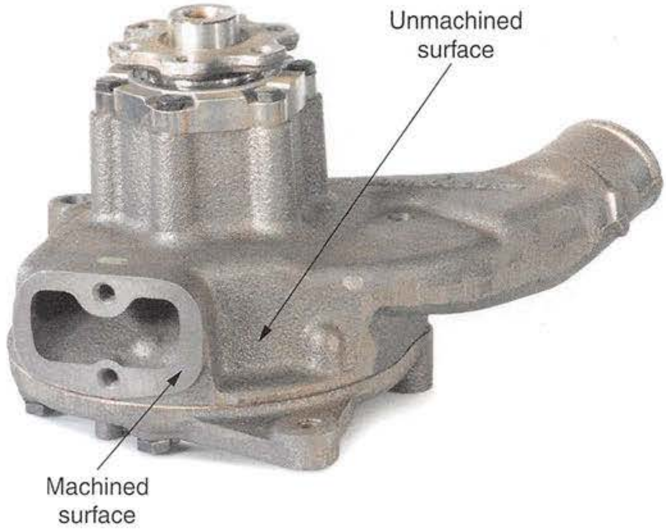
Technical Terms

alloy
Aluminum Association Designation System
base metal
carbon content
cold-finished steel
composite
ductility
ferrous
high-carbon steels
high-speed steels (HSS)
honeycomb sandwich panels
hot-rolled steel
low-carbon steels
medium-carbon steels
mild steel
nonferrous
red hardness

More than a thousand different metals and alloys are used by the metalworking industry. Most of them are worked in the machine shop. However, a "new materials revolution" is taking place, especially in the aerospace and medical equipment industries. Many nonmetals and composites of metals and nonmetals are rapidly emerging in applications that were once the exclusive domain of metals.

Can you tell by looking at a piece of metal whether it is ***ferrous*** (containing iron), or ***nonferrous*** (containing no iron)? Is it an ***alloy*** (a mixture of two or more metals)? Could it be a ***base metal*** (a pure, nonferrous, nonprecious metal), such as tin, copper, or zinc? From a practical point of view, it is almost impossible to find out much about a piece of metal by just looking at it.

A machinist is not expected to know everything there is to know about all metals. However, a working knowledge of common materials, and the terms associated with them, is essential. In addition to the conventional metals worked in a modern machine shop, several newer materials are described in this chapter.



Shell114/Shutterstock.com

Figure 28-1. Cast iron water pump. Note that the machined surfaces on the part are shiny silver, while the unmachined surfaces are dull gray with a pebbled texture.

28.1 Classifying Metals

Modern industrial metals can be classified into several categories:

- Ferrous metals.
- Nonferrous metals.
- High-temperature metals.
- Rare metals.

Because of space exploration and military research, great strides have been made in the development of the last two groups in recent years. Refer to the *Reference Section* of this text for tables that show physical properties of metals, dimensional tolerances, and feeds and speeds for machining.

28.2 Ferrous Metals

Irons, steels, and their alloys, make up the family of ferrous metals. For simplicity and easier understanding, ferrous metals can be further subdivided into various categories.

28.2.1 Cast Irons

Cast irons are iron alloys that contain 2.0% to 5.0% carbon with small quantities of silicon and manganese. There may also be traces of other elements in the alloy.

Of the cast irons, gray iron and malleable iron are most widely used in industry. They can be found in great quantities in automotive, railroad, farm equipment, and machine tool bodies. See **Figure 28-1.**

Malleable iron can be hammered into shape without fracturing. Most cast irons can be readily machined once the hard surface scale has been penetrated. Carbide cutting tools are recommended because of the abrasive nature of the iron scale. No cutting fluids should be used on cast iron. Compressed air is recommended if a coolant is needed.

SAFETY NOTE

Use extreme care when using compressed air as a coolant. To prevent injury, chips and dust must be contained.

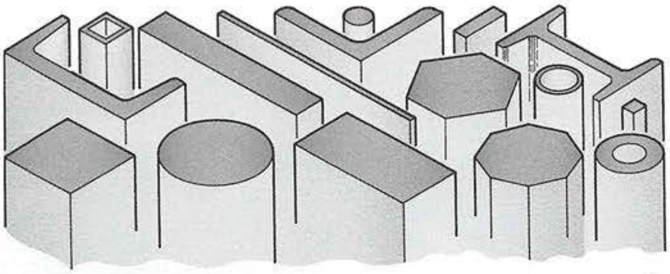
28.2.2 Steels

Steel is often considered the "backbone" of the metalworking industry. Carbon steel is very common. It is an alloy of iron and various alloying elements, with carbon as the major alloying agent. The alloying elements give the iron the characteristics needed to perform a specific job.

The physical properties of steel are unique. Steel can be made soft enough to be easily machined. By careful heat treatment, the soft steel can be transformed into a very hard material. By slightly varying the heat treatment procedure, steel can be given a hard, wear-resistant surface while retaining a soft, tough core to resist breaking. The magnetic and electrical qualities of steel also make it ideal for many electrical applications.

Carbon steels are classified according to their ***carbon content*** (amount of carbon they contain). The carbon content is measured in percentage or in points (100 points equal 1%). Carbon steels are available in all standard mill forms, **Figure 28-2.**

Low-carbon steels do not contain enough carbon (less than 0.30%, or 30 points) to be hardened. They are easy to work and can be case-hardened by heating them while



Goodheart-Willcox Publisher

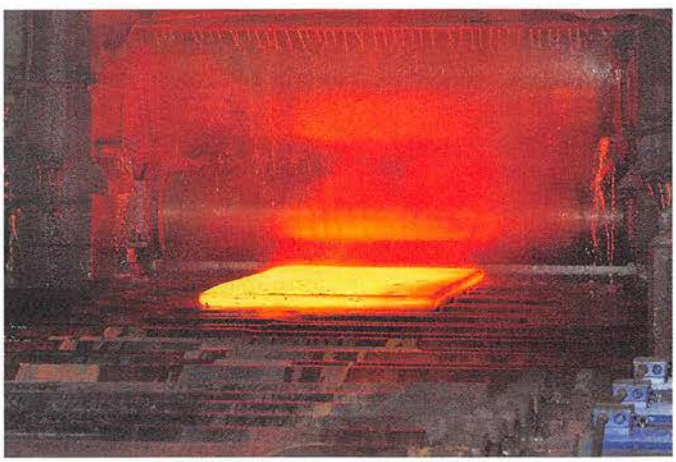
Figure 28-2. A small sample of the hundreds of shapes in which steel stock is available.

exposing their surface to an external source of carbon. Low-carbon steel is often called ***mild steel*** or machine steel. It is used for nuts, bolts, screws, gun parts, precision shafting, tie rods, tool cylinders, and similar applications.

Hot-rolled steel is steel that has been rolled to finished size while hot, **Figure 28-3**. It is easily identified by its black oxide surface scale. ***Cold-finished steel*** is steel that has been "pickled" or treated with a dilute acid solution to remove the oxide coating. After pickling, the steel is drawn or rolled to its finished size and shape while cold. Cold-finished steel is characterized by a smooth, bright finish, **Figure 28-4**. Cold-finishing improves the machinability of the steel.

Medium-carbon steels contain 0.30% to 0.60% (30 to 60 points) carbon. The carbon content is sufficient to allow partial hardening with proper heat treatment. The heat treatment process also improves the strength of the steel. Available in all standard mill forms, medium-carbon steels are used for items such as machine parts, automotive gears, camshafts, crankshafts, cap screws, and precision shafting.

High-carbon steels contain 0.60% to 1.50% (60 to 150 points) carbon. They are available in hot-rolled form. However, some high-carbon steel shapes may be purchased with ground surfaces. Drill rod and ground flat stock are examples.



Mircea BEZERGHEANU/Shutterstock.com

Figure 28-3. Hot-rolled steel is formed into the desired shape and thickness while it is still hot.

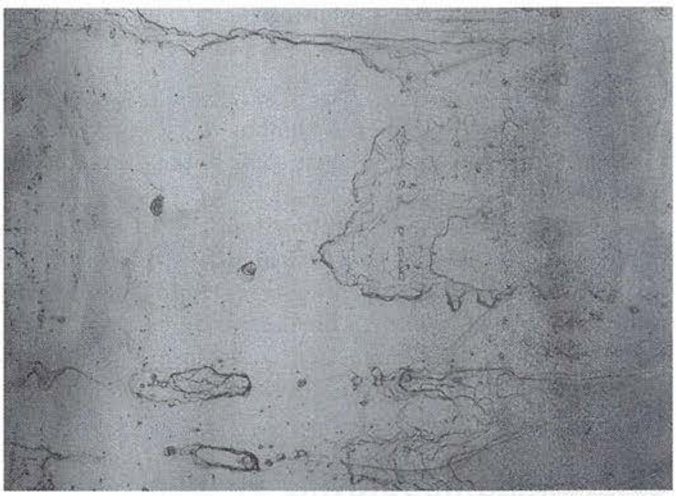
High-carbon steels are used in products that must be heat-treated. Applications include heavy machinery parts, control rods, tools, springs, and a variety of agricultural equipment.

Adding controlled amounts of sulfur or lead to carbon steels results in improved machinability without greatly affecting the metal's mechanical properties. Usually, machining involves the removal of considerable metal. Machining of high-carbon steels is economical only with machine tools capable of high cutting speeds.

28.2.3 Alloy Steels

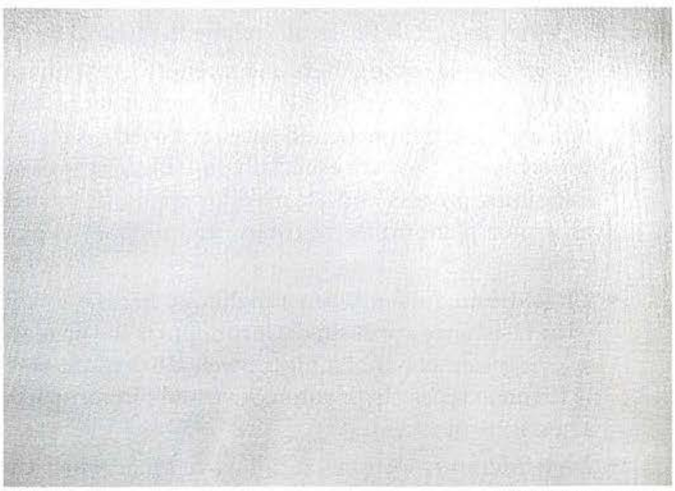
Alloy steels have other metal elements added to change their characteristics. Alloy steels are more costly to produce than carbon steels because of the increased number of operations that must be performed in their manufacture, **Figure 28-5**.

Elements such as nickel, chromium, molybdenum, vanadium, manganese, and tungsten are used to make alloy steels



A

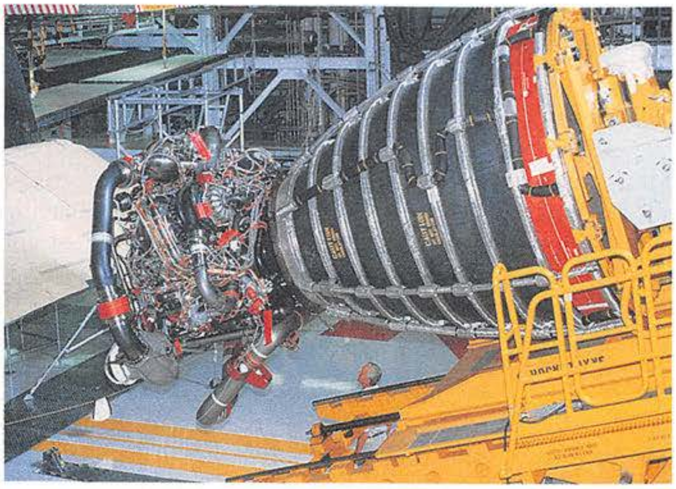
upixa/Shutterstock.com



B

R-studio/Shutterstock.com

Figure 28-4. Comparison of the appearance of hot-rolled steel and cold-finished steel. A—Hot-rolled steel has a black oxide coating. B—Cold-finished steel has a smooth, shiny surface.



NASA

Figure 28-5. Many alloy and high-temperature steels were used in the manufacture of this large rocket engine (for size comparison, note the technician at bottom center). The engine has been designed to safely handle the super-cold temperature of liquid oxygen at one end and the blast furnace temperature of exhaust gases at the other, while operating in the cold regions of space.

harder, stronger, or tougher. A combination of two or more of these elements usually imparts some of the characteristic properties of each.

Chromium-nickel steels, for example, develop good hardening properties with good ***ductility*** (a property of metal that permits permanent deformation by hammering, rolling, and drawing without breaking or fracturing). Chromium-molybdenum combinations develop excellent hardenability with satisfactory ductility and a certain amount of heat resistance.

28.2.4 Metallic Alloying Elements

Metallic elements that may be added to alloy steels, and properties they impart to the steel, include the following:

- **Nickel.** Imparts toughness and strength, particularly at low temperatures. Nickel steels permit more economical heat treatment and have improved resistance to corrosion. They are especially suitable for the case-hardening process and are used for applications such as armor plate, roller bearings, and aircraft engine parts.
- **Chromium.** Added when toughness, hardness, and wear resistance are desired. Chromium is the primary alloying element in stainless steel. Chromium steel is found extensively in automotive and aircraft parts, **Figure 28-6.**
- **Molybdenum.** Used as an alloying agent when the steel must remain tough at high temperatures.
- **Vanadium.** An alloying element that gives steel a fine grain structure and increased toughness at high temperatures.

US Air Force Thunderbirds

Figure 28-6. Chromium steel is used extensively in the landing gear assembly of this aircraft. It has the ability to withstand the repeated shocks of landing the aircraft.

- **Manganese.** Purifies steel and adds strength and toughness. Manganese steel is used for parts that must withstand shock and resist wear, **Figure 28-7.**
- **Tungsten.** Gives steel a fine, dense structure and improved heat treatment qualities. It is one of the principal alloying agents in many tool steels. Tools made with these steels retain their strength and hardness at high temperatures.
- **Cobalt.** The chief alloying element in high-speed steels because it improves the ***red hardness*** (quality of remaining hard when red-hot) of cutting tool materials. Wear resistance is also improved.

28.2.5 Tool Steel

Tool steel is the term usually applied to the steels used to make tools that cut, shear, or form materials. They may be either carbon or alloy steels. Tool steels in the lower carbon content range (0.70% to 0.90% or 70 to 90 points) are used in tools that are subjected to shock. The low carbon content gives the steel the toughness it needs to withstand impact. Higher-carbon-content tool steels (1.10% to 1.30% or 110 to 130 points) are used to make tools with sharp cutting edges. The hardness of these steels prevents the cutting edge from dulling with use.

Drills, reamers, milling cutters, punches, and dies are made from alloy tool steels. Although several tool steels can be hardened using water as a quenching medium, most must be hardened in oil or in air. These are known as oil-hardened and air-hardened steels.

kaband/Shutterstock.com

Figure 28-7. Many components of earthmoving vehicles are made from manganese steel. This type of steel is strong and tough enough to withstand constant abrasive wear.

Some tool steels are also classified as ***high-speed steels (HSS)***, because they are capable of making deeper cuts at higher cutting speeds than regular tool steels. They have red hardness, or the ability to retain their hardness at high temperatures, **Figure 28-8**. They also possess high abrasion resistance. Despite the development and widespread use of cemented carbides and ceramics, high-speed steel remains a major cutting tool material.

Matee Nuserm/Shutterstock.com

Figure 28-8. End mill cutter made from high-speed steel.

CAREER CONNECTION

Metallurgist

What does a metallurgist do?

Metallurgists are materials engineers who are trained in the extraction, refining, testing, alloying, and fabrication of metals. There are three types of metallurgists:

- Chemical metallurgists study the properties of different metals and develop new ways to strengthen existing metals. They also test ores for extraction and alloying.
- Physical metallurgists research how different metals react under the stress of use. They also study accidents related to the failure of metal materials.
- Process metallurgists design metal parts and the processes used to shape them. They may build parts themselves as well.

What education and skills are needed to be a metallurgist?

Future metallurgists need a strong background in engineering, chemistry, and mathematics. A bachelor's degree in materials science, metallurgy, or engineering is required. Getting a master's degree is desirable for advancement along with professional licensure.

What is it like to be a metallurgist?

Metallurgists often work in laboratories and offices where they test and research metals, design new processes, and write on their findings. They can also be found in the field sampling ores and on the floor of manufacturing facilities. They are involved in aerospace production, parts manufacturing, and scientific research and development.

According to the *Occupational Outlook Handbook*, annual wages for metallurgists and materials engineers ranged from $80,000 to $110,000 in recent years.

28.2.6 Tungsten Carbide

Tungsten carbide is the hardest known metal. It is almost as hard as a diamond. The metal is shaped by molding tungsten, carbon, and cobalt powders under heat and pressure in a process known as *sintering*. The metals fuse together without melting.

Although it is not a true steel, tungsten carbide is usually classified with the steels. Tools made from this family of materials can cut many times faster than cutting tools made from high-speed steel, **Figure 28-9**.

Applying a 0.0001″ (0.002 mm) thick coating of titanium nitride (TiN) and titanium carbide (TiC) to the surface of tungsten carbide tools extends their life 3 to 8 times longer than uncoated tools. Material builds up on the cutting edge of uncoated tools, resulting in a ragged surface finish on the work. Coated tools resist buildup from the chips. They also run cooler, last longer, hold tolerances better, and produce a better surface finish. See **Figure 28-10**. Recommended tungsten carbide grade applications and product listings from various manufacturers are shown in chart form in **Figure 29-11**.

28.2.7 Stainless Steels

There are more than a hundred different stainless steels. However, one characteristic common to all of them is that they contain enough chromium to render them corrosion resistant. Stainless steels can be divided into three basic groups:

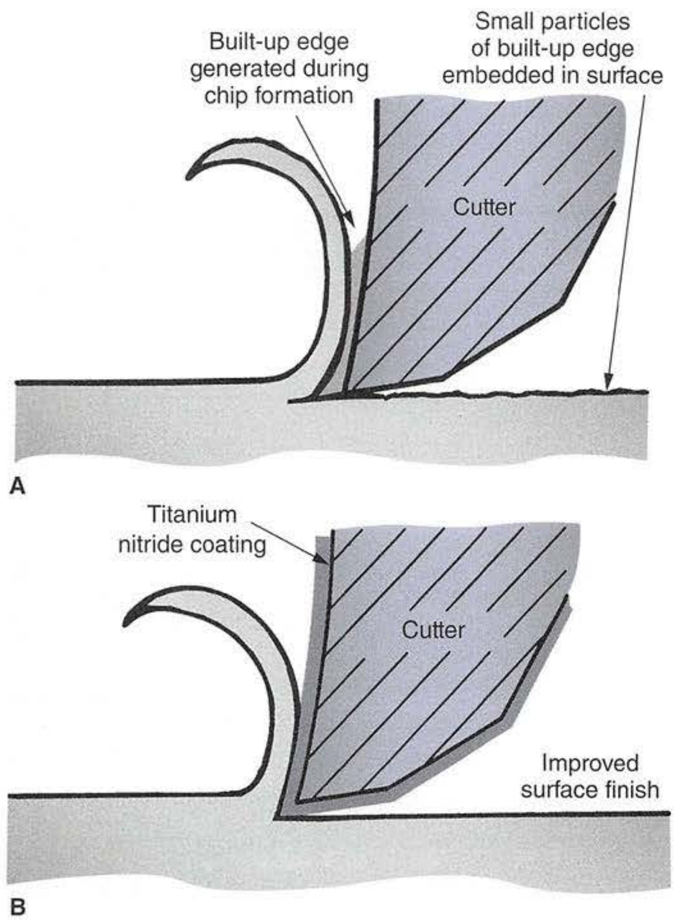
- **Austenitic.** This classification includes the chromium-nickel and chromium-nickel-manganese stainless steels. Generally, they can be hardened only by cold working and are nonmagnetic. The American Iron and Steel Institute (AISI) 300 series of stainless steels are in this category.
- **Martensitic.** These stainless steel alloys of iron, carbon, and chromium are characteristically magnetic in nature and obtain their hardness through normal heat treatment processes.
- **Ferritic.** These stainless steels have more than 18% chromium. They are magnetic and cannot be hardened.

Stainless steels can be machined using the same techniques used for mild steels. However, some precautions must be observed with stainless steel:

- Feeds must be high enough to ensure that the cutting edge(s) can get under the previous cuts and thus avoid the hardened portions.
- Tools must be as large as possible, because the life of the cutting edge(s) depends on good heat dissipation into the body of the cutting tool.

Valenite, Inc.

Figure 28-9. Cutting tools made from tungsten carbide can cut many times faster than high-speed steel cutters.



Goodheart-Willcox Publisher

Figure 28-10. Tool coating benefits. A—Uncoated cutting tools acquire a buildup of material on the cutting edge, which produces a ragged surface finish as parts of the buildup flake off. B—Tools coated with titanium nitride (TiN) and titanium carbide (TiC) resist the buildup from the chips. The coated cutting tools run cooler, stay sharp longer, and produce a better surface finish.

- Finishing cuts should be used when working to close tolerances.
- The machine should be adjusted so there is minimum play. Otherwise, the cutting tool may "ride" the work and glaze and/or harden the surface.

28.2.8 Identifying Steels

Because different kinds of steels look alike, several methods of identification have been devised. These include identification by chemical composition, mechanical properties, the ability to meet a standard specification or industry-accepted practice, and the ability to be fabricated. The shape and intended use for the metal can also determine the method of identification.

AISI/SAE Codes

The American Iron and Steel Institute (AISI) and the Society of Automotive Engineers (SAE) have devised almost identical standards that are widely used for identifying steel. Both systems use an identical four-digit code (some steels require a fifth digit) that describes the physical characteristics of the steels. The AISI system also makes use of a prefix letter (A, B, C, etc.) that indicates the steel manufacturing process used.

The four-digit code works as follows: The first digit classifies the steel. The second digit indicates the approximate percentage of the alloying element in the steel. The last two digits show the approximate carbon content of the steel in points (hundredths of one percent). For example, a steel designated SAE 1020 is a carbon steel with approximately

Recommended Tungsten Carbide Insert Grade Applications	
Grade C2 uncoated	For cast iron, nonferrous materials, and general-purpose use.
Grade C4 uncoated	Light finishing cast iron, nonferrous, and general-purpose.
Grade C6 uncoated	For steel, steel casting, malleable cast iron, stainless steels, and free cutting steels.
Grade C5–C6 Titanium nitride coated (TiN)	For carbon steels, tool steels, alloy steels, steel castings, malleable cast iron, austenitic and martensitic stainless steels, and free-cutting steels.
Grade C6–C7 Titanium carbide coated (TiC)	For steel, steel castings, malleable cast iron, nodular iron, and martensitic stainless steels.
Grade C2–C4–C6–C8 Aluminum oxide, ceramic coated	For carbon steels, tool steels, stainless steels, alloy steels, steel castings, gray cast iron, malleable cast iron, and nodular iron.

A

Tungsten Carbide Comparison Chart

Class	RTC	Newcomer	Carboloy	Iscar	Kennametal	Mitsubishi	Sandvik	Seco	Sumitomo	Valenite	V.R. Wesson	RTW
C-2	RTC2	N21	833	IC20	K-68	UTi20T HTi10	CG-20	HX	G10E	VC-1 VC-2/VC-28	2A5	CQ-2
C-6	RTC5	N60	395 78	IC54	K420	STi20 UTi20T	S-2 S-35	S-2 S-4	ST20E	VC-6	VR-75	Cy-5
TiN	RTC052	NN60	516	IC656	KC810 KC950	–	GC-425 GC-315 GC-225	TP15	AC720 AC815 AC15	VN-5 VC-7 VC-88 VO1	663 650 660	TRW-755
TiN & AlO_2 & TiN	RTC054	1000	550 560	IC635	KC850	U610	GC4025 GC235	TP10	–	VN2	653	718

B

Goodheart-Willcox Publisher

Figure 28-11. Recommendations for various tungsten carbide insert cutting tools. A—Recommended applications. B—Tungsten carbide insert comparisons.

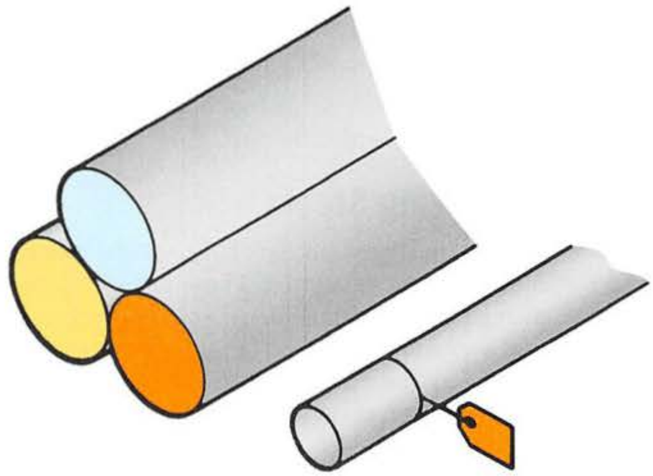
20 points or 0.20% carbon. The AISI and SAE four-digit code applies primarily to bar, rod, and wire products.

Color Coding

Color coding is another method of identifying the many kinds of steel. Each commonly used steel is designated by a specific color. The color coding is painted on the ends of the stock. In cases where the color code may not be visible on the ends of the stock, it can be placed on an attached tag. See **Figure 28-12**.

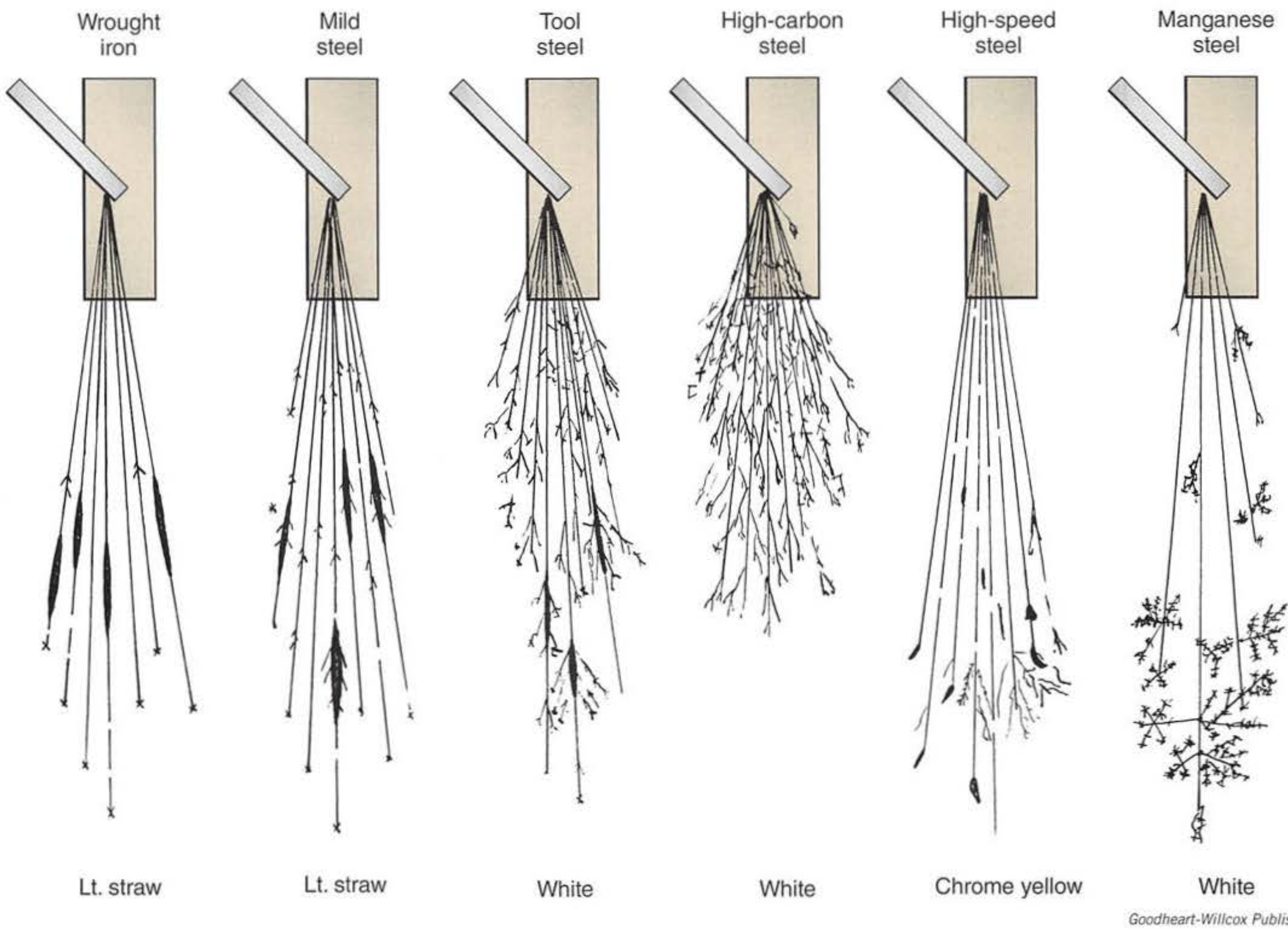
Spark Test

A spark test is also used at times to determine grades of steel. The metal is touched to a grinding wheel lightly and the resulting sparks are carefully observed. See **Figure 28-13**. The patterns and characteristics of the sparks created differ from alloy to alloy. Because of this, it is possible to identify an unknown steel by observing the sparks it makes and comparing them to the spark characteristics of the different steels.



Goodheart-Willcox Publisher

Figure 28-12. Steel bar stock can be color-coded to identify the type of steel.



Goodheart-Willcox Publisher

Figure 28-13. A spark test is sometimes used to determine the grade of steel. To perform the test, touch the steel lightly to a grinding wheel and observe the color and form of the resulting sparks.

SAFETY NOTE

When performing or observing a spark test, wear approved eye protection. The grinder's eye shield should be clean and in position. The tool rest must also be properly adjusted.

28.3 Nonferrous Metals

Many metals, including aluminum, magnesium, and titanium, do not have iron as their basic ingredient. These metals are known as *nonferrous metals*. Nonferrous metals have specific properties that make them ideal for tasks for which ferrous metals are not suitable. See **Figure 28-14**.

28.3.1 Aluminum

Aluminum has come to include a large family of aluminum alloys, not just a single metal. As first produced, aluminum is 99.5% to 99.76% pure. It is somewhat soft and not very strong.

The strength of aluminum can be greatly increased by adding small amounts of alloying elements, by heat treatment, or by cold working. A combination of the three techniques has produced aluminum alloys that, pound for pound, are stronger than structural steel. In addition to increasing strength, alloying elements can be selected to improve welding characteristics, corrosion resistance, machinability, and other desirable traits.

Classes of Aluminum Alloys

There are two main classes of aluminum alloys: wrought alloys and cast alloys. The shape of wrought alloys is changed mechanically by forging, rolling, extruding, hammering, or other techniques. Cast alloys are shaped by pouring metal into a mold and allowing it to solidify, **Figure 28-15**.

Each alloy is given an identifying number that contains a four-digit code plus a temper designation. This method of identifying aluminum alloys is known as the ***Aluminum Association Designation System***. The temper designation indicates the degree of hardness of the alloy. It follows the alloy identification number and is separated from it by a dash.

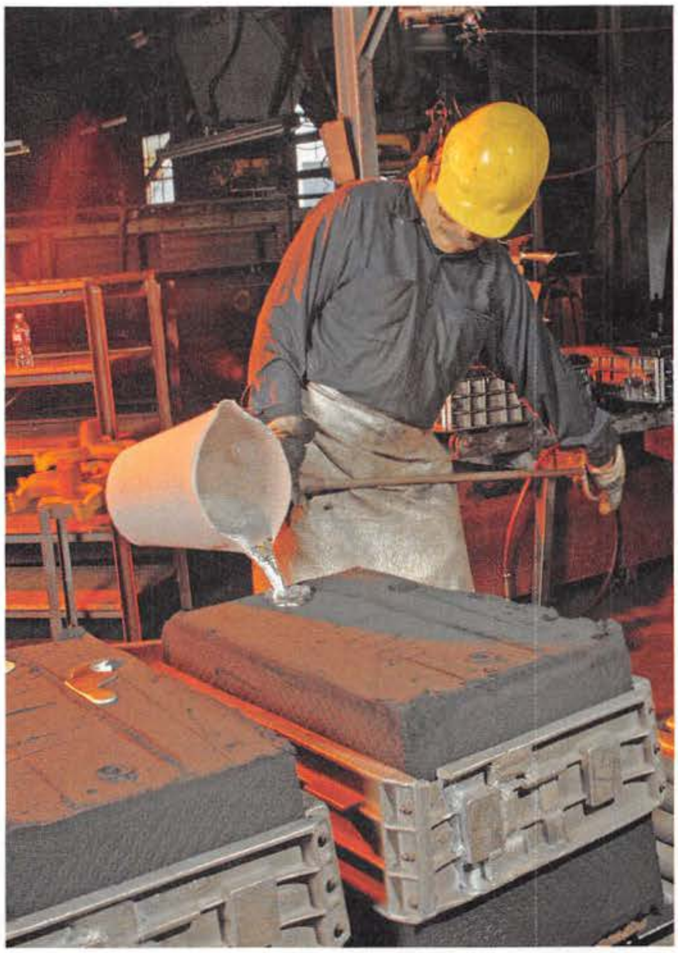
Aluminum alloys possess many desirable qualities. They are extremely strong and corrosion-resistant under most conditions. Aluminum alloys are lighter than most other commercially available metals. They can be shaped and formed easily, and they are readily available in a multitude of sizes, shapes, and alloys.

Machining Aluminum

Most of the wrought aluminum alloys possess excellent machining characteristics. They can be machined to intricate shapes at high cutting speeds. However, the makeup of an aluminum alloy can affect machinability. Some nonabrasive aluminum alloys (those containing copper, magnesium,

Alexandar Iotzov/Shutterstock.com

Figure 28-14. Aluminum is used in many airframes because of its high strength-to-weight ratio.



Chuck Rausin/Shutterstock.com

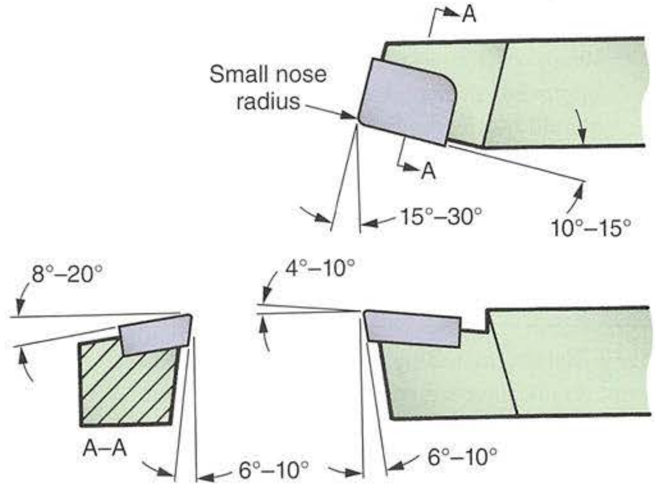
Figure 28-15. Aluminum casting. Note that the molten aluminum is the same silver color as solid aluminum.

or zinc) have improved machinability. Other alloys with abrasive constituents (such as silicon) reduce tool life, and machined surfaces may have a slightly gray finish with little luster.

Most aluminum alloys are easier to machine to a good finish when in full hard temper than when in an annealed state. Machining characteristics of the more commonly used aluminum alloys are listed below:

- **Number 1100 and 3003 alloys.** Have good machinability but are gummy in nature. Turnings are long and stringy, causing difficulty in chip disposal. Good results can be obtained if the cutting tools have large top and side rake angles, with keen, smooth cutting edges.
- **Number 5052 alloy.** Has turnings that are long and stringy, and the machined surface is not as good as on 3003. Machinability is good, however.
- **Number 5056 alloy.** Has good machinability with the advantage of fairly easy chip disposal.
- **Numbers 2017-T4 and 2014-T6 alloys.** Machine to an excellent finish. Of the two, 2014-T6 has better machinability because of the heat-treating method used. It causes greater tool wear.
- **Number 2024-T3 alloy.** Has good machining characteristics with properly sharpened and honed tools. Surface finishes are excellent.
- **Number 6061-T6 alloy.** Contains silicon and magnesium. It is more difficult to machine than the 2000 series alloys. Properly sharpened cutting tools and coolants with good lubricating qualities are essential. Fine finishes are obtainable with moderately heavy cuts.
- **Number 7075-T6 alloy.** The highest strength aluminum alloy commercially available. Machining qualities are good.

High-speed cutting tools produce satisfactory results when machining most aluminum alloys. However, the gumminess of the material can cause it to stick to the tool. As a result, tungsten carbide or ceramic tools are needed to produce the best results. The recommended lathe tool geometry for aluminum is shown in **Figure 28-16**.



Goodheart-Willcox Publisher

Figure 28-16. Configuration of a tungsten carbide lathe tool for machining aluminum.

GREEN MACHINING

Recycling Aluminum

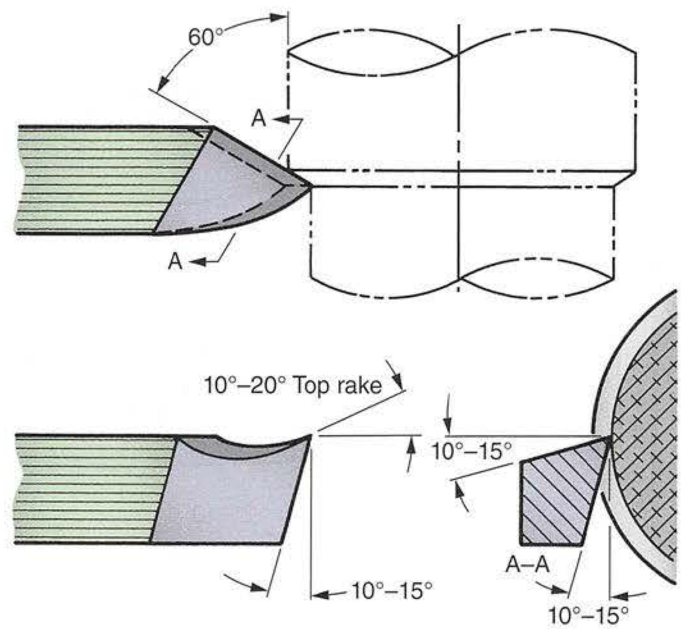
Aluminum is a very durable metal that retains its properties through repeated recycling. Recycling aluminum prevents more than 90 million tons of carbon dioxide emission each year and represents energy savings of more than 90% compared to producing new metal. You are probably most familiar with recycling aluminum in beverage cans, but aluminum is also a component in a wide variety of other alloyed materials. For example, various aluminum alloys are used in sheet metal fabrication, mainly due to their light weight and short machining time without the need for additional finishes. Manufacturers currently use about 35% recycled aluminum and 65% primary aluminum for machining or manufacturing purposes. As the high demand for aluminum continues to increase, local authorities and agencies are encouraging more recycling efforts to help recover more of the metal for our use.

28.3.2 Magnesium

Magnesium alloys are the lightest of the structural metals. They have a high strength-to-weight ratio. These alloys have excellent machining properties, and can be machined by all common metalworking techniques. The recommended lathe tool geometry for magnesium is shown in **Figure 28-17**.

Despite their many advantages, magnesium alloys must be worked with extreme care. Several of the alloys developed for use at elevated temperatures in aerospace vehicles contain thorium, a low-level radioactive material. They must be handled according to strict safety precautions for radioactive materials.

Another concern when machining magnesium is that extreme care must be taken because the chips or particles are highly flammable. (Because of the relatively high thermal conductivity of magnesium, there is normally no fire hazard when a solid section of the metal is exposed to fire, however.) Burning magnesium chips are intensely hot (5610°F or 3100°C). In addition, due to the reactive nature of the metal, magnesium fires cannot be extinguished by conventional firefighting techniques. Water or commercial extinguishing agents will actually intensify the fire. Use a Class D extinguishing agent to extinguish fires involving flammable metals. To guard against magnesium fires, do not allow magnesium chips to accumulate on or around the machine, and use a straight mineral oil cutting fluid in adequate quantities to flood the work.



Goodheart-Willcox Publisher

Figure 28-17. Configuration of HSS lathe tool recommended for machining magnesium.

SAFETY NOTE

When machining magnesium and its alloys, do not use water-based coolants. Water will react with the magnesium chips and will actually intensify a magnesium fire once it gets started.

28.3.3 Titanium

Titanium is a metal as strong as steel but only half as heavy. It bridges the gap between aluminum and steel. It is silvery in appearance, and extremely resistant to corrosion. Most titanium alloys are capable of continuous use at temperatures up to about 800°F (427°C). These characteristics make titanium ideal for use in high-speed aircraft components.

Titanium can be machined with conventional tools if the following practices are observed:

- Tool and work setups must be rigid.
- Tools must be kept sharp.
- Good coolants must be applied in adequate quantities.
- Cutting speeds must be slower, with heavier feeds than those used for steel.

Turning titanium that is commercially pure is very similar to turning 18-8 stainless steel. The alloys are somewhat more difficult to machine. Tungsten carbide and some types of ceramic tools produce the best results.

Milling titanium is more difficult than turning because the chips tend to weld to the cutter teeth. Climb milling may alleviate the problem to a great extent. Cast alloy tools often prove more economical to use than tungsten carbide tools. A water-based coolant is recommended.

Drilling titanium with conventional high-speed steel drills will produce satisfactory work. Drills should be no longer than necessary to produce the required hole depth and still allow the chips to flow unhampered.

Tapping titanium is one of the more difficult machining operations. The tap has a tendency to freeze or bind in the hole. Careful selection of cutting fluid will minimize this problem. Black-oxide-coated taps are recommended.

Sawing titanium requires a slow speed of about 50 fpm (15 mpm) with heavy, constant pressure. The tooth geometry of the blade must be designed for sawing titanium.

28.3.4 Copper

Copper can be shaped easily, but it becomes hard when worked and must be annealed or softened. It is difficult to machine because of its toughness and softness. Copper also has good thermal and electrical conductivity.

With copper, keep tools honed sharp and make as deep a cut as possible. Coolant should be used, because copper heats up quickly. Contaminated coolant can cause corrosion (discoloration of surfaces).

Brass and bronze are the most familiar of the copper-based alloys. However, lesser-known heat-treatable alloys are available. The newer alloys combine copper and exotic metals, such as zirconium and beryllium. Most copper-based alloys are available in rod, bar, tube, wire, strip, and sheet forms.

Brass

Brass is an alloy of copper and zinc. It ranges in color from reddish yellow to a silvery yellow, with the color determined by the percentage of zinc it contains. Most brasses can be readily machined.

Bronze

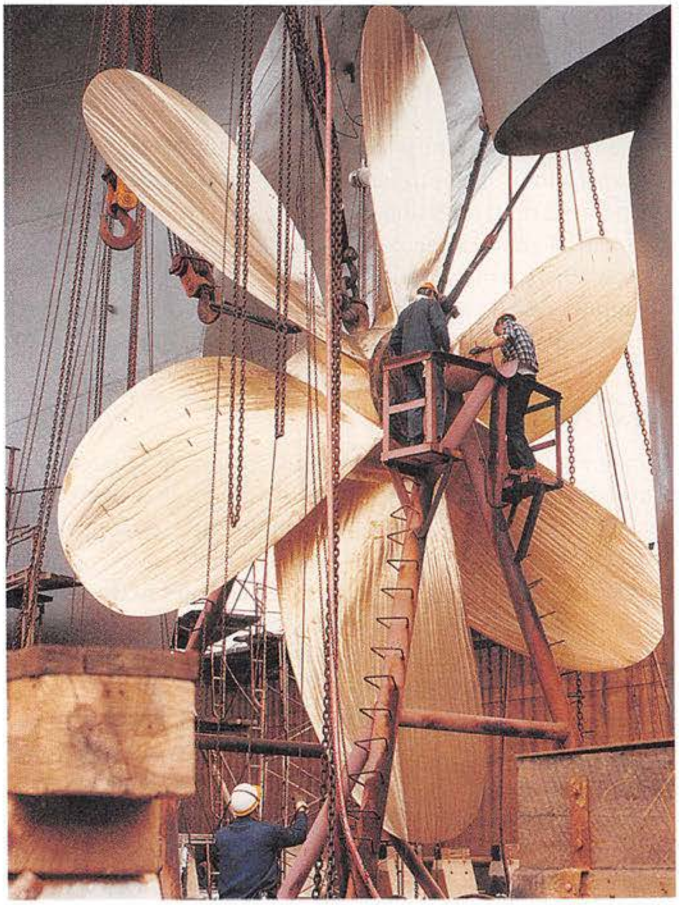
Bronze, **Figure 28-18**, is an alloy of copper and tin. It is harder than brass and is much more expensive. Many special bronze alloys include additional alloying elements, such as aluminum, nickel, silicon, and phosphorous. Most bronzes are relatively easy to machine with sharp tools. The recommended lathe tool geometry for brass and bronze is shown in **Figure 28-19**.

Beryllium Copper

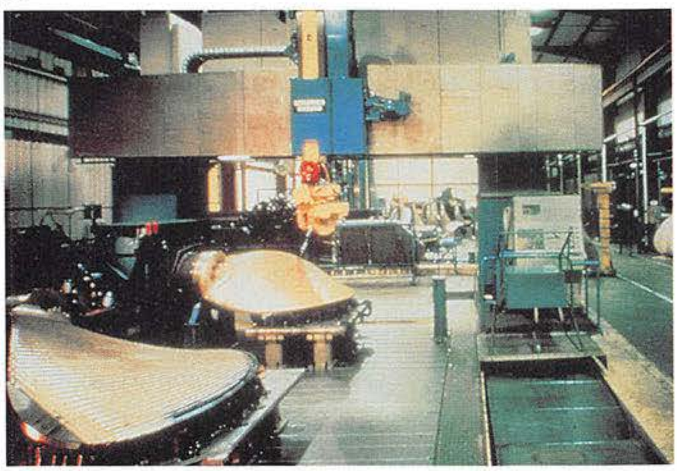
Beryllium copper is one of the newer copper-based alloys. Its machining qualities are similar to those of copper.

SAFETY NOTE

Machining beryllium copper can pose a definite health hazard if precautions are not observed. Beryllium is toxic. The fine dust generated by machining and filing beryllium copper can cause severe respiratory damage. A respirator-type face mask must be worn. Special procedures must also be followed when cleaning machines used to machine beryllium copper.



A Bethlehem Steel Co.

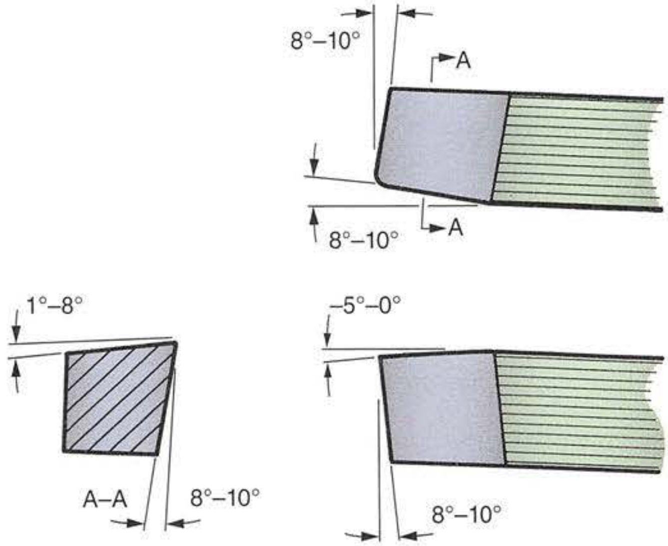


B Bird-Johnson Company

Figure 28-18. Bronze is an excellent metal for ship propellers. A—The propeller shown is 31 feet in diameter. B—Ship propellers are machined on large CNC multiaxis machine tools.

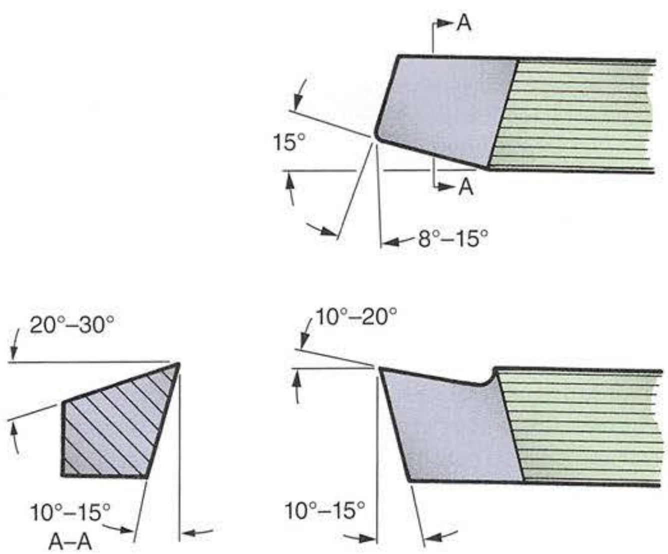
A vacuum system should be used to remove the beryllium copper dust, or the work should be liberally flooded with cutting fluid. Do not permit cutting tools to become dull, because dull tools generate more dust than sharp tools.

Beryllium copper can be heat-treated. It should not be machined in the annealed state. The recommended lathe tool geometry for beryllium copper is shown in **Figure 28-20**.



Goodheart-Willcox Publisher

Figure 28-19. Configuration of HSS lathe tool for turning brass and bronze.



Goodheart-Willcox Publisher

Figure 28-20. Configuration of HSS lathe tool for turning beryllium copper.

When machining beryllium copper, use a mineral oil-based cutting fluid. Select the fluid for its cooling properties, rather than for lubrication.

28.4 High-Temperature Metals

The nuclear and aerospace industries are chiefly responsible for the development of a number of high-temperature metals. These metals maintain their high strength during extended periods at elevated temperatures. They are sometimes called superalloys.

28.4.1 Nickel-Based Alloys

Examples of commercial nickel-based alloys are Iconel X, Hastelloy X, and Rene 41. They have many uses in jet engines, rocket engines, and electric heat-treating furnaces. These applications require metals that can operate at temperatures of 1200°F to 1900°F (649°C to 1038°C). Nickel-based alloys are not easy to machine by conventional methods.

28.4.2 Molybdenum

Molybdenum has excellent strength at elevated temperatures in the 1900°F to 2500°F (1038°C to 1372°C) range, and it has found many applications in modern technology. It also has great resistance to corrosion by acids, molten glass, and metals.

If both the work and cutting tool are mounted rigidly, molybdenum can be machined using methods similar to those for cast iron. For most work, tungsten carbide and ceramic tools are preferred over high-speed steel tools.

28.4.3 Tantalum

Tantalum alloys are specified where dependability at temperatures above 2000°F (1094°C) is required. Tantalum is used for rocket nozzles, for heat exchangers in nuclear reactors, and in some space structures.

Tantalum is not an easy metal to machine. It is gummy and has a tendency to tear. High-speed steel tools are usually recommended. Extreme cutting angles are used to keep the tool and chips clear of the work. The tool should be well supported, with little overhang.

28.4.4 Tungsten

Tungsten melts at 6200°F (3429°C), a higher temperature than any other metal. However, tungsten is not resistant to oxidation at high temperatures (above 930°F or 499°C) and must be protected with a suitable coating, such as one of the silicides. It has many uses in rocket engines, welding electrodes, and high-temperature furnaces. Tungsten is an ideal metal for breaker points in electrical devices.

Machining is quite difficult, but it can be done with carbide and ceramic tools if the work is preheated to about 400°F (204°C). The final shaping of a tungsten part is frequently done by grinding. Adequate cooling of the grinding wheel with an oil-based compound is recommended.

28.5 Rare Metals

Name just about any rare metal, and the odds are that someone is experimenting with its use in aerospace applications. Most metals in this category are available only in small quantities for experimental purposes. Many of them cost considerably more than gold.

Included in the rare metals group are elements such as scandium, yttrium, cerium, europium, lanthanum, and holmium. While they may seem strange and almost unknown at the present time, it was not too long ago that uranium, titanium, and beryllium were in the same category. In fact, until the 1880s, aluminum was considered a rare metal worth many times more than gold!

28.6 Other Materials

In addition to conventional metals and plastics, the modern machine shop is also expected to work other types of materials. These include honeycomb panels and various composites.

28.6.1 Honeycomb

Many ways have been devised to give existing metals greater strength and rigidity while reducing weight. ***Honeycomb sandwich panels*** are an example. Sections of thin material (aluminum, stainless steel, titanium, and nonmetals such as Nomex™ fabric) are bonded together to form a structure that is similar in appearance to the wax comb that bees create to store honey, **Figure 28-21**.

A strong structural panel is formed by rigidly bonding the honeycomb between two metal or composite sheets. Many advanced aerospace vehicles use large quantities of aluminum, stainless steel, and titanium honeycomb in their structures. The resulting material has a very high strength-to-weight ratio and rigidity-to-weight ratio. The bonding is done with an adhesive, or the materials are fused by brazing or resistance welding.

Because the honeycomb material is fragile before being shaped and bonded into rigid units, it can be difficult to

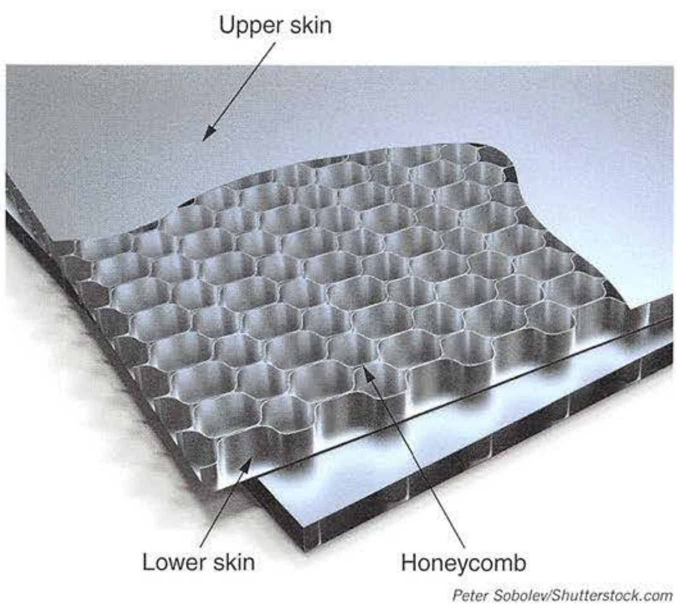


Figure 28-21. Honeycomb sandwich panels have great strength and rigidity for their weight. They have many applications in aerospace industries.

machine. In addition to special tools that literally pare the material off, electrolytic grinding is the most rapid method for machining honeycomb. It does not leave a burr that would create problems in its removal.

28.6.2 Composites

Composites are a relatively new class of material composed of fibers that are bonded together in a special plastic matrix (binding substance, such as an epoxy) under heat and pressure. The fibers in a composite can be made of many different materials, including various metals, graphite, boron, or fiberglass. Composite materials are generally lighter, stronger, and more rigid than many conventional metals. Some are capable of withstanding temperatures of 600°F (315°C) for long periods of time, and temperatures of 1000°F (540°C) for short periods of time.

Present uses for composites are concentrated in the aerospace and automotive industries, **Figure 28-22**. However, composites are used in many other products, including fishing poles, skis, golf clubs, tennis rackets, and bicycle frames, **Figure 28-23**. Much research is being done to reduce the cost of composites. As their cost comes down, the role of composite materials in manufacturing will continue to increase.

mirounga/Shutterstock.com

Figure 28-22. The Boeing 787 Dreamliner was the first commercial airliner to be built primarily from composite materials. By weight, the aircraft is composed of 50% composite materials, 20% aluminum, 15% titanium, 10% steel, and 5% other materials.

Compositek Corporation

Figure 28-23. One of the many types of composite materials is glass or carbon fiber bound together with a resin. Here, glass fiber is being wound around a large metal mandrel before resin is applied. Once the resin cures (hardens), grinding and other machining techniques can be used to achieve final dimensions. Note the dust mask being worn by the technician to prevent inhaling tiny airborne fibers.

Chapter Review

Summary

- Ferrous metals contain iron; nonferrous metals do not. An alloy is a mixture of two or more metals. A base metal is a pure, nonferrous, non-precious metal.
- Ferrous metals include cast irons (2%–5% carbon), low-carbon steels (< 0.3% carbon), medium-carbon steels (0.3%–0.6% carbon), high-carbon steels (0.6%–1.5% carbon), and various alloy steels. In alloy steel, various metallic elements are mixed with the steel to give it specific properties.
- The term *tool steel* refers to steels used to make tools that cut, shear, or form materials. Tool steel can be carbon steel or alloy steel. Tool steels are heat-treated to give them the hardness or toughness they need to perform well. High-speed steel (HSS) is a type of tool steel that has high red hardness and high abrasion resistance.
- Tungsten carbide is a nonferrous metal that is commonly grouped with ferrous metals. It is the hardest known metal and is commonly used to make cutting tools.
- Stainless steels are steels that contain enough chromium to make them corrosion resistant.
- Steels are identified by their AISI/SAE codes, by color coding, or by a spark test.
- Aluminum is a soft, light, nonferrous metal. It can be strengthened by adding alloying elements. Most aluminum alloys have high corrosion resistance.
- Magnesium is a nonferrous metal with a high strength-to-weight ratio and excellent machining properties. Because some magnesium alloys contain thorium and because magnesium chips and dust are highly flammable, strict safety precautions must be observed when working with magnesium.
- Titanium is a nonferrous metal that is strong, lightweight, and corrosion resistant. Titanium's light weight and ability to withstand high temperatures without losing strength makes it ideally suited for aerospace applications.
- Copper is a soft, tough, nonferrous metal. It is easily shaped but becomes hard when worked. The two most familiar copper-based alloys are bronze and brass.
- High-temperature metals are those capable of maintaining high strength under extended exposure to high temperatures. High-temperature metals include nickel-based alloys, molybdenum, tantalum, and tungsten.
- In addition to the traditional metals, rare metals (such as scandium, yttrium, cerium, europium, lanthanum, and holmium), honeycomb panels, and composite materials are seeing increased use in manufacturing.

Review Questions

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

1. What are the four main classifications of modern industrial metals?
2. Iron and steel are classified as ______ metals.
3. Carbide cutting tools are recommended for machining cast iron because ______.
   A. cast iron is soft
   B. cast iron has an abrasive surface scale
   C. high-speed steel is incompatible with the recommended cutting fluid
   D. All of the above.
4. Carbon steel is an alloy of ______ and carbon.
5. How are carbon steels classified?
6. Hot-rolled steel is characterized by the ______ on its surface.
7. The machinability of carbon steel is improved if sulfur or ______ is added as an alloying element.
8. Drills, reamers, and some milling cutters are usually made from ______ steel.
9. The primary alloying element in stainless steel is ______.
10. The chief characteristic of stainless steel is its resistance to ______.
11. List the three basic groups of stainless steel.
12. Aluminum, magnesium, and titanium are ______ metals.
13. Magnesium alloys are the ______ of the structural metals.
14. ______ is a metal that is as strong as steel but only half as heavy.
15. Brass is an alloy of copper and ______.
16. Bronze is an alloy of copper and ______.
17. Why must a machinist take special precautions when working beryllium copper?
18. Nickel-based alloys, molybdenum, tantalum, and tungsten are classified as ______ metals.
19. Name four metals that are currently included in the rare metals group.
20. What is the structural material known as honeycomb?
21. What are composites?

CHAPTER 29

Heat Treatment of Metals

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- List some of the metals that can be heat-treated.
- Describe various heat treatment techniques for steel and other ferrous metals.
- Explain heat treatment techniques for aluminum, brass, copper, and titanium.
- List considerations for quenching media and furnaces used in heat treatment processes.
- Summarize the process for hardening and tempering carbon steel.
- Compare hardness testing techniques.
- List the safety precautions that must be observed when heat-treating metals.

Technical Terms

annealing
box annealing
Brinell hardness test
case hardening
critical temperature
hardening
heat treatment
normalizing
process annealing
quench
Rockwell hardness test
scleroscope
stress-relieving treatment
surface hardening
tempering
Webster hardness tester

Since many parts produced in the machine shop must be heat-treated before use, it is important that the machinist be familiar with the basic science of heat-treating metals. ***Heat treatment*** is the controlled heating and cooling of a metal or alloy to obtain certain desirable changes in its physical characteristics. These changes include improving resistance to shock, developing toughness, and increasing wear resistance and hardness, **Figure 29-1**. Similar techniques can be used to anneal (soften) metals to make them easier to machine, or to case harden (produce a hard exterior surface) steel for better resistance to wear.

In a heat treatment process, the metal is heated to a predetermined temperature and then ***quenched*** (cooled rapidly) in water, brine, oil, blasts of cold air, or liquid nitrogen. The desired qualities are not always achieved immediately after quenching. Stresses can develop that, under certain conditions, may cause some steels to shatter. Therefore, the metal may need to be reheated to a lower temperature, followed by another cooling cycle to develop the proper degree of hardness and toughness.

29.1 Heat-Treatable Metals

Steel and most of its alloys can be hardened. Magnesium, copper, beryllium, titanium, and many aluminum alloys are also capable of being heat-treated. However, when carbon steel is heat-treated, the carbon content of the metal is an important consideration. Carbon steels are classified by the percentage of carbon they contain in "points" or hundredths of a percent. For example, 60 point carbon steel contains 0.60% carbon. Steel with less than 50 points of carbon cannot be hardened.

MARCELODLT/Shutterstock.com

Figure 29-1. Many parts of this huge ore-carrying truck are heat-treated. Without heat-treated parts (including wheels, drive shaft, gears, and axles), the vehicle would not be able to maintain its grueling workload for long without part wear and failure.

29.2 Heat Treatment of Steel

The heat treatment of metals can be divided into two major categories. One deals with ferrous metals, and the other concerns nonferrous metals. Heat treatment of both ferrous and nonferrous metals can be further divided into different processes. The following sections explain basic heat treatment processes and how they affect steel. Details regarding the heat treatment of nonferrous metals will be presented later in the chapter.

29.2.1 Stress-Relieving Treatment

Stress-relieving treatments are done to remove internal stresses that have developed in parts that have been cold-worked, machined, or welded. Stress in steel parts is relieved by heating the parts to 1000°F to 1200°F (538°C to 649°C). The parts are held at this temperature for 1 hour or more per inch of thickness, and then the parts are slowly air- or furnace-cooled. The technique is sometimes called ***process annealing***.

29.2.2 Annealing

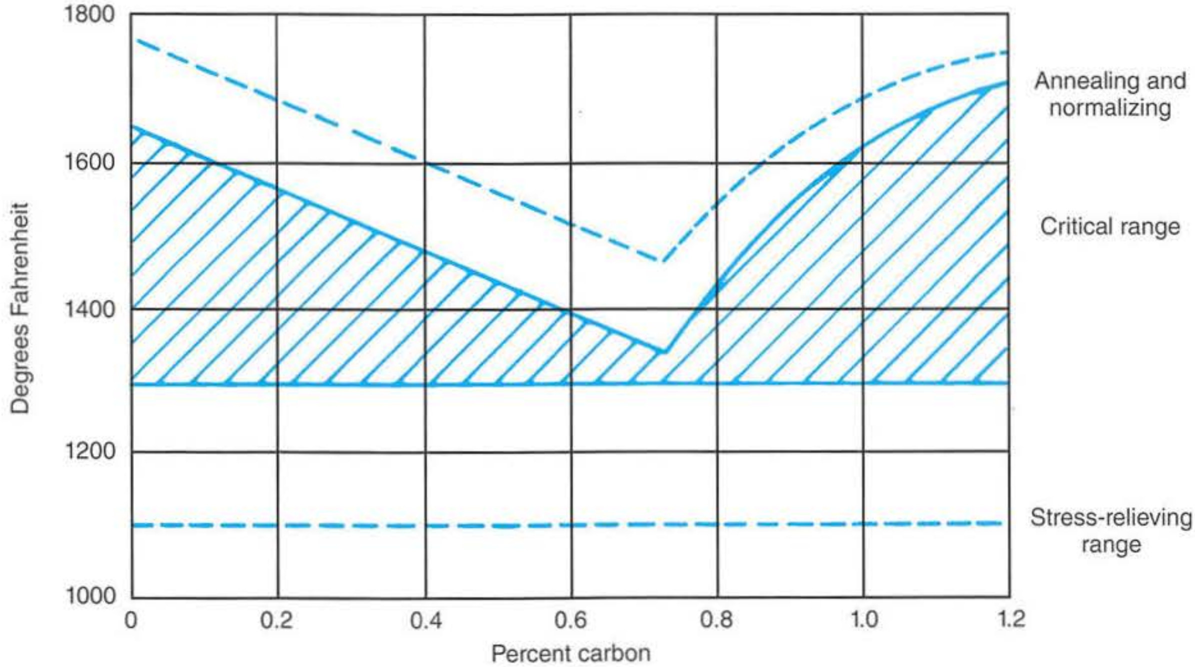
Annealing is a process that reduces the hardness of a metal to make it easier to machine or work. It involves heating the metal to slightly above its ***critical temperature*** (temperature at which the metal's crystal structure changes), but never more than 50°F to 75°F (10°C to 24°C) above this point, **Figure 29-2**. The time it is held at this temperature depends on the shape and thickness of the part. After the holding period, the piece is allowed to cool slowly in the furnace or other insulated enclosure.

For some steels, it may be necessary to use the box annealing method or a controlled-atmosphere furnace to prevent the work from scaling or decarbonizing (losing carbon at the surface). When annealing is done in a controlled atmosphere, oxidation does not take place and the part remains bright. In the ***box annealing*** process, the part is placed in a metal box and the entire unit is heated and then allowed to cool slowly in the sealed furnace.

When the hardness of the metal is reduced, its machinability is improved. Many nonferrous metals can also be softened by annealing.

29.2.3 Normalizing

Normalizing is a process that refines the grain structure of some metals and thereby improves their machinability. In the normalizing process, the metal is heated to slightly above its upper critical temperature and allowed to cool to room temperature. It is a process closely related to annealing.



Goodheart-Willcox Publisher

Figure 29-2. Critical range diagram for plain carbon steel.

29.2.4 Hardening

Hardening is a heat treatment that reduces the ductility of steel. This is accomplished by heating the metal to a predetermined temperature for a specified period of time. The temperature at which steel hardens is called its *critical temperature.* The critical temperature of steel ranges from 1400°F to 2400°F (760°C to 1316°C), depending on the alloy and carbon content. For a rule-of-thumb range of hardening temperatures for carbon steel, see **Figure 29-3.**

After being heated, the part is quenched in water, brine, oil, liquid nitrogen, or blasts of cold air. Water or brine is used to quench plain carbon steel. Oil is usually used to quench alloy steels. Blasts of cold air or liquid nitrogen are used for high-alloy steels.

Quenching leaves the steel hard and brittle. It may fracture if exposed to sudden changes in temperature. For most purposes, this brittleness and hardness must be reduced by a tempering or drawing operation.

29.2.5 Surface Hardening

Surface hardening is often used when only a medium-hard surface is required on high-carbon or alloy steels, **Figure 29-4.** The internal structure of the metal is not affected. Flame hardening, induction hardening, and laser hardening are used to achieve this.

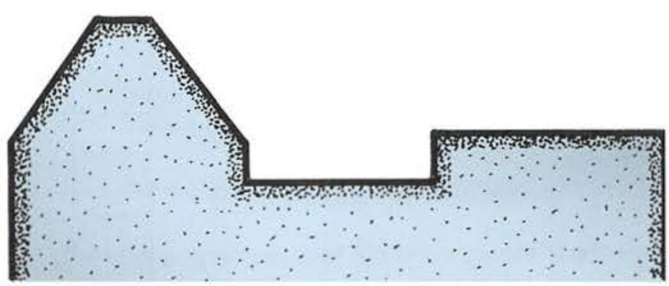
Flame hardening is the rapid heating of the surface with an acetylene torch, followed by immediate quenching of the heated surface. The flame must be moved constantly to prevent burning or hardening the metal too deeply.

Induction hardening is the use of a high-frequency electrical induction current to heat the metal, **Figure 29-5.** The quenching medium follows the induction coil. The technique

Carbon Content	Hardening Temperature Range
0.65% to 0.80%	1450°F to 1550°F (788°C to 843°C)
0.80% to 0.95%	1410°F to 1460°F (766°C to 793°C)
0.95% to 1.10%	1400°F to 1450°F (760°C to 788°C)
Over 1.10%	1380°F to 1430°F (749°C to 777°C)

Goodheart-Willcox Publisher

Figure 29-3. This chart lists approximate ranges of hardening temperatures for carbon steel. The heated metal must be quenched using water, brine, light oil, or blasts of cold air.

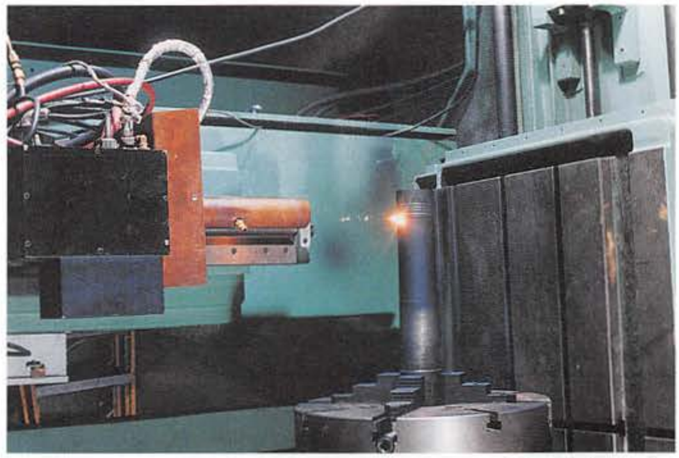


Goodheart-Willcox Publisher

Figure 29-4. Lathe ways are frequently surface-hardened for improved wear resistance. The denser dot pattern at the edges of this lathe way cross section shows that only the outer surface is hardened.

Radyne

Figure 29-5. A stand-alone vertical-scanning induction system for heat treatment. It is programmable and equipped with a touchscreen. The furnace can be loaded and unloaded manually or by a robotic manipulator.



Light Beam Technology, Inc.

Figure 29-6. A laser beam being used to heat-treat the bearing area of a shaft. Black paint on the bearing area prevents the beam from being reflected back from the surface without heating the metal. The small flame rising from the spot being treated is caused by the black paint burning off.

is quick and tends to minimize distortion in the piece being heat-treated. Induction hardening is ideal for production hardening operations.

Laser hardening relies on lasers to heat the metal being hardened. A laser beam 1/8″ to 5/8″ (3.2 mm to 15.9 mm) wide is focused on the area to be hardened, **Figure 29-6**. The light energy emitted by the laser is converted into heat energy and is absorbed by the metal.

The surface heats rapidly, so the part must be moved under the beam or the area will be heated to its melting temperature. Because the laser is focused on a small area and the part is continuously moved, very little heat is generated in the part outside of the target area. As a result, the hardened area cools rapidly (self-quenches) enough to touch within a few seconds.

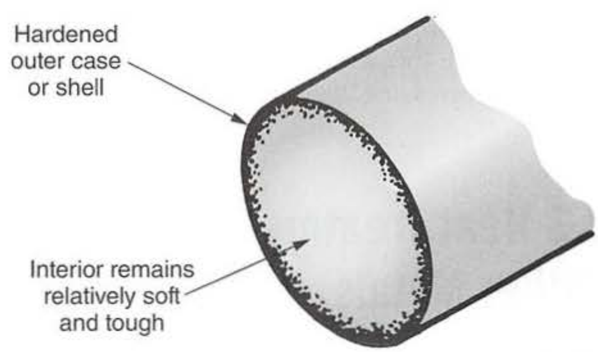
With the laser technique, the area being hardened can be carefully controlled. It may be as small as 1/4″ (6.5 mm) square. There is very little chance of part warping or distortion with laser hardening. The process produces a fine-grain structure that has a tougher wearing surface than other techniques.

29.2.6 Case Hardening

Low-carbon steel cannot be hardened to any great degree by conventional heat treatment. However, a hard shell can be put on the surface, while the inner portion remains relatively soft and tough. The process used to do this is called ***case hardening***, **Figure 29-7**.

Case hardening is accomplished by heating the piece to a red heat and introducing small quantities of carbon or nitrogen to the metal's surface. This can be done by carburizing, cyaniding, or nitriding.

During carburizing, sometimes called the *pack method*, the steel is buried in a dry, carbonaceous material (material rich in carbon) and heated to just above its transformation range, **Figure 29-8**. In the transformation range (1350°F–1650°F or 732°C–900°C), steels undergo internal structural changes that radically affect their properties. The part is held at this temperature for 15 minutes to 1 hour, until the desired case thickness is attained. The part is then removed from the furnace and quenched. Deep (very thick) cases can be obtained by this method.



Goodheart-Willcox Publisher

Figure 29-7. Cross section of a case-hardened part shows that the interior remains relatively soft and tough, while a hard "shell" is formed on the exterior.

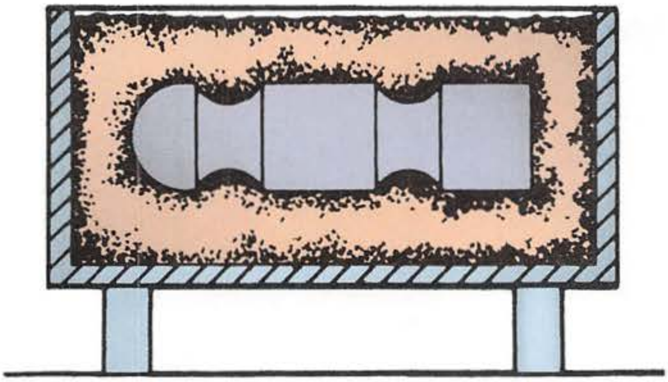


Figure 29-8. A part packed in a container of carbonaceous material, ready to be case hardened.

During the liquid salt method of case hardening, also known as *cyaniding*, the part is heated in a molten cyanide salt bath, then quenched. The immersion period is usually less than 1 hour. A high hardness is imparted to the work, and the treated parts have good wear resistance.

The *nitriding* method of case hardening develops high hardness without quenching, and distortion is virtually nonexistent. In the nitriding method, or gas method, of case hardening, parts are placed in an airtight heating chamber where ammonia gas is introduced at high temperature. The ammonia decomposes into nitrogen and hydrogen. The nitrogen enters the steel to form nitrides, giving the metal's surface an extreme hardness. Wear-resistance and high-temperature hardness are greatly increased.

29.2.7 Tempering

Tempering is a heat treatment process that reduces a metal's brittleness or hardness. In the tempering process, steel is heated to a temperature below its critical range. Refer to **Figure 29-2**. The exact temperature depends on the type of steel being treated and its intended application. This information can be found in steelmakers' catalogs and the various machinists' handbooks. The temperature is held until complete penetration is achieved, and then the steel is quenched.

With the internal stresses released, the toughness and impact resistance increase. As the temperature is raised, ductility is improved, but there is a decrease in hardness and strength.

29.3 Heat Treatment of Other Metals

In addition to steel, many other metals and their alloys are heat-treatable. You should have a basic understanding of these processes.

29.3.1 Heat Treatment of Aluminum

Aluminum is a general term applied both to the base metal and to its many alloys. When heat-treating aluminum, it is imperative that the exact alloy be known or the part being heat-treated may be ruined.

Aluminum alloys are heat-treatable in much the same manner as steel. That is, the metal is heated to a predetermined temperature, quenched, and then reheated to a lower temperature. However, some aluminum alloys age-harden at room temperature. These alloys must be kept refrigerated to remain soft and ductile (property of metal which permits it to be drawn out or hammered thin) while they are being worked.

Heat treatment temperatures for several of the more common aluminum alloys are shown in **Figure 29-9**. Information on other alloys, each of which requires special treatment to bring out optimum physical qualities, can be obtained from handbooks available from the various producers of aluminum.

29.3.2 Heat Treatment of Brass

Brass can be annealed after cold-working by heating it to 1100°F (593°C) and cooling. The rate of cooling has no appreciable effect on the metal.

29.3.3 Heat Treatment of Copper

Copper is annealed in much the same manner as brass. The metal may be quenched or allowed to cool slowly at room temperature.

29.3.4 Heat Treatment of Titanium

Most titanium alloys are heat-treatable. However, special facilities are necessary because titanium is a reactive metal: it readily absorbs oxygen, carbon, and nitrogen. These elements greatly affect the strength of titanium and its resistance to fatigue and corrosion.

29.4 Equipment for Heat Treatment

Heat treatment involves three distinct steps: high temperature heating, quenching (rapid cooling) to harden, and tempering for final hardness and physical properties. Each step has a significant effect on the final result.

29.4.1 Quenching Media

The main problem in heat treatment is to cool the metal at a uniform rate over its entire area. Water, oil, air, and liquid nitrogen are standard quenching media used to draw heat from the part being treated. Quench tanks for oil and

Heat-Treatable Aluminum Alloys						
Solution				**Precipitation (Aging)**		
Aluminum alloy	**Heat to F (C)**	**Quench**	**Resulting temper**	**Heat to F (C)**	**Hold for hours**	**Resulting temper**
2024-0	910°F–930°F (488°F–499°C)	In cold water as quickly as possible after removal from furnace	—	Room	48–96	2024–T4
6061-0	960°F–980°F (516°C–527°C)		6061-W	315°F–325°F (157°C–163°C)	16–20	6061–T6
				345°F–355°F (174°C–179°C)	6–10	
7075-0	860°F–930°F (460°C–499°C)		7075-W	245°F–255°F (118°C–124°C)	20–26	7075–T6

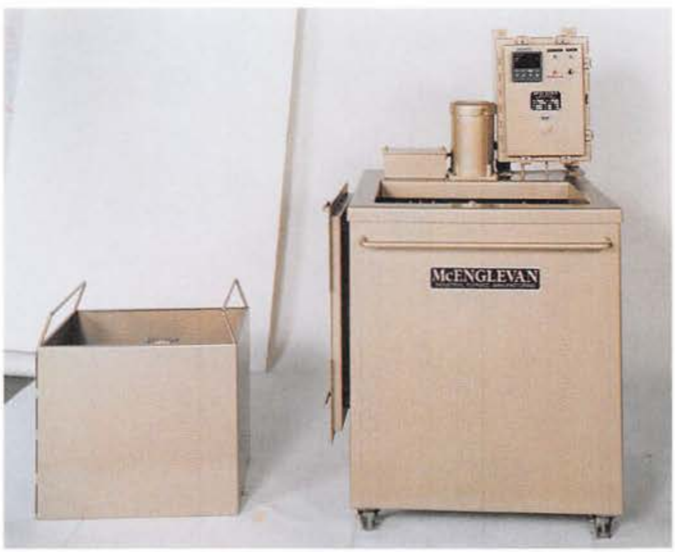
Note: When heat-treating clad 2024 and clad 7075 aluminum, hold temperature for shortest possible time.

Goodheart-Willcox Publisher

Figure 29-9. Heat treatment temperatures for several common aluminum alloys.

water are available with temperature controlling systems, **Figure 29-10**.

Water cools metal very quickly. It is used mainly when plain carbon steel is being treated for maximum hardness. Water has the disadvantage of forming gases when the hot metal is immersed in it. The gas bubbles cling to the metal, slow the cooling process, and cause soft spots in the treated piece. The use of brine (5% to 10% salt in the quench water) prevents the formation of gases and gives better cooling results.



MIFCO McEnglevan Industrial Furnace Co., Inc.

Figure 29-10. A circulating oil-quench system. The shop rule for a circulating quench tank is one gallon of quench medium for each pound of steel at 1500°F (816°C) quenched per hour. Standard quench oils give best results when heated to 120°F to 140°F (49°C to 60°C).

Mineral oils cool more slowly and produce less distortion in the treated part than water. Special quenching oils have been developed. They have a high flash point (the lowest temperature at which the oil vapors will ignite in air) and do not have a disagreeable odor. The mineral oil quenching baths used for production must be filtered and cooled down to room temperature. Oils are used to harden alloy steels.

SAFETY NOTE

Quenching heated metal in oil should be done only in a well-ventilated area. Avoid inhaling any of the fumes.

Freely circulated air is used to cool some highly alloyed steels. The air used as a cooling medium must be dry. Any moisture in the air could cause the steel to fracture.

Liquid nitrogen is a cryogenic medium that is used to quench several aluminum alloys and space-age metals. Because liquid nitrogen has a temperature of about –320°F (–196°C), its use as a quenching medium requires special facilities.

29.4.2 Furnaces

Industry uses many types of heat treatment furnaces, including furnaces heated by electricity, gas, or oil. They must be capable of reaching and maintaining the temperatures needed for heat treatment. Most are automated and continuous in operation.

Modern electric furnaces are fitted with numerous safety devices. Avoid using furnaces until you are thoroughly versed in their safe operation. Gas-fired furnaces are also widely used for heat treatment. For safe operation of a gas-fired furnace, closely follow the manufacturer's operating instructions.

SAFETY NOTE

Gas- and oil-fired furnaces are noisy. Hearing protectors must be worn when working near them.

Most small furnaces are heated by electricity. They are safe to operate, quiet, require no elaborate venting systems, reach temperature quickly, and can be controlled with accuracy. If the furnace is equipped with a microprocessor-based controller, it is possible to program the furnace for precise time/temperature cycles, **Figure 29-11**.

Some models are equipped with two chambers, **Figure 29-12**. The upper high-temperature chamber can provide atmospheric control for heat-treating with inert gases. The lower chamber is used for tempering and drawing.

To provide atmospheric control, the furnace is sealed and a vacuum is drawn to remove atmospheric gases that might contaminate the metal being heat-treated. The chamber is then flooded with an inert gas (one that will not oxidize or be absorbed by the metal's surface) during the heat treatment operation.

Heat-treating any metal requires maintaining accurate temperatures. A pyrometer is an instrument that accurately measures furnace temperature. In some furnaces, the pyrometer is integrated into a thermostat that can be set to the desired temperature and used to maintain that temperature once it is reached.

If the furnace is not equipped with a pyrometer, it is necessary to judge the temperature by the color of the metal as it heats. Color charts are available from the metal suppliers. See **Figure 29-13**.

29.5 Hardening Carbon Steel

The following procedure is recommended for hardening carbon steels. Oxidation of the metal during heat treatment can be avoided by wrapping the part in stainless steel foil, **Figure 29-14**.

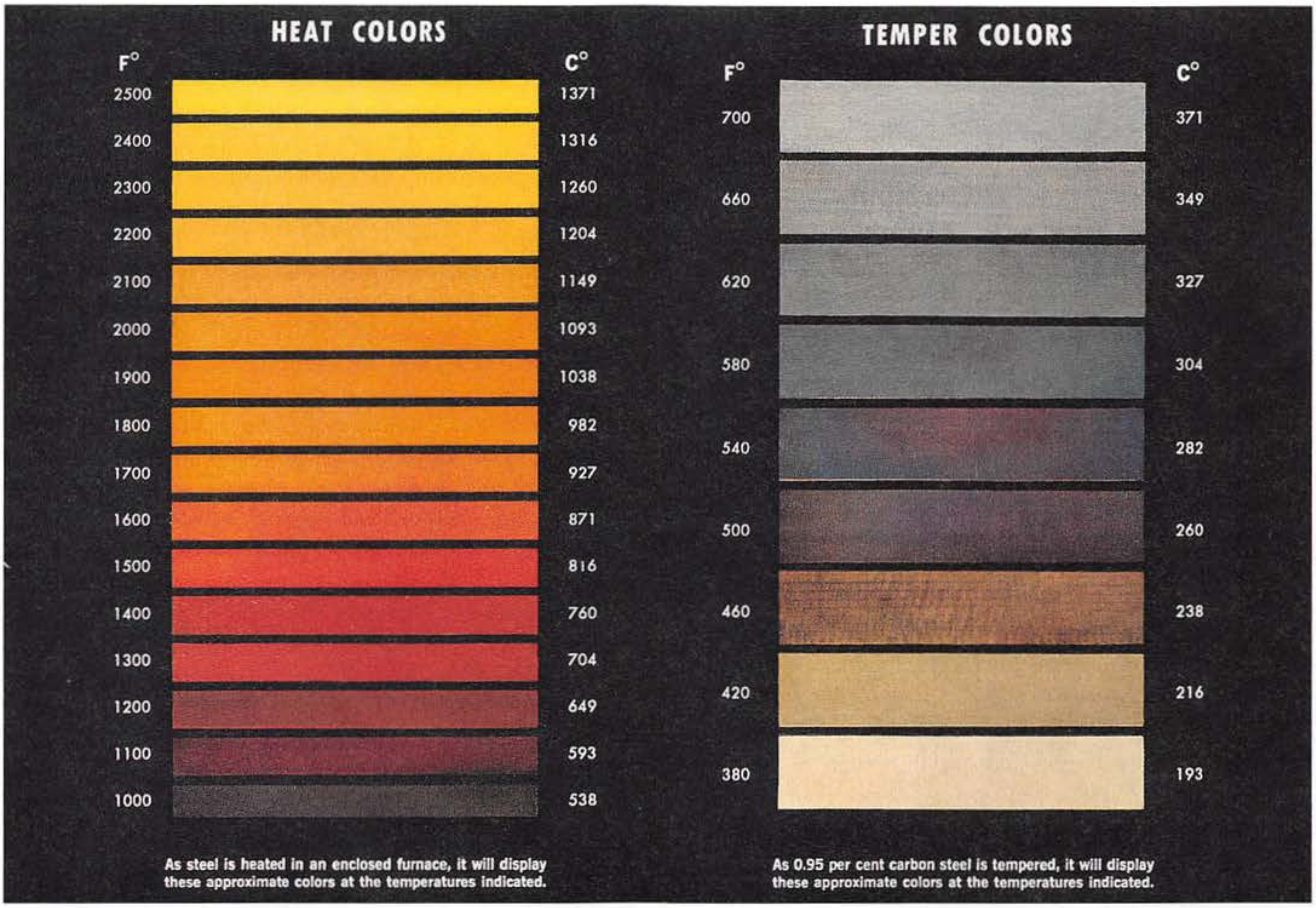
1. Place the metal in the furnace and set the controls (if applicable) for the desired temperature.
2. Heat the metal to its critical temperature (1300°F to 1600°F, or 704°C to 871°C). Avoid placing the part being heat-treated directly in the gas flames. If it has not been wrapped in stainless steel foil, position it in a piece of iron pipe, as shown in **Figure 29-15**. Allow the part to "soak" in the furnace until it is heated evenly throughout.

NEYTECH

Figure 29-11. Benchtop muffle furnace with a programmable controller for precise time/temperature management. This electric furnace has a temperature range of 90°F to 2012°F (32°C to 1100°C).

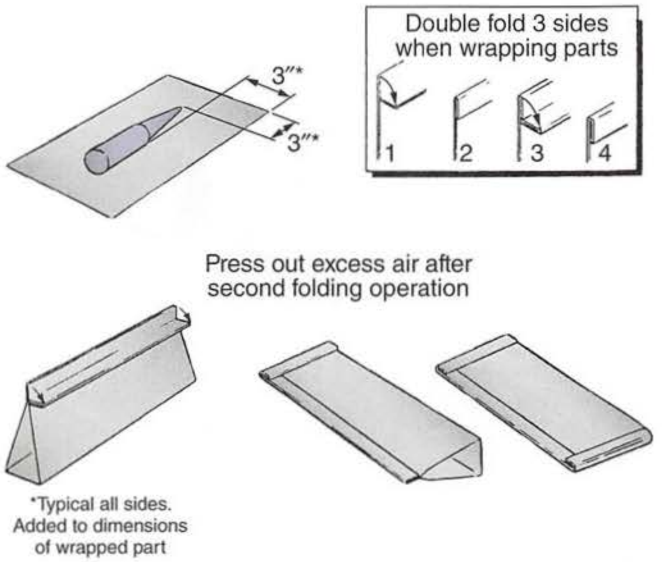
MIFCO McEnglevan Industrial Furnace Co., Inc.

Figure 29-12. Dual electric furnace with digital readout and microprocessor-based controller. The top unit features atmospheric control for hardening with inert gases; the lower unit is used for drawing and tempering.



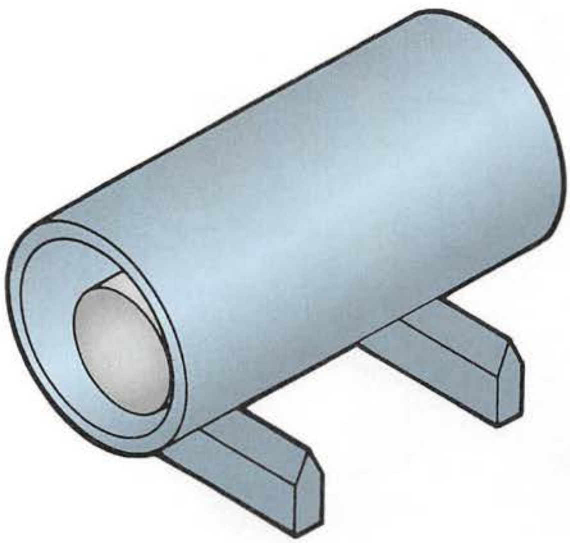
Bethlehem Steel Co.

Figure 29-13. As steel is heated, it changes to the colors shown in the left column of this chart. The colors in the right column are useful for tempering steel if no pyrometer is fitted to the furnace.



Goodheart-Willcox Publisher

Figure 29-14. Procedure for wrapping parts in stainless steel foil to protect the parts against oxidation and decarburization during heat treatment.



Goodheart-Willcox Publisher

Figure 29-15. Work being heat-treated must be protected from direct flames in a gas furnace by inserting it in a section of pipe. Keep the pipe raised off the furnace floor to permit uniform heating.

3. Preheat the tong jaws and then remove the piece from the furnace.

SAFETY NOTE

Dress properly for any operation involving the furnace, **Figure 29-16**. The metal is very hot, and serious burns can result from relatively minor accidents.

4. Quench the piece in water, brine, or oil (depending on the type of steel being treated). Except for some alloy steels, steels are usually classified as water-hardening or oil-hardening types, based on the quenching medium to be used on them. The quenching technique is critical. To ensure an even hardness throughout the piece, dip long, slender sections straight down into the quenching fluid with an up-and-down motion. Avoid a circular motion, because this may warp the piece. Parts with other shapes should be moved around in a manner that will permit them to cool quickly and evenly.

Steel that has been hardened properly is "glass-hard" and too brittle for most purposes. Hardness can be checked by trying to file the work surface. A file will not cut the surface if the piece has been hardened properly. Do not use a new file for testing hardness—it will be damaged!

29.6 Tempering Carbon Steel

As mentioned earlier, tempering relieves the stresses that develop in the metal during hardening. Until it is tempered, the brittle hardened steel may crack or shatter from shock (such as being dropped or struck) or from sudden changes in temperature.

Tempering is done as follows:

1. Polish the hardened piece with abrasive cloth.
2. Heat the piece to the correct tempering temperature. The correct temperature is based on the type of steel and the job the finished part is to do. Proper tempering temperatures range from about 380°F or 193°C (the metal will turn a silvery yellow or straw color) to 700°F or 371°C (the metal will turn a light gray or blue color). Color charts are available from steel companies and can be used as a guide for determining when the desired temperature is reached. Refer to **Figure 29-13**. Quench the metal as soon as it reaches the required temperature.
3. Small tools are best tempered by placing them on a steel plate that has been heated red-hot. Have the point of the tool extend beyond the edge of the plate, as shown in **Figure 29-17**. Watch the temper color as the metal heats up. Quench the metal as soon as the tool point turns the proper color.

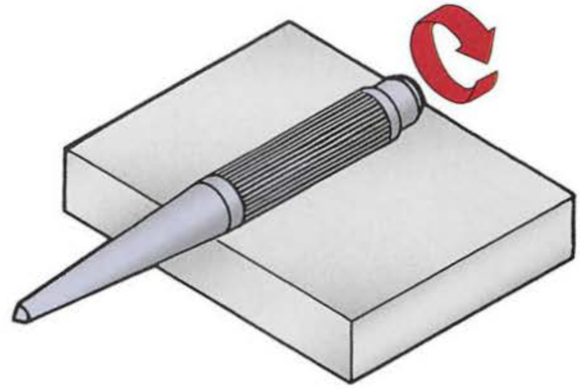
Hot liquid baths of oil, molten salts, or lead are often used instead of a furnace to heat parts to their proper tempering temperature. The pieces are held in the bath until the heat permeates them. They are then removed from the bath and allowed to cool in still air.

29.7 Case Hardening Low-Carbon Steel

Of the several case hardening techniques, the simplest is carburizing, which requires a minimum of equipment. It uses a nonpoisonous commercial case hardening compound.

MIFCO McEnglevan Industrial Furnace Co., Inc.

Figure 29-16. Heat treatment temperatures are very hot. Dress properly for job and keep the area around furnace clean so there is no danger of slipping or stumbling. Also, preheat tongs before grasping the heated part.



Goodheart-Willcox Publisher

Figure 29-17. Use a heated steel plate when tempering small tools. Extend the point of the tool beyond edge of the plate. Rotate the tool as it heats up to ensure that it is heated evenly throughout.

SAFETY NOTE

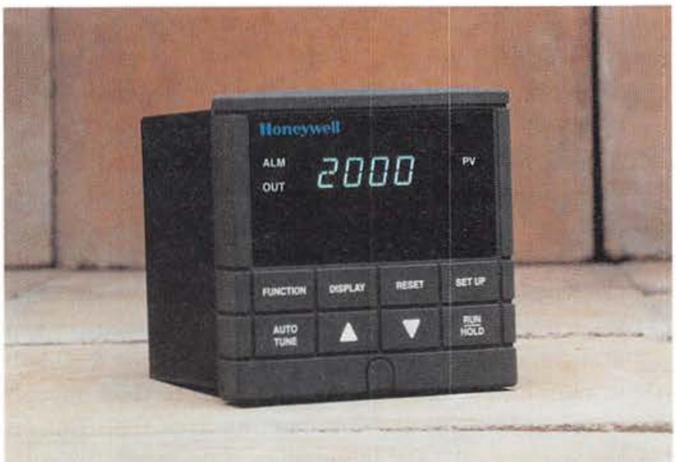
The case hardening technique known as *cyaniding* is not recommended. Cyanide is a deadly poison and is very dangerous to use under any but ideal conditions.

Two methods are recommended for using case hardening compound to harden low-carbon steel. The first method is as follows:

1. Bring the furnace to temperature.
2. Bring the workpiece to a bright red (1650°F–1700°F, or 899°C–927°C). Use a pyrometer or thermocouple to monitor the temperature, **Figure 29-18**.
3. Dip, roll, or sprinkle the case hardening compound on the piece, **Figure 29-19**. The powder melts and adheres to the surface, forming a shell.
4. Reheat the workpiece to a bright red and hold at that temperature for a few minutes.
5. Quench in cold water.

The second method for case hardening low-carbon steel is to:

1. Find a steel container large enough to hold the work.
2. Put the part to be hardened into the container and completely cover it with case hardening compound. Refer to **Figure 29-8**.
3. Place the container in the furnace and heat it to a red heat. Hold the temperature for 5 to 30 minutes, depending on the depth of case required.
4. Use dry tongs to remove the piece from the container and then quench the piece in clean, cool water.

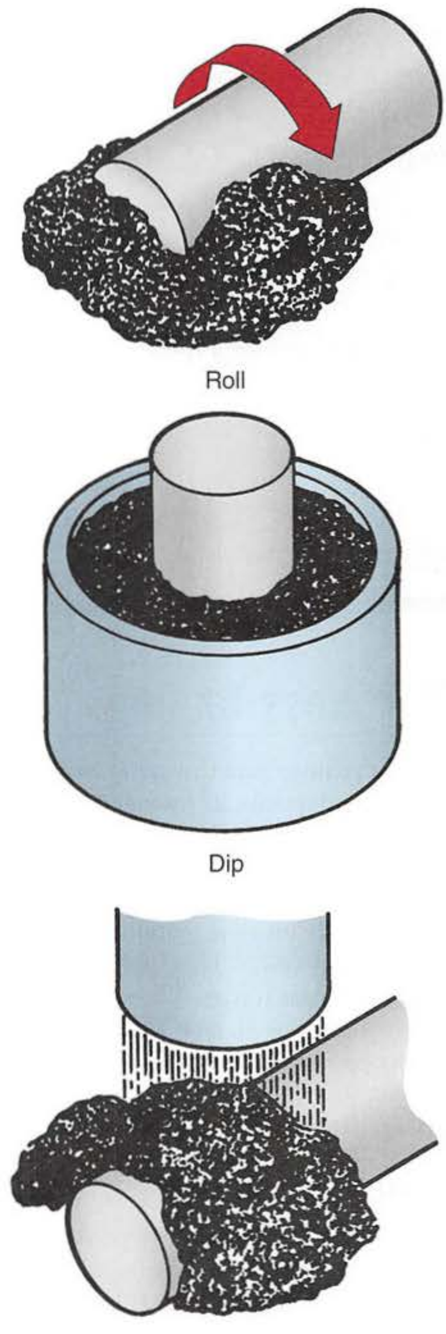


Honeywell, Inc., Industrial Automation and Control

Figure 29-18. A microprocessor-based digital controller for electric heat treatment furnaces. It can accept ten different thermocouple types.

SAFETY NOTE

With either method of using case hardening compound, work in a well-ventilated area and wear full face protection, leather apron, and heat-resistant gloves.



Goodheart-Willcox Publisher

Figure 29-19. Dip, roll, or sprinkle case hardening compound on the work until a shell of compound has formed.

CAREER CONNECTION

Foundry Worker

What does a foundry worker do?

A foundry is a metalworking facility that produces metal parts from molds. To create these parts, metals are melted down and poured into molds. Foundry workers may be responsible for creating molds, setting molds, and pouring material into molds. Depending on the product's design, molten metal may be poured into the mold manually or by remote automation. While a foundry works primarily with metals, similar facilities exist for melting and molding plastics.

What education and skills are needed to be a foundry worker?

Employers prefer candidates with high school diplomas and applicable technical education in computer programming or technology. Strong math skills and general knowledge of metals and plastics are also useful. Further education and training are available through technical courses at community colleges and certificate programs in metal/plastics machinery operation.

What is it like to be a foundry worker?

Foundry workers are part of a wider profession of metal and plastics machinery workers. These workers are employed primarily by manufacturing industries. Like other manufacturing professions, foundry work can be physically demanding and hazardous. Risk of illness and injury is high if proper safety measures are not observed.

The median annual wages for these professions is $34,000, as reported by the *Occupational Outlook Handbook*. Foundry workers, pourers, and casters fall near the group average at $33,800 per year.

29.8 Hardness Testing

Hardness testing ensures that the metal has been given the proper degree of hardness for its intended use. It is also possible to establish standards for hardness that can be cited on drawings and specifications.

The most commonly used technique is to press a steel ball or diamond probe into the metal under a specific load to see how deep it penetrates. This type of test is performed using indention hardness testers. The most common testers include the Brinell, Rockwell, and Webster hardness testers and the scleroscope. The testing machines display the results as a hardness number, which indicates the degree of the metal's hardness. See **Figure 29-20**.

29.8.1 Brinell Hardness Test

The ***Brinell hardness test*** is used extensively in both laboratory and production situations to measure a metal's resistance to deformation, **Figure 29-21**. It is an excellent index of factors such as machinability, uniformity of grade, temper after heat treatment, and body hardness of the metal.

The Brinell hardness test is performed by applying a known load to the surface of the metal through a hardened steel ball of known diameter. The diameter of the resulting impression is measured. The Brinell hardness number is calculated using the applied load, the diameter of the steel ball, and the diameter of the impression, as follows:

$$BHN = \frac{P}{\frac{\pi D}{2}\left(D - \sqrt{D^2 - d^2}\right)}$$

Where:

BHN = Brinell hardness number in kilograms per mm^2
P = Applied load in kilograms
D = Diameter of steel ball in millimeters
d = Diameter of impression in millimeters

To perform a Brinell hardness test using a compressed air tester, follow these instructions:

1. Adjust the air regulator until the desired load is indicated on the load gauge. Check the load reading when making the initial test of a series, and adjust the air regulator valve, if necessary, so that the desired load is indicated on the dial when a specimen is actually under load.
2. Place the test specimen on the anvil, and adjust the gap between the surface of the specimen and the steel ball to a minimum. The test specimen must be thick enough to prevent a bulge or other marking from appearing on the side opposite the impression. If necessary, the surface on which the impression is to be made should be filed, ground, machined, or polished with abrasive material. Preparing the surface this way will help make the indentation clear enough that it can be measured with the necessary accuracy.
3. Make sure the test specimen is in place and then apply and release the load. Do not apply a load without a

Hardness Conversion Table

(Approximate)

Values vary depending on grades and conditions of material involved. Rockwell "B" Scale should not be used over B-100. The "C" Scale should not be used under C-20.

Brinell	Rockwell		Shore Scleroscope	Tensile lb sq. in.	Brinell	Rockwell	Shore Scleroscope	Tensile lb sq. in.
Hardness No.	B Scale	C Scale	Hardness No.	In 1000 lb	Hardness No.	B Scale	Hardness No.	In 1000 lb
782	...	72	107	383	163	84	25	84
744	...	69	100	365	159	83	25	82
713	...	67	96	350	156	82	24	80
683	...	65	92	334	153	81	24	79
652	...	63	88	318	149	80	23	78
627	...	61	85	307	146	78	23	77
600	...	59	81	294	143	77	22	76
578	...	58	78	284	140	76	..	74
555	...	56	75	271	137	75	..	73
532	...	54	72	260	134	74	..	71
512	...	52	70	251	131	72	..	70
495	...	51	68	242	128	71	..	69
477	...	49	66	233	126	70	..	67
460	...	48	64	226	124	69	..	66
444	...	47	61	217	121	67	..	65
430	...	45	59	210	118	66	..	63
418	...	44	57	205	116	65	..	62
402	...	43	55	197	114	64	..	61
387	...	41	53	189	112	62	..	60
375	...	40	52	183	109	61	..	59
364	...	39	50	178	107	59	..	58
351	*(110)*	38	49	172	105	58	..	57
340	*(109)*	37	47	167	103	57	..	56
332	*(108.5)*	36	46	162	101	56	..	55
321	*(108)*	35	45	157	99	54	..	54
311	*(107.5)*	34	44	152	97	53	..	53
302	*(107)*	33	42	148	96	52	..	53
293	*(106)*	31	41	144	95	51	..	52
286	*(105.5)*	30	40	140	93	50	..	52
277	*(104.5)*	29	39	136	92	49	..	51
269	*(104)*	28	38	132	90	48	..	50
262	*(103)*	27	37	128	88	47	..	49
255	*(102)*	26	36	125	87	46	..	48
248	*(101)*	25	36	121	86	45	..	48
241	100	24	35	118	85	44	..	47
235	99	*(22)*	34	115	83	43	..	47
228	98	*(21)*	33	113	82	42	..	46
223	97	*(20)*	33	109	81	41	..	46
217	96	*(19)*	32	106	80	40	..	45
212	95	*(18)*	31	104	79	39	..	45
207	94	*(17)*	30	101	78	38	..	44
202	93	*(15)*	30	99	77	37	..	44
196	92	*(13)*	29	96	76	36	..	43
192	91	*(12)*	29	94	75	35	..	43
187	90	*(10)*	28	91	74	33	..	42
183	89	*(9)*	28	90	73	31	..	42
179	88	..	27	89	72	30	..	41
174	87	..	27	88	71	29	..	41
170	86	..	26	86	70	27	..	40
166	85	..	26	85	69	26	..	40

Goodheart-Willcox Publisher

Figure 29-20. Hardness conversion table.

Buehler, a division of ITW Company

Figure 29-21. A Brinell hardness tester with a digital readout.

specimen in place. Doing so will damage the machine's anvil.

4. Read the impression made in the test specimen with the special microscope and obtain the Brinell hardness number from the hardness table.

29.8.2 Rockwell Hardness Test

The most widely used of all hardness testing methods is the ***Rockwell hardness test,*** **Figure 29-22**. Either a steel ball or a diamond cone penetrator is used in the Rockwell

A Mitutoyo/MTI Corp.

B Wilson Instruments/Instron Corporation

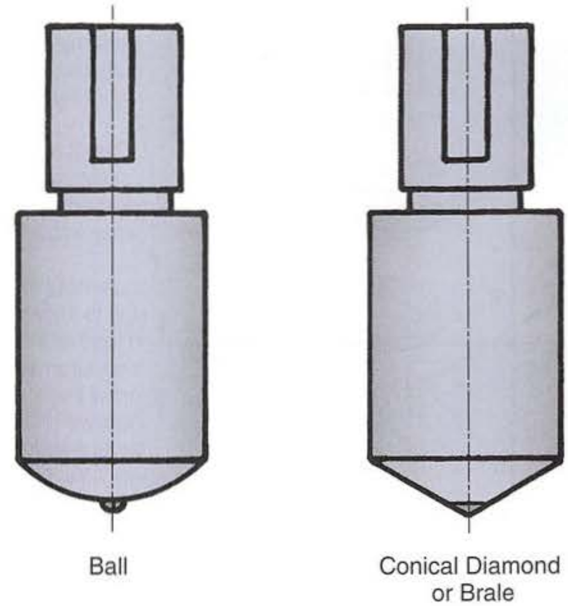
Figure 29-22. Rockwell hardness testers. A—The basic tester. B—This tester has pushbutton controls for all functions. It digitally processes test results, statistical calculations, hardness scale conversions, and corrections for round parts.

hardness tester, depending on the material being tested. See **Figure 29-23**.

A minor load of 10 kg is first applied, and the dial gage is set to zero. The major load is then added and removed, **Figure 29-24**. The hardness number is based on the additional depth to which the test ball or conical diamond penetrator is driven by the major load beyond the depth of the previously applied light load. The hardness number is automatically indicated on the gage.

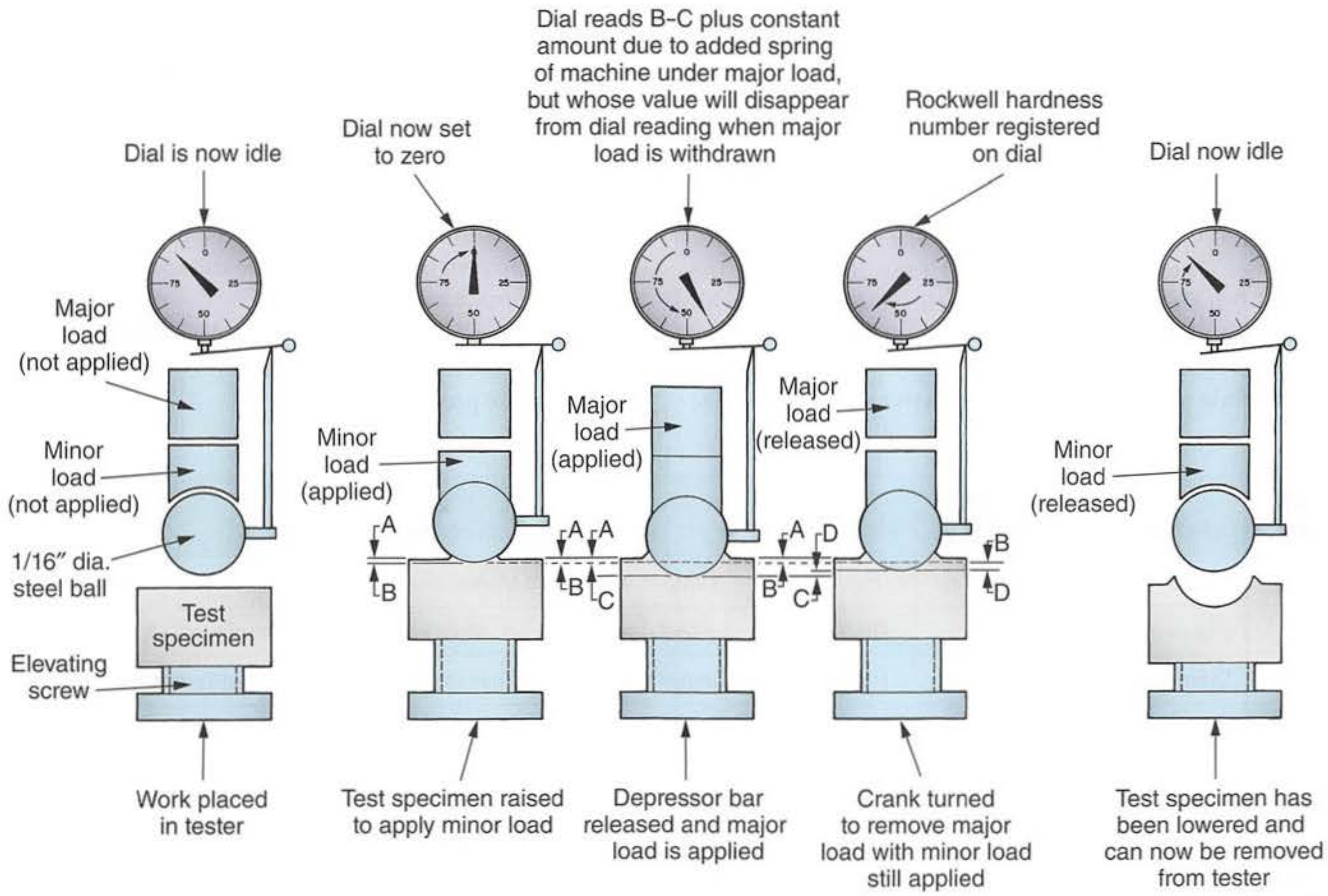
Typically, a 1/16″ steel ball is used with a 100 kg major load for testing metals, such as brass, bronze, and soft steel. All readings made with the 1/16″ ball and 100 kg load are *Rockwell B readings*; the letter *B* must be placed before the number. Rockwell hardness cannot be designated by a number alone. It must always be prefixed by the proper scale letter.

The conical diamond test point, known as a *brale penetrator*, is used with a 150 kg major load for testing hardened steel or any hard metals, **Figure 29-25**. All readings with the brale penetrator and 150 kg load are *Rockwell C readings*; the letter *C* must precede the hardness number.



Goodheart-Willcox Publisher

Figure 29-23. Rockwell penetrators.



Goodheart-Willcox Publisher

Figure 29-24. This diagram shows how a 1/16″ steel ball penetrator is used to make a Rockwell B hardness reading. The size of the ball has been greatly exaggerated for clarity.

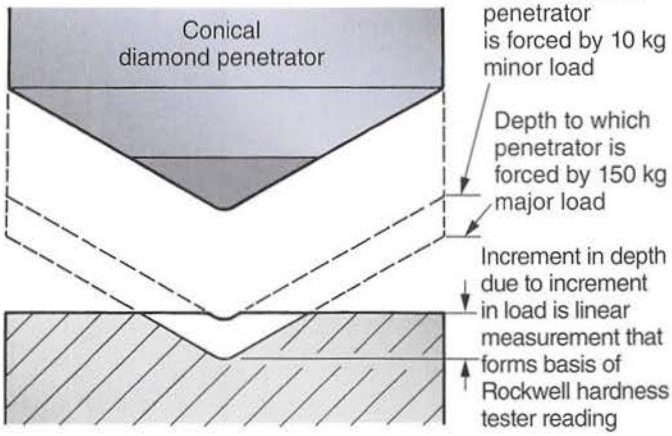


Figure 29-25. A conical diamond penetrator (brale) is used to determine Rockwell C hardness.

Special penetrators are available for testing soft materials. The scale designation is based on the ball size used to make the test. See **Figure 29-26**.

Two weights are normally supplied with a Rockwell hardness tester. One of them has a red marking and the other black. However, three different loads—60 kg, 100 kg, and 150 kg—can be applied. The weight arm, together with the link and weight pan, will apply a load of 60 kg. The weight with the red marking is placed on the weight pan for the 100 kg load. When the weight with the black marking is added, a 150 kg load is applied. The black weight is never used alone. When tests are made with the 100 kg load, the Rockwell B scale is used. The Rockwell C scale is used with the 150 kg load.

The 60 kg load (weight arm and weight pan alone) is used with the brale penetrator for testing extremely hard metals, such as tungsten carbide alloys. The 1/16″ ball is extensively used with the 60 kg load for testing sheet brass.

To operate the Rockwell hardness tester, select and mount the proper penetrator point. Then check that the weight for the desired test load is in position. Place the correct anvil in the elevating screw with extreme care to avoid damaging the penetrator. Inspect the test specimen and remove any scale or burr that would flatten under the test and give a false reading.

The following procedure gives the most precise hardness readings.

1. Place the test specimen on the anvil.
2. Gently raise the specimen until it comes into contact with the penetrator. Continue to raise the specimen until the minor load of 10 kg has been applied.
3. Set the machine to zero.
4. Carefully apply the major load. The penetrator is forced into the work. The depth to which it penetrates depends on the metal's hardness.
5. Wait until the reading stops changing.
6. Lift the major load, but leave the minor load still applied.
7. Read the Rockwell hardness number. If the test has been made with the 1/16″ ball and the load is 100 kg, the reading is made with the Rockwell B scale, and the letter B is prefixed to the number to signify the condition of the test. If the brale penetrator and a 150 kg load were used, the Rockwell C scale is used and the letter C is prefixed to the number.
8. After recording the number, lower the specimen away from the penetrator and remove the specimen from the testing machine.

Like other precision tools, a Rockwell hardness tester must be handled with care if it is to maintain its accuracy. Observe these precautions:

Scale Symbol	Penetrator	Major Load (kg)	Dial Figures	Typical Applications of Scales
B	1/16″ ball	100	Red	Copper alloys, soft steels, aluminum alloys, malleable iron, etc.
C	Diamond cone	150	Black	Steel, hard cast iron, titanium, deep case-hardened steel, etc.
A	Diamond cone	60	Black	Cemented carbides, thin steel, and shallow case-hardened steel.
D	Diamond cone	100	Black	Thin steel, medium case-hardened steel, and pearlite malleable iron.
E	1/8″ ball	100	Red	Cast iron, aluminum and magnesium alloys, and bearing materials.
F	1/16″ ball	60	Red	Annealed copper alloys; thin, soft sheet metals.
G	1/16″ ball	150	Red	Phosphor bronze, beryllium copper, malleable iron, etc.
H	1/8″ ball	60	Red	Aluminum, zinc, lead.

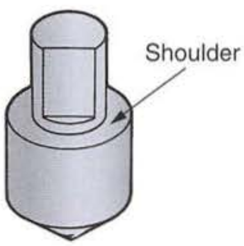
Goodheart-Willcox Publisher

Figure 29-26. A letter is used as a prefix to the hardness value read from the Rockwell tester dial. The letter depends on the load, type of penetrator, and scale from which the dial readings are taken.

- When moving the tester, grasp it only by the cast iron base, never by any parts that are attached to the base.
- Level the tester on a solid bench, in a location free from grit and vibration.
- Keep the machine covered when not in use.
- Carefully wipe the penetrator with a clean, soft cloth before installing it in the tester. False readings will result if the shoulder on the brale penetrator is not kept clean, **Figure 29-27**.
- Use only lubricants specified by the manufacturer.
- Support long work properly.
- Clean the work with an abrasive cloth to remove scale and roughness; a smooth surface is needed to ensure accurate readings. For castings and forgings, grind or machine a spot where the test is to be made, so that the penetrator will test the metal beneath the surface.
- Make sure the specimen is thick enough that the underside of the specimen does not show the slightest indication of the test.
- Add corrections to readings made on round stock, if it is not possible to file or grind a flat spot in the test area.
- Never force the penetrator and the anvil together when a test piece is not in the machine. Doing so will damage the equipment.
- Remember that if the tester is used on case-hardened steel, accurate readings cannot be made unless the "case" is several times thicker than the indentation depth.

29.8.3 Webster Hardness Tester

The ***Webster hardness tester***, **Figure 29-28**, is a portable testing device for checking the hardness of materials such as aluminum, brass, copper, and mild steel. It can be used on assemblies that cannot be brought into the laboratory, or to test a variety of shapes that other testers cannot check, such as extrusions, tubing, or flat stock. The tester's dial reading is converted to the Rockwell hardness scale by referring to a conversion chart furnished with the tool.



Goodheart-Wilicox Publisher

Figure 29-27. The shoulder of the brale penetrator must be free of dirt and burrs before it is mounted in the tester in order to prevent incorrect readings.

Webster Instrument, Inc.

Figure 29-28. The Webster hardness tester is a portable device used with materials such as aluminum, brass, copper, and mild steel. By referring to a conversion chart, the tester's dial indicator reading can be converted to the appropriate Rockwell scale.

29.8.4 Scleroscope Hardness Testing Machine

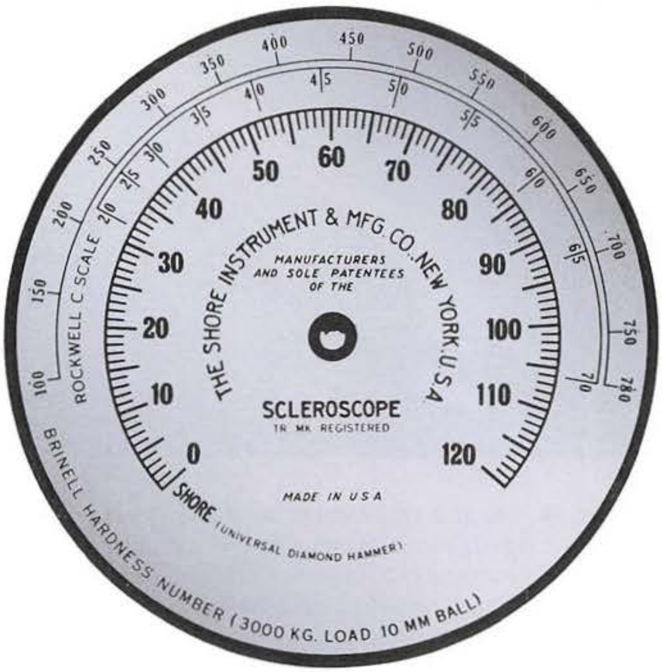
A ***scleroscope*** is a testing device that drops a hammer onto the test piece. The resulting rebound of the hammer determines the hardness.

Two styles of scleroscope are in use. One is fitted with a vertical scale. The other has a dial that records the test results. Scleroscopes can be used to test the hardness of all metals, ferrous and nonferrous, polished or unpolished, with virtually no limitation in size or shape. Hardness testing with the scleroscope is essentially a non-marring test. No craters are produced that would require refinishing of the test area.

In a scleroscope hardness test, a diamond hammer is dropped from a fixed height and makes a tiny indentation in the metal. The hammer rebounds, but not to its original height. The rebound height is always lower than the initial height because some of the energy in the falling hammer is dissipated in producing an indentation. The rebound of the hammer varies in proportion to the hardness of the metal—the harder the metal, the smaller the indentation and the higher the rebound.

The tester scale consists of units determined by dividing into 100 parts the average rebound of the hammer from quenched tool steel of ultimate hardness. These rebounds range from 95 to 105. The scale is carried higher than 100 to cover super-hard metals. See **Figure 29-29**. A scleroscope is capable of making accurate hardness readings on the softest or the hardest metals without changing the scale or diamond hammer.

For hardness testing on a scleroscope, the specimen must be mounted on the anvil. The unit should be leveled,



Goodheart-Willcox Publisher

Figure 29-29. Scales on a scleroscope dial. Note that the dial includes scales for the equivalent Brinell and Rockwell C hardness values.

which is done by turning the leveling screws while observing the built-in spirit level. The instrument is operated pneumatically by means of a rubber squeeze bulb.

To perform a test, bring the barrel cap firmly into contact with the test specimen. It is essential to maintain a firm pressure on the specimen during the test. Squeeze and release the rubber bulb to draw the hammer to the up position. While maintaining torque on the knob, again squeeze and release the rubber bulb to release the hammer. Observe the reading on the scale or dial.

The height to which the hammer rebounds on the first bounce indicates the hardness of the specimen. For greatest accuracy, several tests should be averaged. To avoid false readings, do not make more than one test at a given spot. While the scleroscope hardness testing method may sound unorthodox, the results are very close to those obtained with Brinell and Rockwell testers.

29.9 Heat Treatment Safety

Follow these safety guidelines while you perform heat treatment procedures:

- Never attempt to heat-treat metals while your senses are impaired by medication or other substances.
- Make sure that the furnace is in good operating condition before attempting to use it. Avoid lighting a furnace until you have been instructed in its safe operation. Never stand in front of a gas furnace while igniting it.
- Never look into the furnace unless you are wearing tinted goggles or glasses under your face shield.
- Heat treatment involves raising metal to very high temperatures. Handle the hot metal with appropriate tools, and always wear an approved full face safety shield and the proper protective clothing. Wear heat-resistant gloves and a leather apron, never a cloth apron.
- Work only in areas that are well ventilated.
- Do not stand over the quenching bath when immersing hot work.
- Never use potassium cyanide in a school shop or lab as a case-hardening medium. If you work in a situation that permits the use of potassium cyanide, never breathe the fumes, as they are extremely toxic. Wash thoroughly after completing the heat treatment operations.

Chapter Review

Summary

- Steel and most of its alloys can be hardened.
- Stress-relieving removes internal stresses that have developed in parts that have been cold-worked, machined, or welded.
- Annealing is a process that reduces the hardness of a metal to make it easier to machine or work.
- Normalizing is a process that refines the grain structure of some metals and improves their machinability.
- Hardening is a heat treatment that reduces the ductility of steel.
- Surface hardening is a heat treatment technique used to create a medium-hard surface on high-carbon or alloy steels without affecting the internal structure of the metal.
- Case hardening is a heat treatment technique that creates a hard outer shell on low-carbon steel.
- Tempering is a heat treatment technique that reduces a metal's brittleness or hardness.
- In addition to steel, metals that can be heat-treated include aluminum, brass, copper, and titanium.
- Water, oil, air, and liquid nitrogen are standard quenching media used to draw heat from the part being treated.
- Furnaces are used to heat the metal for most heat treatments. Furnaces can be heated by electricity, gas, or oil.
- The temperature of steel can be determined using a pyrometer or by comparing the color of the heated steel to a color chart supplied by the steel company.
- The simplest method of case hardening is carburizing, in which the low-carbon steel is covered with a case hardening compound and heated.
- Brinell, Rockwell, Webster, and scleroscope hardness testers can be used to measure hardness.
- The use of potassium cyanide in case hardening is dangerous. Never breathe the toxic fumes this procedure produces, and wash thoroughly after completing the heat treatment operations.

Review Questions

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

1. Heat treatment can be used to _____.
   A. change a ferrous metal to nonferrous metal
   B. increase the hardness of a metal
   C. change the thermal conductivity of a metal
   D. All of the above.
   E. None of the above.
2. Water, brine, oil, blasts of cold air, or liquid _____ can be used as a quenching medium.
3. What are the general steps in heat-treating metal?
4. List four metals other than steel that can be heat-treated.
5. Carbon steels are classified by the percentage of carbon they contain, expressed in _____ or hundredths of a percent.
6. The _____ heat treatment process reduces the stress that has developed in parts that have been welded, machined, or cold-worked during processing.
7. The _____ heat treatment process reduces the hardness of metal.
8. The _____ heat treatment process refines the grain structure of steel to improve its machinability.
9. The _____ heat treatment process is used to create a medium-hard surface on high-carbon or alloy steels.
10. The _____ heat treatment process is used to create a hard outer surface on low-carbon steel while keeping the inner portion relatively soft and tough.
11. Tempering a piece of hardened steel makes it _____.
    A. harder
    B. tougher
    C. more brittle
    D. All of the above.
    E. None of the above.
12. Some _____ alloys age-harden at room temperature and must be refrigerated to remain soft and ductile while they are being worked.
13. Although _____ can be heat-treated, special facilities are required because it is a reactive metal.

14. Water is mainly used as the quenching medium when ______.
    A. moderate hardness is desired in plain carbon steel
    B. moderate hardness is desired in alloy steel
    C. maximum hardness is desired in plain carbon steel
    D. All of the above.
    E. None of the above.
15. List three characteristics of electric heat treatment furnaces.
16. A(n) ______ is used to measure and monitor the high temperatures needed in heat treatment.
17. Steel that has been hardened properly is too ______ for most purposes.
18. Briefly describe the best procedure for tempering small objects.
19. Why is cyaniding not a recommended procedure for case-hardening low-carbon steel?
20. The most commonly used technique for testing the hardness of metal is to press a steel ball or ______ probe into the metal under a specific load.
21. List three types of commonly used hardness testers.
22. A ______ determines the hardness of a metal based on the distance that a diamond hammer rebounds after it is dropped on the metal.
    A. Rockwell hardness tester
    B. Brinell hardness tester
    C. Webster hardness tester
    D. scleroscope
23. List three furnace-related safety precautions.

CHAPTER 30

Metal Finishing

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Describe how the quality of a machined surface is determined.
- Explain the effect of the quality of a machined surface on its production costs.
- Identify organic coatings and describe how they are applied.
- List common inorganic coatings and summarize how they are applied.
- Describe four methods of applying metal coatings.
- Explain various methods of deburring.

Technical Terms

anodizing
chemical blackening
electroplating
lay
metal finish
metal spraying
microinches
micrometers
nitriding
roller burnishing
surface roughness standards
vitreous enamel
waviness

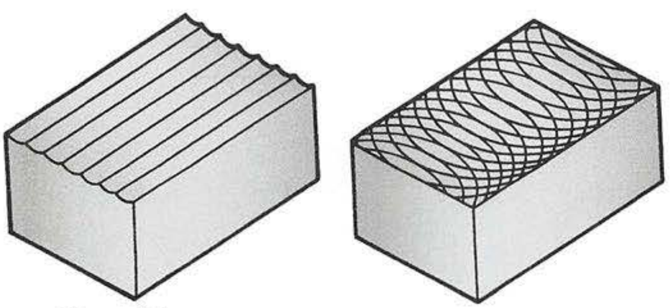
The term ***metal finish*** refers to the degree of smoothness or roughness remaining on the surface of a part after it has been machined. A machined surface has geometric irregularities that are produced by the cutting action of the tool, **Figure 30-1**.

30.1 Quality of Machined Surfaces

At one time, the quality of a machined surface was noted by the symbol "*f*". This was not based on specific standards. Therefore, the engineer or drafter included explanatory notes on the drawing. These notes indicated the desired quality of general surface finish, such as rough grind, smooth turn, or surface grind.

The technique left much to be desired, since each machinist interpreted the specifications differently. Often, the piece was better finished than it had to be, increasing its production cost. The problem reached such proportions that in the early 1940s, the standards associations of Canada, Great Britain, and the United States developed tentative surface roughness or texture standards.

The terms and ratings of ***surface roughness standards***, or texture standards, specify how surfaces produced by machining, grinding, casting, molding, forging, or similar processes are to be measured and communicated. These standards deal only with the height, width, and direction of surface irregularities, since these are of practical importance in specifications. They are not concerned with luster, appearance, color, corrosion resistance, wear resistance, hardness, or any of the other characteristics that may affect a part's suitability for specific applications. They do not define the different degrees of surface roughness and waviness that are suitable for specific purposes, nor do they address how the irregularities are to be made.



Goodheart-Willcox Publisher

Figure 30-1. Each type of cutting tool leaves characteristic markings.

The current surface roughness standards arrive at roughness values by mathematically averaging the size of irregularities on a surface. The values are given in ***microinches*** (millionths of an inch, shown as XX μin) or ***micrometers*** (millionths of a meter, shown as XX μm). With established standards, a universal set of numbers and symbols indicating surface roughness or texture are now used on drawings and in specifications. See **Figure 30-2** and **Figure 30-3**.

GREEN MACHINING

Metal Finishes

Eco-friendly metal finishes, including coatings and anodizing, are nontoxic in their use and disposal.

Traditional coatings used to finish metal products contain harsh chemicals, produce toxic fumes in their application, and generate hazardous waste. Metal coatings may be environmentally friendly in a variety of ways:

- Coatings may be quick-set to save energy required for heating or drying.
- Some coatings are made using advanced technology, such as nanotechnology, to give them properties of traditional coatings without the hazardous materials.
- Powder coating is applied dry and heat-bonded to a metal surface. This method produces less air pollution than the application of typical liquid coatings.

Anodizing is an environmentally friendly alternative to metal coating. Anodizing changes the outside layer of the metal itself, rather than adding a coating. The anodizing process requires no toxic materials and generates no hazardous waste. Anodized metals can be recycled normally, further enhancing their eco-friendliness.

The same standards that address surface roughness also address other surface conditions and give values for them. ***Waviness*** describes the presence of smoothly rounded peaks and valleys caused by tool and machine vibration and chatter, **Figure 30-4**. Waviness is of greater magnitude than roughness. It is measured with reference to a nominal or geometrically perfect surface.

Waviness is specified by the maximum allowable peak-to-valley height in inches or millimeters. It is measured using a sensitive dial indicator with a ball contact 0.06" (1.5 mm) in diameter. **Figure 30-5** shows how acceptable waviness, roughness, and lay tolerances are specified on drawings, in reports, and as specifications. ***Lay*** describes the direction of the predominant tool marks, grain, or pattern of surface roughness. See **Figure 30-6**.

Symbol	Description
√	*Basic surface roughness/texture symbol.* Surface may be produced by any method.
√ (with horizontal bar)	*Material removal by machining required.* Horizontal bar indicates that material removal by machining is required. Material must be provided for that purpose.
.187 √ (with horizontal bar)	*Material removal allowance.* Number indicates amount of material that must be removed in inches/millimeters. Tolerances may be added.
√ (with circle in vee)	*Material removal prohibited.* Circle in vee indicates that surface must be produced by processes, such as casting, forging, hot finishing, cold finishing, powder metallurgy, or injection molding, without subsequent removal of material.
√‾	*Surface texture symbol.* To be used when any surface characteristics are specified above horizontal line or to right of symbol. Surface may be produced by any method.

Goodheart-Willcox Publisher

Figure 30-2. Surface roughness or texture symbols.

Roughness Height Rating		Surface Description	Process
Microinch	Micrometer		
1000 √	25.2 √	Very rough	Saw and torch cutting, forging, or sand casting
500 √	12.5 √	Rough machining	Heavy cuts, coarse feeds in turning, milling, and boring
250 √	6.3 √	Coarse	Very coarse surface grind, rapid feeds in turning, planing, milling, boring, and filing
125 √	3.2 √	Medium	Machining operations with sharp tools, high speeds, fine feeds, and light cuts
63 √	1.6 √	Good machine finish	Sharp tools, high speeds, extra fine feeds, and cuts
32 √	0.8 √	High grade machine finish	Extremely fine feeds and cuts on lathe, mill, and shaper required Easily produced by centerless, cylindrical, and surface grinder
16 √	0.4 √	High quality machine finish	Very smooth reaming or fine cylindrical or surface grinding, or coarse hone or lapping of surface
8 √	0.2 √	Very fine machine finish	Fine honing and lapping of surface
2–4 √	0.05 0.1 √	Extremely smooth machine finish	Extra fine honing and lapping of surface

Goodheart-Willcox Publisher

Figure 30-3. Roughness values. When used on a drawing, a number indicates the roughest surface in microinches or micrometers that is acceptable for that specific application.

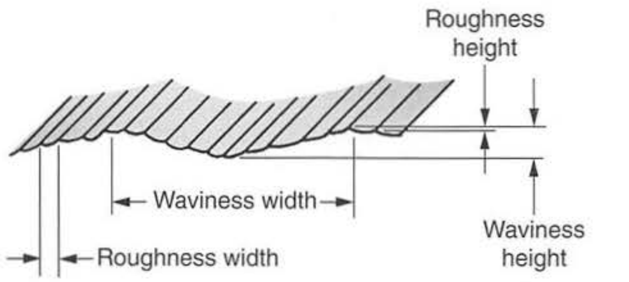


Figure 30-4. The difference in magnitude between waviness and roughness.

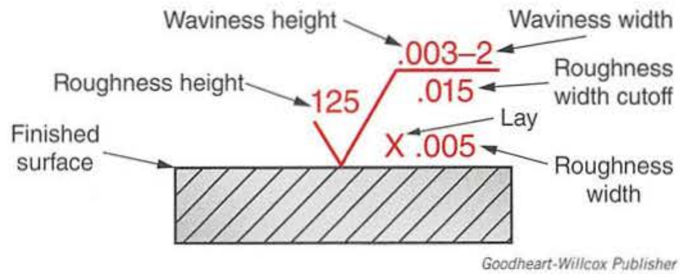


Figure 30-5. On drawings, symbols and numbers show roughness, waviness, and lay. They specify the finishes required on a surface.

30.1.1 Degrees of Surface Roughness

Milling and turning can produce surface finishes on the order of 125 μin to 8 μin (3.2 μm to 0.2 μm). Grinding, depending on the coarseness of the wheel and the feed rate, has a range of 63 μin to 4 μin (1.6 μm to 0.1 μm).

Lapping produces the smoothest finish on a production basis. It is used by automotive manufacturers to produce mating surfaces that are flat and smooth enough to form gasketless oil-tight seals in automatic transmissions and other applications. Surfaces are as fine as 2 μin (0.05 μm). See **Figure 30-7** and **Figure 30-8**.

Several tools have been developed to measure surface quality. The most accurate is the profilometer, which measures and amplifies surface roughness electronically, **Figure 30-9**. Surface roughness is usually read on a meter. A printout of the results can be provided by some models.

A microfinish surface comparator, or surface roughness gage, is a visual comparison tool. It contains sample specimens of the various degrees of surface finishes that conform to values established by the American National Standards Institute (ANSI Y14.36M). The part being inspected is placed next to the gage, **Figure 30-10**. The finish on the part is determined by finding the corresponding finish on the gage.

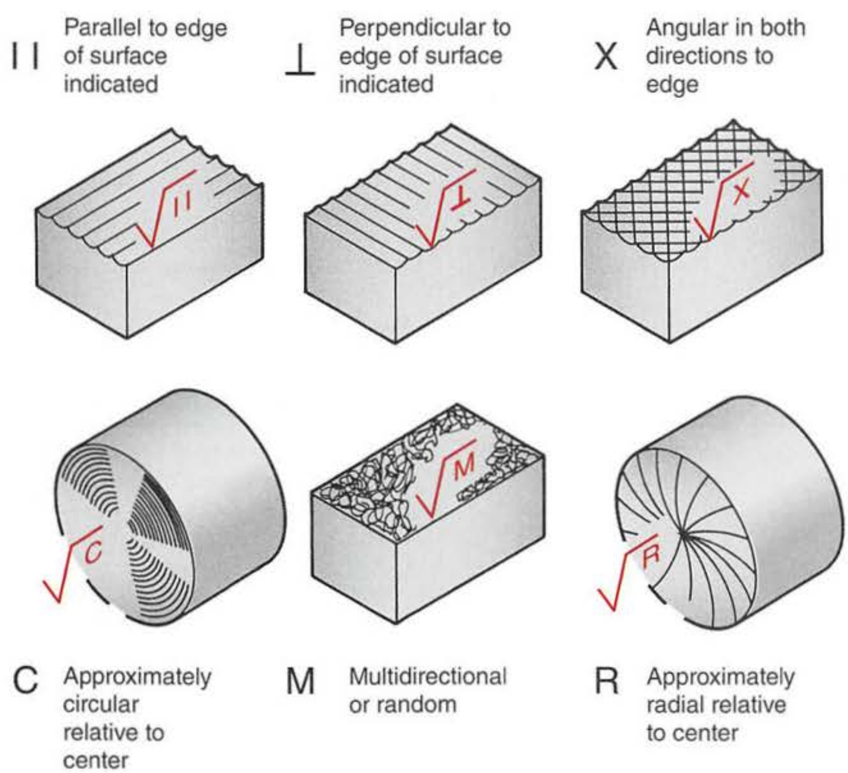


Figure 30-6. The lay symbols and their meanings. Lay symbols are located beneath the horizontal bar on a surface texture symbol.

Sunnen Products Company

Figure 30-7. Computerized vertical honing machine. It is designed for precision honing of bores in applications such as small engine blocks, air compressor cylinders, valve bodies, and aircraft cylinders. The unit has a microprocessor-based control system that enables the operator to control all aspects of the production cycle.

30.1.2 Economics of Machined Surfaces

The quality of a machined surface has a direct bearing on production costs: the finer the finish, the higher the cost. If costs are to be kept within acceptable limits, care must be taken to meet, but not exceed, the required specifications.

The chart in **Figure 30-11** illustrates the range of surface finishes that can be achieved by the various machining processes. Values are relative and vary depending on the condition of the machine, the sharpness of the cutting tool, and the material being machined.

30.2 Other Metal Finishing Techniques

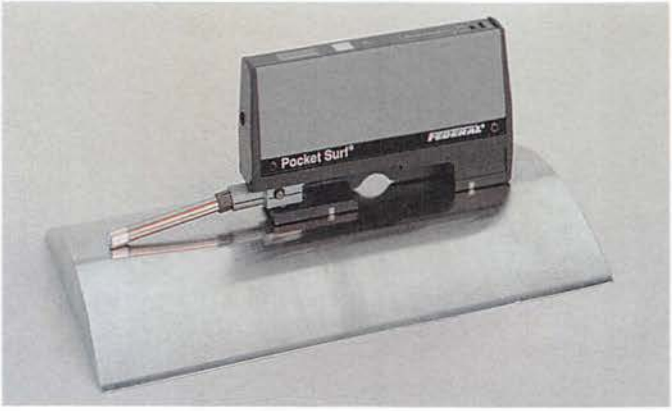
While the quality of the machined surface is of paramount importance in the machining of metal, other finishing techniques may be used for one or more of the following reasons:

- **Appearance.** A proper finish affects a product's salability. A product is much more attractive with a proper finish than when left unfinished. Finishes can also be applied to make the product blend into its surroundings.
- **Protection.** All metals are affected to some degree by contaminants in the atmosphere and by abrasion. A proper finish helps protect materials from exposure to contaminants, reducing corrosion and wear.

A *SpeedFam*

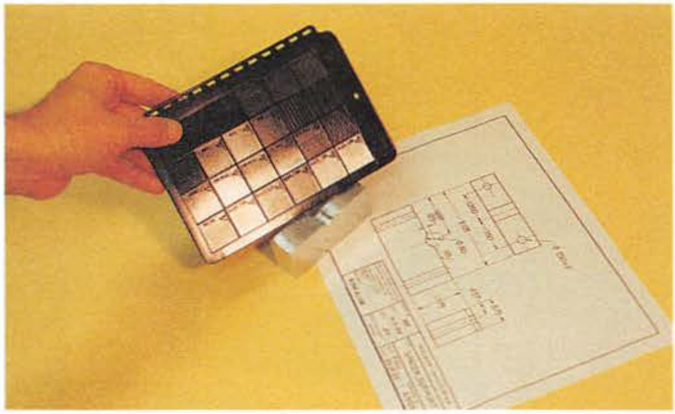
B *Engis Corporation*

Figure 30-8. Lapping machines. A—Free abrasive lapping machines provide exceptional accuracy with part thickness accuracies of ±0.00015″ (0.00381 mm), flatness of 0.00001″ (0.000254 mm), and a fine surface finish. B—A twin-plate diamond lapping system. It laps or polishes both sides of a workpiece simultaneously and produces parallel parts with uniform edge-to-edge flatness. The unit uses a menu-driven microprocessor controller.



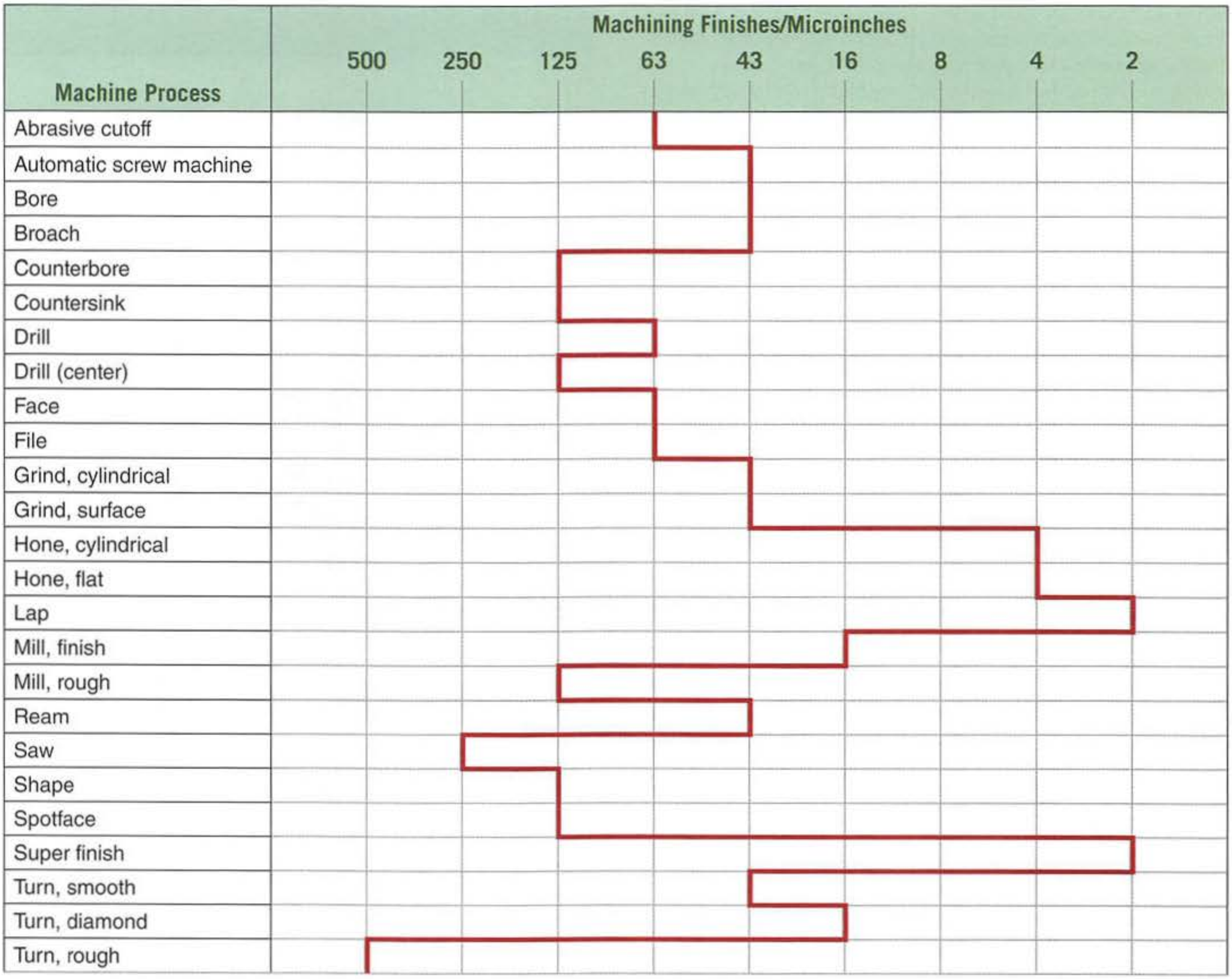
Federal Products Co.

Figure 30-9. Handheld surface roughness gage (profilometer) with a digital display that shows the measured value in microinches or micrometers. It has a variety of interchangeable probes for different applications.



Goodheart-Willcox Publisher

Figure 30-10. A surface roughness gage is used to visually compare the surface of a machined block. Since it is often difficult to check machined surfaces visually, a test by feel is also used.



Goodheart-Willcox Publisher

Figure 30-11. Typical surface finishes that result from various machining processes. The finer the finish (the lower the roughness value in microinches), the higher the cost of achieving it.

- **Identification.** Some manufacturers use distinctive finishes on their products that make their products stand out from their competition.
- **Cost reduction.** An expensive metal can be coated onto a less costly metal or other material. The finished part is less costly than if it were made entirely from the more expensive metal. The part retains the properties of the higher-priced metal. Silver and gold, for example, are often applied to steel to improve electrical conduction, heat distribution properties, solderability, and appearance.

Figure 30-12 shows the many finishes that can be applied to aluminum. Regardless of the finish used, the surfaces must be cleaned of all contaminants before the finish can be applied. Oxidation can be removed mechanically (sandblasting or burnishing) or chemically (etching with an acid). Solvents are often used to remove oil and grease.

Methods have also been devised to clean complex machined castings. Special methods are needed to remove loose casting sand, metal chips, and other foreign matter trapped in recesses and passages during manufacturing operations. One method is to dip the part in a solvent tank. The flow of the solvent and agitation of the part removes foreign matter from deep cavities, holes, and recesses of the casting. The final operation of the cleaning cycle is the application of rust-inhibiting chemicals.

30.2.1 Organic Coatings

A wide range of finishes fall into the category of organic coatings: paints, varnishes, lacquers, enamels, various plastics, and epoxies in both clear and pigmented (color) formulas. With the exception of the epoxies, these coatings are set by the evaporation of their solvents. This may be accomplished by air drying or baking. Epoxies require the addition of a catalyst or hardener to set. Organic finishes are seldom applied to machined surfaces.

A primer is often required to ensure satisfactory bonding between the metal and the finish. Some types of castings may need a filler to smooth out the rough cast surface.

Organic coatings are applied in the following ways:

- **Brushing.** With this method, a brush is used to apply the finish directly to the part. At one time, brushing was the only way to apply finishes with any degree of control.
- **Spraying.** In this method, the finishing material is atomized and carried to the work surface by air pressure. Spraying is easily adapted to mass-production techniques, **Figure 30-13**. For small jobs, small, pressurized spray cans are available in a wide range of colors.
- **Roller coating.** A roller is used to apply the finish directly on the part. This method can be used only on flat surfaces. It is a low-cost technique that can be mechanized.
- **Dipping.** The part is submerged in the finish, removed, and allowed to dry. Dipping is widely used today by the automotive industry to apply body primer and rustproofing, **Figure 30-14**. The coating is dried by warm air.
- **Flow coating.** The part is flooded with the finish and allowed to drain while held in an atmosphere saturated with solvent vapor. Drying is thus delayed until draining is complete.

30.2.2 Inorganic Coatings

Several well-known finishing techniques, such as anodizing, glass coating, and chemical blackening, are inorganic coatings. ***Anodizing*** is the best known inorganic finish. This process forms a protective layer of aluminum oxide on aluminum parts, **Figure 30-15**.

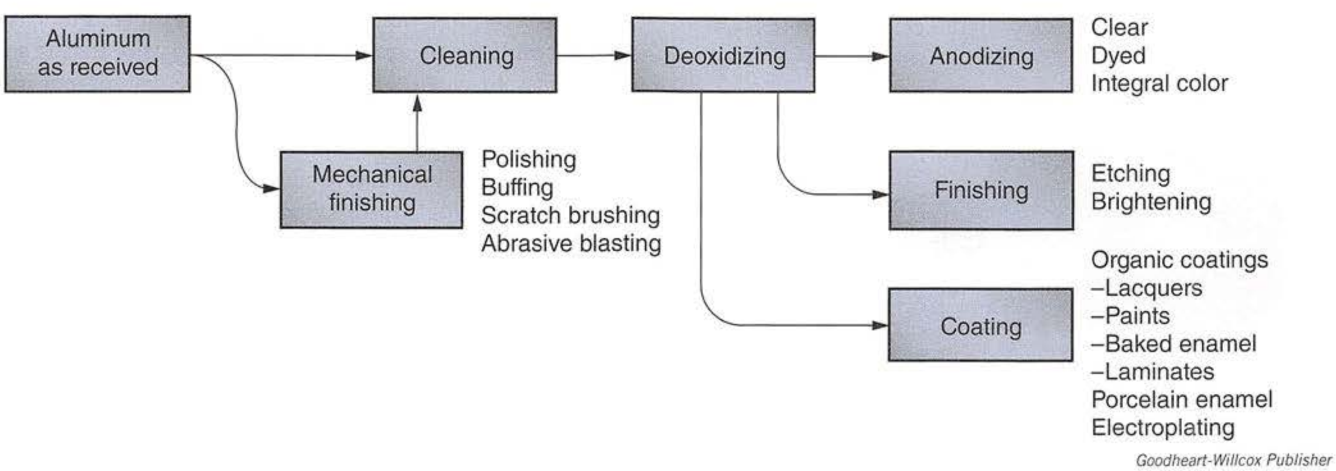
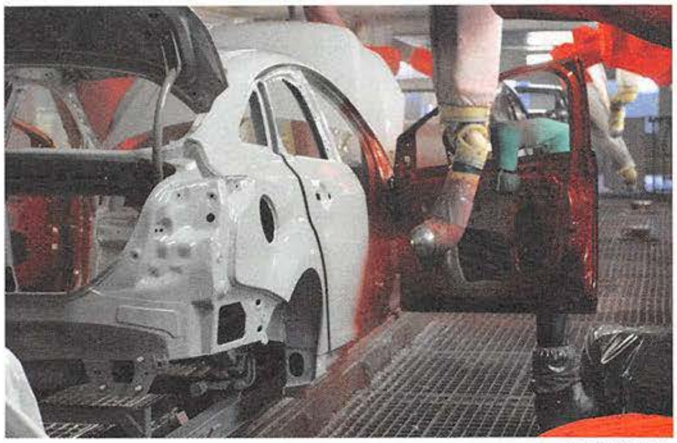


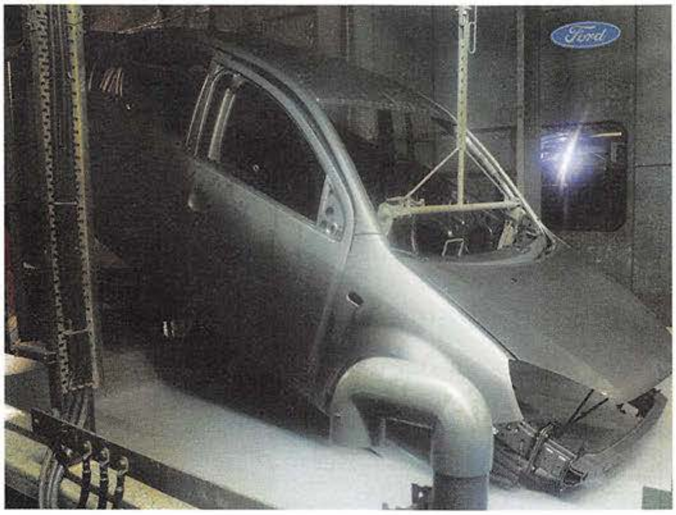
Figure 30-12. Various finishes that can be applied to aluminum.

A Corepics VOF/Shutterstock.com



B Ford Motor Co.

Figure 30-13. With spraying, the finishing material is atomized and carried to the work surface by air pressure. A—Many finishing materials and solvents pose a serious health hazard when atomized. This worker is wearing the proper protective gear, including rubber gloves, air-supplied respirator, and a protective suit. B—Because of the health hazards and repetitive nature of the task, mass-production spray painting is often automated.



Ford Motor Co.

Figure 30-14. Electrically charged particles of paint will evenly coat all bare surfaces of this vehicle body as it is submerged into a tank of primer. Because the paint will be drawn into every crevice, this technique provides better corrosion prevention than spraying.

Kalabi Yau/Shutterstock.com

Figure 30-15. Anodized aluminum fitting.

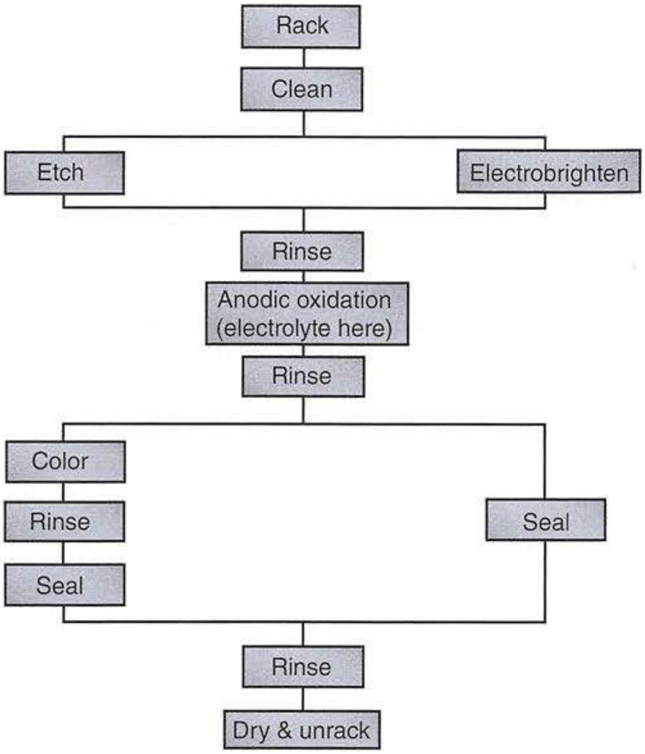
The three classes of anodizing are ordinary anodizing, hard-coat anodizing, and electrobrightening. The basic procedure for all three is the same, **Figure 30-16**.

The anodized coating of aluminum oxide forms when an aluminum anode reacts with an electrolyte. Oxygen is liberated at the surface of the aluminum, and an oxide forms. Ordinary anodizing leaves an oxide layer of 0.0001″ to 0.0006″ (0.003 mm to 0.015 mm) thick on the surface of the aluminum. Hard-coat anodizing produces an oxide layer about 10 times thicker than that produced by ordinary anodizing. This gives it superior resistance to abrasion, erosion, and corrosion. However, the strength of the material is slightly reduced.

Electrobrightening is an anodizing process in which the oxide film is dissolved by electrolyte at about the same rate that it is formed. The process leaves a smooth, bright, mirror-like finish. The anodized coating can be dyed in a wide range of colors. The color becomes part of the surface of the metal.

Vitreous enamel, or porcelain, is a glass coating that can be fused to most metals, including steel sheet and cast iron surfaces. It forms an extremely hard coating that is smooth and easy to clean, **Figure 30-17**. Vitreous enamel is available in many colors. It can be applied to most metals that remain solid at the firing (melting) temperatures of the coating material. The finish is applied as a powder (frit) or as a thin slurry (slip). After drying, the material is fired at about 1500°F (815°C) until it fuses to the metal surface.

Chemical blackening is a finishing process that produces a black oxide coating on the surfaces of ferrous metals, **Figure 30-18**. It chemically bonds with the metal surface, instead of merely adhering to it like paint or other applied finishes. Blackening is used as a finish on precision parts that cannot tolerate the added dimensional thickness of paint or plating. The blackening is only 0.00003″ (0.0008 mm) thick. It will not chip or peel.



Goodheart-Willcox Publisher

Figure 30-16. Procedure for anodizing aluminum.

Kamira/Shutterstock.com

Figure 30-17. Vitreous enamel, or porcelain, is a glass coating fused to steel or cast surfaces. It is an extremely hard coating that is smooth, easy to clean, and heat resistant. These qualities make it a popular finish for a wide range of products, including the exterior of kitchen cookware.

The blackening process can be carried out at room temperature and takes about 15 minutes to complete. Operators must wear eye protection and gloves that provide chemical protection and a plastic or rubber apron to protect clothing.

mrHanson/Shutterstock.com

Figure 30-18. Chemical blackening is a surface finishing technique that can be applied to steel parts and products. It bonds with the metal's surface, rather than coating it; there is no surface buildup. The bottom of the drill chuck and the chuck key have been chemically blackened.

Chemical blackening offers several advantages:

- The black finish enhances the appearance of many items. Because the cost is so low, it is possible to finish parts that were previously left unfinished.
- It aids in protecting the machined surface against humidity and corrosion.
- It reduces glare from the material. Using blackened tools and other machine parts results in less eye fatigue in the operator, improving safety.
- It improves abrasion resistance.
- It improves the adhesion qualities of the treated metal. Paint and other finishes applied over blackening take hold faster, adhere more strongly, and last longer.

There are two drawbacks to the process, both involving environmental treatment and disposal laws:

- The water used in the process must be filtered and treated before it can be discharged as wastewater.
- Chemical residue from the process must be handled as toxic waste.

30.2.3 Metal Coatings

With the exception of electroplating, which can be used on a number of materials, metal coatings are applied primarily to steel. Metal coatings adhere tightly to steel surfaces and provide protection from corrosion, improved wear resistance, enhanced appearance, or a combination of these qualities, **Figure 30-19**. The most commonly used metal coating processes include electroplating, nitriding, metal spraying, and detonation gun coating.

Baloncici/Shutterstock.com

Figure 30-19. The cutting edges of several of these cutting tools are coated with titanium nitride. The titanium nitride has a golden appearance. It is extremely hard and improves a tool's cutting ability and wear resistance.

Electroplating

Electroplating is a process in which an electric current is used to deposit a layer of one metal on the surface of another metal. Practically any metal can be used as a coating. Chrome plating is one common example of electroplating. See **Figure 30-20**.

With electroplating, coating thickness can be closely controlled. Unlike many other metal coating processes, electroplating can deposit coatings that improve both wear resistance and appearance. When a product is to be electroplated, a machining allowance must be made for the additional metal thickness that will be deposited by the plating process.

Nitriding

Nitriding is a process in which nitrogen is diffused into a base metal to create a case-hardened surface. Many alloying elements are used to vary the hardness of the coating. For instance, alloying titanium with the nitrogen produces a rich, gold color. Chromium nitride produces a finish similar to chrome plating, but with a hint of the color black in the deposit. Titanium carbon nitride produces a bluish-black

WORKPLACE SKILLS

Discrimination in the Workplace

Federal and state laws protect workers from discrimination because of their race, color, religion, sex (including pregnancy), national origin, age, or disability. Federal laws cover businesses with 15 or more employees. However, the laws in many states cover more businesses and prohibit harassment based on other characteristics.

Federal laws that deal with discrimination and harassment include the following:

- The Fair Labor Standards Act (FLSA) of 1938 regulates minimum wage, overtime pay, and child labor. It is updated each time the minimum wage is raised. This law limits the hours that a child under the age of 16 may work.
- The Equal Pay Act of 1963 requires that men and women receive the same pay for jobs with the same requirements and responsibilities. Exceptions are made if a person gets a pay raise based on seniority or merit.
- The 1964 Civil Rights Act bans employment discrimination based on race, color, religion, sex, or national origin. This law also created the Equal Employment Opportunity Commission (EEOC).
- The Age Discrimination in Employment Act of 1967 bans discrimination against workers age 40 and older. The Older Workers Benefit Protection Act of 1990 allows workers to sue employers over age discrimination. These laws are both important as the aging population continues to grow.
- The Immigration Reform and Control Act of 1986 prohibits discrimination against US citizens born outside the United States. The Immigration Act of 1990 made it more difficult for noncitizens to get employment in the United States.
- The Americans with Disabilities Act of 1990 prohibits discrimination against people with disabilities. As long as a person's disability does not interfere with the ability to do the job, he or she must be considered for the position. This law also requires places of employment to be physically accessible to people with disabilities.

Many of these laws also cover issues that do not impact the workplace. To read more about them, check out the websites for the EEOC and the US Department of Labor.

Allison Herreid/Shutterstock.com

Figure 30-20. Many hand tools are chrome plated. Chrome plating improves the wear resistance and corrosion resistance of the tool and also gives the tool an attractive luster.

finish similar to the finish applied to a firearm. While nitriding is too costly for common metal finishing, using the technology for applications such as firearms coating not only provides an attractive surface that protects the base metal from corrosion, but also hardens the coated surface, making it more durable.

Metal Spraying

In ***metal spraying***, a metal wire or powder is heated to its melting point and sprayed by air pressure to produce the desired coating on the work surface. See **Figure 30-21**. Most inorganic materials that can be melted without decomposition can be applied by spraying. Flame-sprayed coatings can be applied to build up worn or scored surfaces so they can be remachined to the required size. Superhard coatings can be sprayed when abrasion-resistant surfaces are needed, **Figure 30-22**.

Photo: Sulzer

Figure 30-21. The plasma spray coating is one of several metal spraying techniques used in industry. In this photo, a hard metallic surface is being applied to the stator case of an aviation turbine.

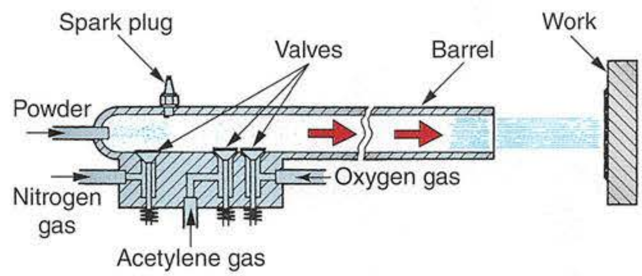
Ford Motor Co.

Figure 30-22. This plasma-transferred wire arc gun applies a hard, low-friction coating to the inside of engine cylinders. The gun is inserted into each cylinder, rotated to evenly apply the material to the cylinder walls, and then withdrawn. After the cylinders are coated, they are machined to the proper size.

Detonation Gun Coating

The detonation gun coating, or D-Gun™, process is another technique for depositing a metallic coating on a workpiece. The process was invented and developed by the Union Carbide Corporation. The detonation gun is essentially a water-cooled barrel several feet long and about 1″ (25 mm) in diameter. It has valves for introducing gases and material to be sprayed. See **Figure 30-23**.

A carefully measured mixture of gases (usually oxygen and acetylene) is fed into the barrel, along with a charge of coating material in powder form. The powder has a particle size of less than 100 microns. A spark ignites the gases. This heats and melts the powder and creates a high-temperature,



Union Carbide Corp.

Figure 30-23. Cross section of a detonation gun used to apply metal coatings. The oxygen and acetylene valves open to allow explosive gases into the barrel and then seal the gas passages during the detonation phase. After detonation, the nitrogen valve opens briefly to allow a pulse of nitrogen gas to clear the barrel before the process repeats.

high-velocity gas stream. The stream of gas carries the molten material to the surface to be coated. A pulse of nitrogen purges the barrel after each detonation. This process is repeated many times per second.

Each detonation, called a *pop*, results in a circle of coating material a few microns thick and about an inch in diameter. The molten coating droplets quickly solidify on the workpiece surface. The complete coating consists of many overlapping pops that build up the required thickness.

The process can be used to apply coatings with very high melting points to fully heat-treated parts without danger of changing the metallurgical properties or strength of the part and without danger of thermal distortion. The application process can also be fully automated.

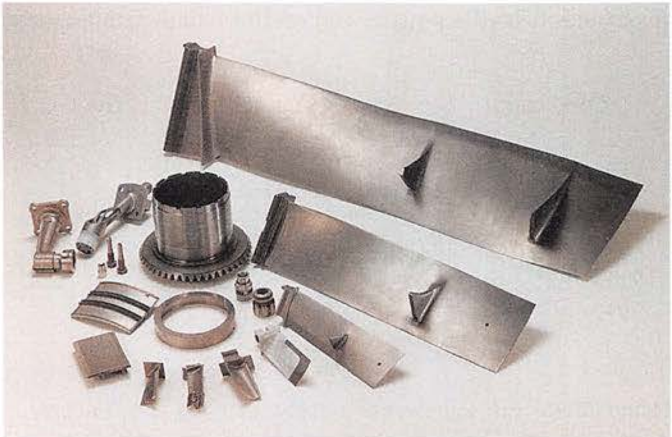
Almost any material that can be melted without decomposing can be sprayed. Coatings include pure metals and metallic alloys, such as nickel and nichrome, tungsten carbide, and ceramics. These coatings are used in many applications, especially for parts that need to resist abrasive, erosive, or adhesive wear, often in very corrosive environments. Refer to **Figure 30-24**.

30.2.4 Mechanical Finishes

Buffing is a power polishing operation. Buffing wheels are attached to a buffing lathe or bench grinder. The wheels are charged with different grades of abrasives that remove scratches and polish the metal's surface to a high luster. Diamond dust and air-powered, handheld polishing units are used to polish the hardened steel dies used for die casting or the injection molding of plastics. See **Figure 30-25**.

SAFETY NOTE

Unless the buffing lathe is equipped with a high efficiency dust-collection system, an approved filter mask must be worn during any buffing operation.

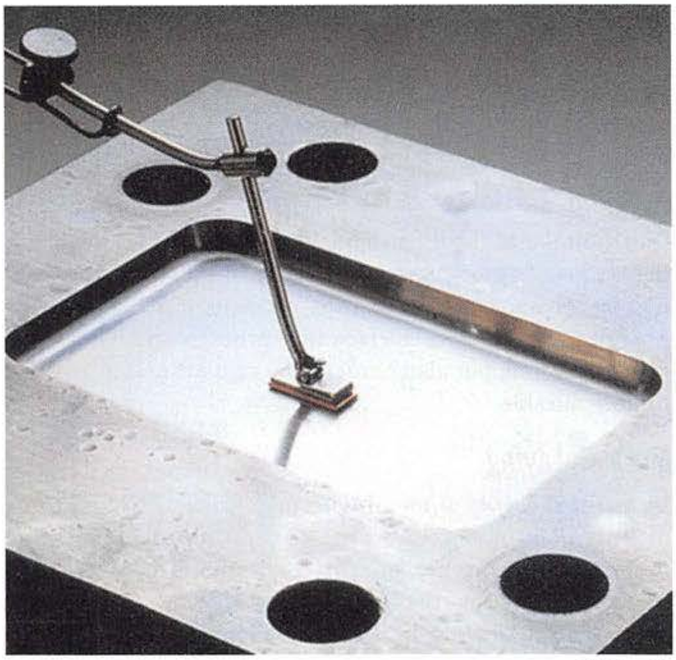


Union Carbide Corp.

Figure 30-24. Some examples of gas turbine engine parts with metal and ceramic coatings that have been applied using the detonation gun coating process.

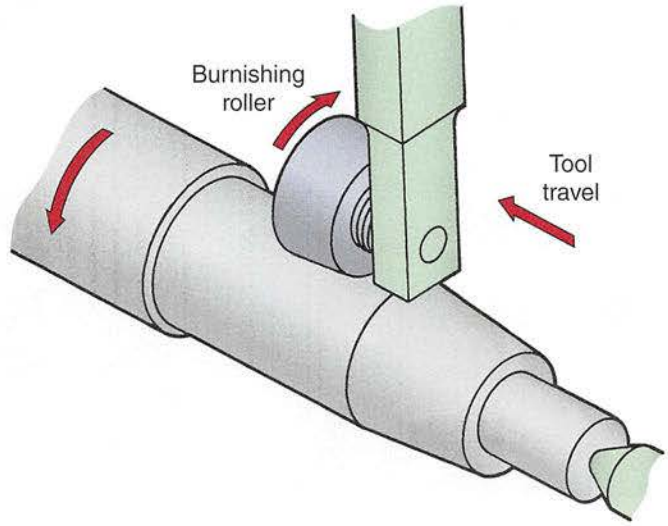
Roller burnishing, **Figure 30-26**, is a cold-working operation. It removes no metal, but rather uses rollers to compress or flatten the peaks of a metal's surface into the valleys. With this process, no honing or grinding is necessary.

Both inner diameter (ID) and outer diameter (OD) burnishing are possible and can be done at speeds and feeds comparable to those of cutting tools. A roller burnishing tool for finishing the inside surfaces of bores is shown in **Figure 30-27**.



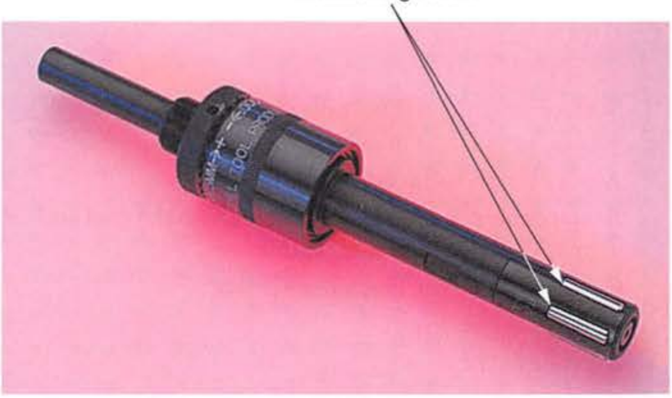
NSK America

Figure 30-25. Polishing a steel die with diamond dust to produce a mirror-like finish.



Goodheart-Willcox Publisher

Figure 30-26. Roller burnishing is a cold-working operation that gives a high quality finish to the metal. It flattens the peaks of a metal's surface into the valleys.



Cogsdill Tool Products, Inc.

Figure 30-27. Roller burnishing tool for finishing holes. It sizes, finishes, and work-hardens the metal. The process removes no metal. Each hole size requires a different roller burnishing tool.

The tool can size, finish, and work-harden metal parts to tolerances as close as 0.0002″ (0.005 mm), and surface finishes as fine as 2 μin (0.05 μm).

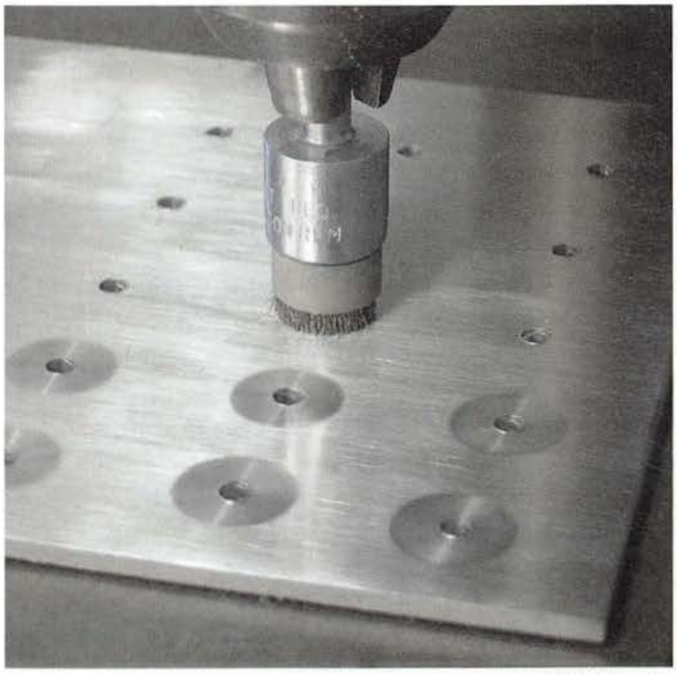
30.2.5 Deburring Techniques

Power brushing is frequently used to remove burrs from machined surfaces. Wire or fiber brushes replace the buffing wheels on the buffing lathe for flat surfaces or are installed in a drill press to deburr bores, **Figure 30-28**. While removing burrs, the wheels produce a satin sheen on the brushed surface, **Figure 30-29**.

Hand deburring of small holes and intricate parts is tedious and expensive, so other methods are preferred for high-volume work. One technique known as *abrasive flow machining*, or *extrude honing*, uses silicon putty permeated with finely divided abrasive particles. The silicon putty is forced into and through the part to be deburred. As the putty flows through passages in the part, the abrasive grains remove any burrs. Abrasive flow machining produces a very uniform finish.

Weiler Corporation

Figure 30-28. Drill press equipped with a wire brush to remove burrs from bores. Always wear a dust mask when performing a deburring, brushing, or buffing operation.



Weiler Corporation

Figure 30-29. A wire brush installed in a drill press is being used to deburr the edges of bores through a metal plate.

Chapter Review

Summary

- Surface roughness standards specify how surfaces produced by machining, grinding, casting, molding, forging, or similar processes are to be measured and communicated.
- Different machining processes create different degrees of roughness in the machined surface. Generally, the finer the finish, the greater the production cost to achieve that finish.
- Finishes can perform any of the following four functions, or any combination of the four: enhance appearance, provide protection against wear and corrosion, make the part more identifiable, and reduce production cost of the part.
- Organic coatings include paints, varnishes, lacquers, enamels, plastics, and epoxies. They are applied by brushing, spraying, roller coating, dipping, or flow coating.
- Inorganic coatings include finishes produced by anodizing, glass coating, and chemical blackening.
- Common processes for applying metal coatings include electroplating, nitriding, metal spraying, and detonation gun coating.
- Other finishing processes include buffing and roller burnishing.
- Methods of deburring include power brushing with wire or fiber brushes and abrasive flow machining (extrude honing).

Review Questions

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

1. At one time, the "*f*" symbol was used on drawings to designate the quality of a(n) ______ surface.
2. Why was the use of the "*f*" symbol discontinued?
3. Surface roughness is now measured in ______ or ______.
4. The term *waviness* describes ______.
   A. very rough surfaces
   B. smoothly rounded undulations caused by tool and machine vibration and chatter
   C. scratches on the machined surface
   D. All of the above.
   E. None of the above.
5. Lay is another surface finish condition. What does it mean?
6. While the quality of a machined surface is of paramount importance in the machining of metal, other finishing methods are used in the machine shop. They are used for one or more of the following reasons. Explain each.
   A. Appearance.
   B. Protection.
   C. Identification.
   D. Cost reduction.
7. Regardless of the type of finish applied to aluminum, the surface must be thoroughly ______ of all contaminants.
8. Paints, lacquers, and enamels are in the family of ______ coatings.
9. List the five ways used to apply the finishes you named in Question 8.
10. List three types of anodizing.
11. What is electroplating?
12. Explain how roller burnishing can finish a surface without removing metal.

CHAPTER 31

Electromachining Processes

Chapter Outline

Learning Objectives

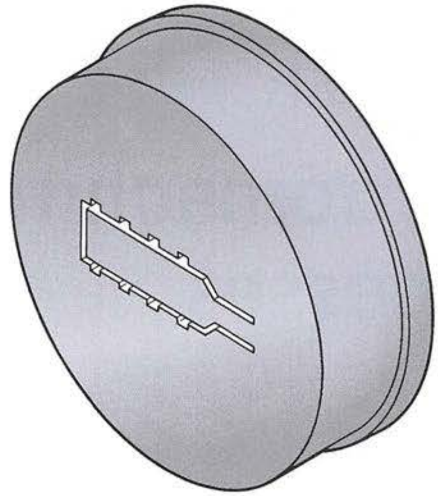
After studying this chapter, you will be able to:
- Explain the advantages and disadvantages of the electromachining processes.
- Describe electrical discharge machining.
- Explain electrical discharge wire cutting.
- Summarize the small-hole EDM drilling process.
- Describe electrochemical machining.

Technical Terms

dielectric fluid
electrical discharge machining (EDM)
electrical discharge wire cutting (EDWC)
electrochemical machining (ECM)
electrode
electromachining
servomechanism
small-hole EDM drilling

The most notable advantage of ***electromachining*** is that mechanical forces have no influence on the processes. The tool is not in direct physical contact with the material being removed, as in conventional machining techniques. Rather, electrical energy is applied to remove metal through erosion.

Electrical discharge machining and electrochemical machining are two electromachining processes that have made a great impact on the field of metalworking and machining. Notably, neither process produces chips as metal is removed. Instead, the particles are either vaporized or reduced to microscopic particles. In order to be machined by either of these processes, the metals must be able to conduct electricity.



Goodheart-Willcox Publisher

Figure 31-1. Die blanks, like this extrusion die used to produce storm window frames, can be machined by EDM after being heat-treated. The job can be done quickly, with no possibility of die warping, and at a considerable cost savings over traditional machining techniques.

31.1 Electrical Discharge Machining (EDM)

Electrical discharge machining (EDM) is a process in which material is eroded from a workpiece by an electric arc between the electrode and workpiece. This process can work tough, hard, fragile, or heat-sensitive metals, which are difficult to machine by conventional techniques, to close tolerances.

Die blanks, for example, can be worked after heat-treating, **Figure 31-1**. This eliminates the warping and distortion that frequently occur when a finished die is heat-treated. Superhard metals can easily be worked to tolerances as close as 0.0002″ (0.005 mm). Surface finishes can vary from very rough to almost mirror-smooth. "Washed-out" (worn) dies can also be reworked in the heat-treated state.

31.1.1 EDM Principle

In a gasoline engine, sparking (or arcing) occurs at the spark plug gap when the ignition coil fires to ignite the fuel mixture. The spark plug's electrodes are gradually eroded by the action of the electric arcs. This is the basis of EDM. See **Figure 31-2**.

An electrical discharge machine, **Figure 31-3**, includes the following parts:

- The power supply provides direct current and controls voltage and frequency.

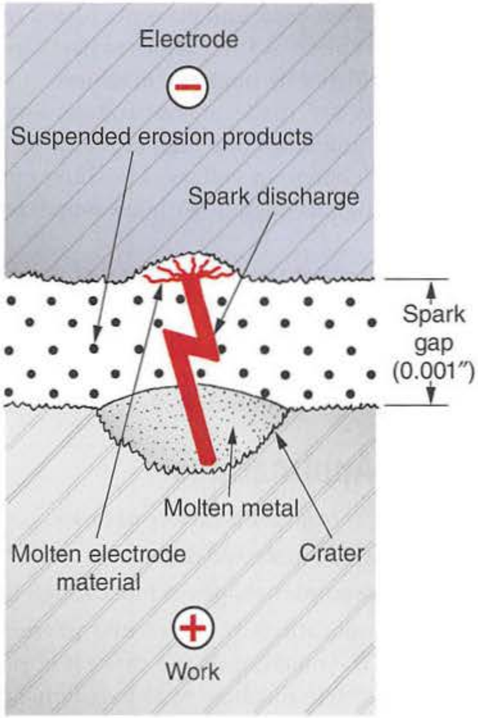
WORKPLACE SKILLS

Entrepreneurship

Entrepreneurs are people who start and run their own business. Many people decide to become entrepreneurs because they enjoy working for themselves and creating their own schedule. Starting a business can provide a great sense of accomplishment and sizable income. Many people begin working from home and advertising on the Internet before renting an office or workspace. These businesses begin with very little overhead or costs, such as rent and employees. Products and completed jobs are easily illustrated online. Most businesses also use social media and customer ratings to help grow their business.

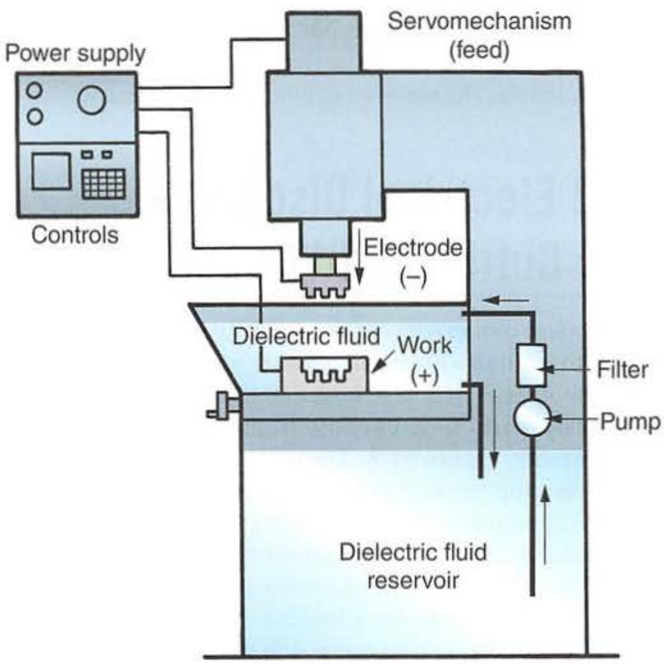
Working for yourself is hard work and sometimes risky. In addition to the work you receive from clients, you must also manage advertising, accounting, and all the other areas associated with a business. Hiring marketing agents and accountants can be costly and not feasible when you first begin a business. It is important that you gain some basic knowledge in these areas before beginning a business venture.

If you are tempted to open your own business, perform adequate research and create a business plan with both short- and long-term goals. Contact other small business owners to gain insight and advice on what you need to make your business successful.



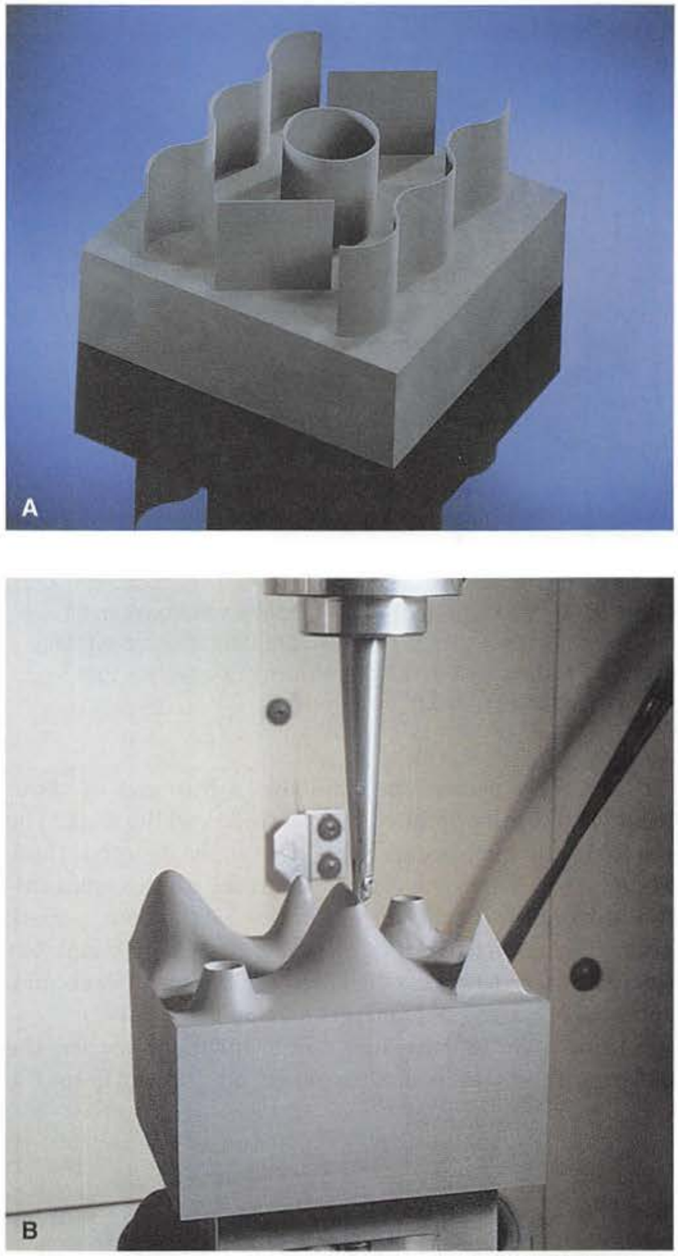
Goodheart-Willcox Publisher

Figure 31-2. The EDM process. A spark (or arc) from an electrode causes the work to erode.



Goodheart-Willcox Publisher

Figure 31-3. Electrical discharge machine.



LeBlond Makino Machine Tool Co.

Figure 31-4. Graphite electrodes. A—This graphite electrode has thin cutting areas. It would be difficult to machine them by conventional means. B—A graphite electrode with a more complex shape. A probe is being used to check whether the electrode's shape meets specifications.

- The ***electrode***, **Figure 31-4**, can be compared to the cutting tool on a conventional machine tool. Graphite has become the material most commonly used for EDM electrodes. Because of the dust produced when machining graphite electrodes, machine tools with enclosed cutting zones and powerful suction systems have been developed, **Figure 31-5**.
- The ***servomechanism*** (drive unit) is used to accurately control electrode movement and to maintain the correct distance between the work and the electrode as machining progresses.
- The ***dielectric fluid***, usually a light mineral oil, is used to form a nonconductive barrier between the electrode and the work at the arc gap.

LeBlond Makino Machine Tool Co.

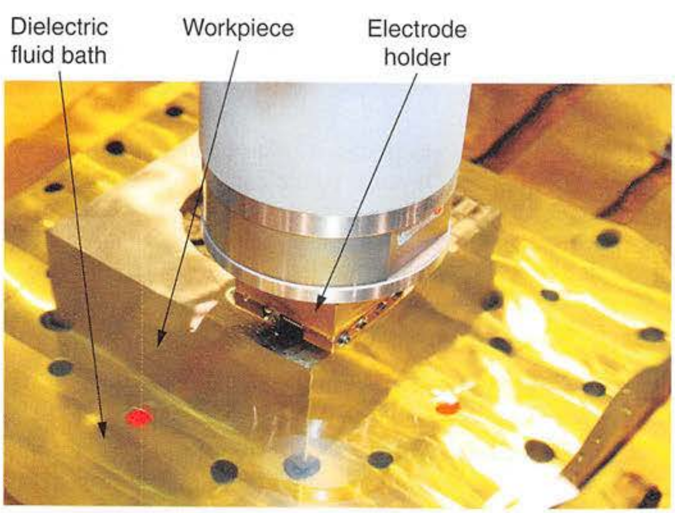
Figure 31-5. Machines for milling graphite electrodes must be environmentally clean. This CNC graphite mill has a totally enclosed cutting zone and a powerful suction system that contains, captures, and removes dust.

The servomechanism maintains a thin gap of about 0.001″ (0.025 mm) between the electrode and the work. The electrode and the work are submerged in the dielectric fluid, **Figure 31-6**. When the voltage across the gap becomes sufficient to cause the dielectric fluid to break down, a spark occurs. Each spark erodes only a tiny particle of metal, but since the sparking occurs 20,000 to 30,000 times per second, appreciable quantities of metal are removed.

In addition to providing a nonconductive barrier, the dielectric fluid also flushes particles from the gap, keeps the electrode and the work cool, and prevents fusion of the electrode with the workpiece. A filter removes particles from the fluid as it is recirculated through the machine.

Roughing cuts are made at low voltage and low frequency, with high amperage and high capacitance providing opposition to any change in voltage. Finishing cuts require high voltage and high frequency, with low amperage and low capacitance.

Hard metals erode at a much slower rate than soft metals. Since the electrode is also consumed, but at a much slower rate than the work, considerable savings can be achieved by making interchangeable electrodes for roughing, sizing, and finishing. Long runs may require several sets of electrodes.



AMT—Association for Manufacturing Technology

Figure 31-6. A thin gap separates the electrode and the work. The dielectric fluid fills the gap and insulates the workpiece from the electrode.

31.1.2 EDM Applications

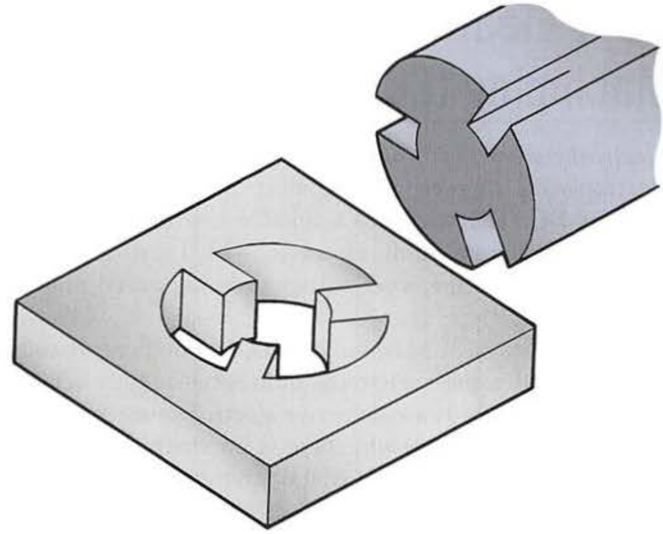
The EDM process is used for a variety of tasks:

- Shaping carbide tools and dies.
- Machining complex shapes in hard, tough metals.
- Machine applications in which the physical characteristics of the metal or its use make it impractical or very expensive to machine by conventional methods.
- Eliminating tedious and expensive handwork in die making. The cavity produced in the metal is a mirror image of the electrode, **Figure 31-7**.
- Drilling holes ranging in size from 0.0012″ to 0.120″ (0.3 mm to 3 mm) in diameter. They can be square, rectangular, triangular, or round. Multiple holes can be produced at the same time. Hardness is not a factor as long as the material can conduct electricity.

31.2 Electrical Discharge Wire Cutting (EDWC)

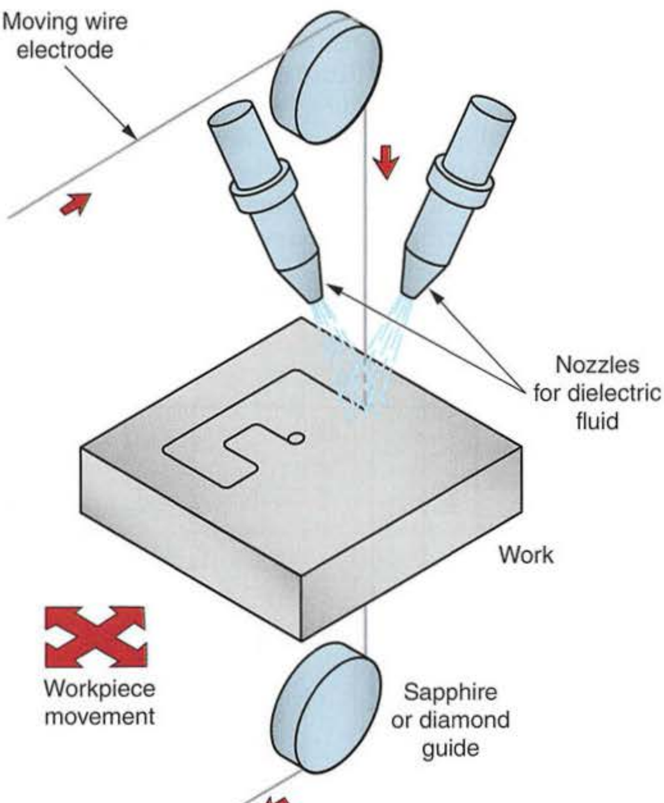
Electrical discharge wire cutting (EDWC) is a cutting process similar to band machining, but it uses a small-diameter wire electrode instead of a saw. Like EDM, this process removes material by spark erosion as the wire electrode is fed through the workpiece, **Figure 31-8**. The wire electrode is fed from a spool over sapphire or diamond guides. A starter or threading hole is required. Some machines start their own holes and thread the wire electrode automatically. A steady stream of dielectric fluid cools the electrode and the work. The wire electrode is used only once because it becomes warped or distorted after just one pass through the work.

EDWC is well-suited for CNC applications and can produce dies, shaped carbide cutting tools, and punches in less than one-third of the time required by conventional methods. With EDWC, layers of sheet metal stacked up to 15″ (381 mm) thick can be gang-cut to produce a number of parts in one pass. EDWC is also useful for cutting intricate shapes that would be difficult to machine by conventional methods, **Figure 31-9**.



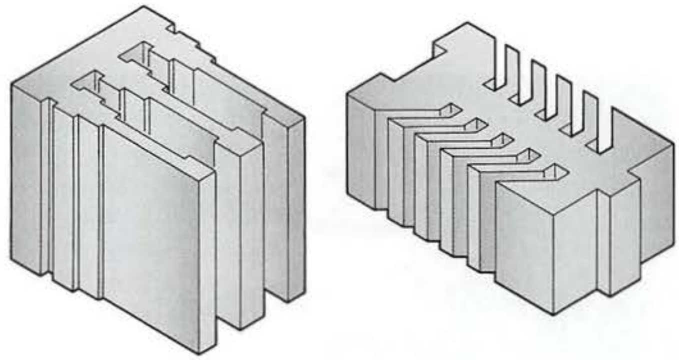
Goodheart-Willcox Publisher

Figure 31-7. An EDM electrode must be an exact reversal or mirror image of the cut to be made.



Goodheart-Willcox Publisher

Figure 31-8. Electrical discharge wire cutting (EDWC) differs from EDM in that cuts are made with a fine, moving wire electrode instead of a solid electrode. This technique is ideal for CNC operations.



Goodheart-Willcox Publisher

Figure 31-9. Examples of work produced using EDWC.

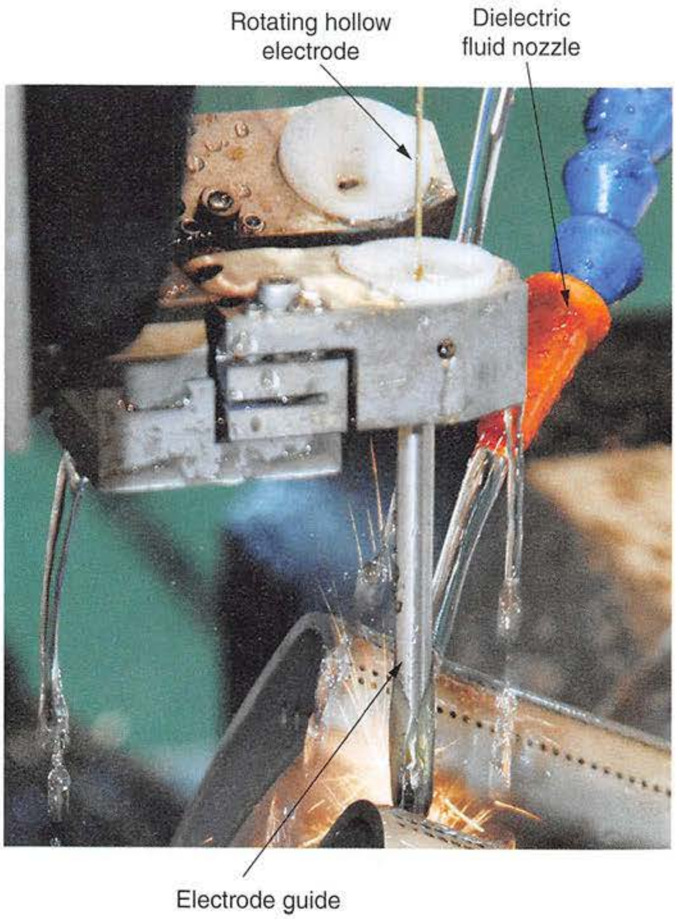
31.3 Small-Hole EDM Drilling

The ***small-hole EDM drilling*** process operates on the same basic principle as the EDM and EDWC processes. In this process, a spinning, hollow electrode is brought close to the workpiece. High-pressure dielectric fluid is pumped through the center of the electrode and floods the small gap between the workpiece and the electrode. When sufficient voltage is built up, an arc jumps the gap, eroding metal from the workpiece. In addition to the high-pressure dielectric fluid provided through the electrode, nozzles can spray additional dielectric fluid on the cut. The additional fluid helps to clear and cool the cut area. See **Figure 31-10**. Small-hole EDM drilling is often used to create the starter hole for EDWC.

Small-hole EDM drilling has certain advantages over other drilling processes.

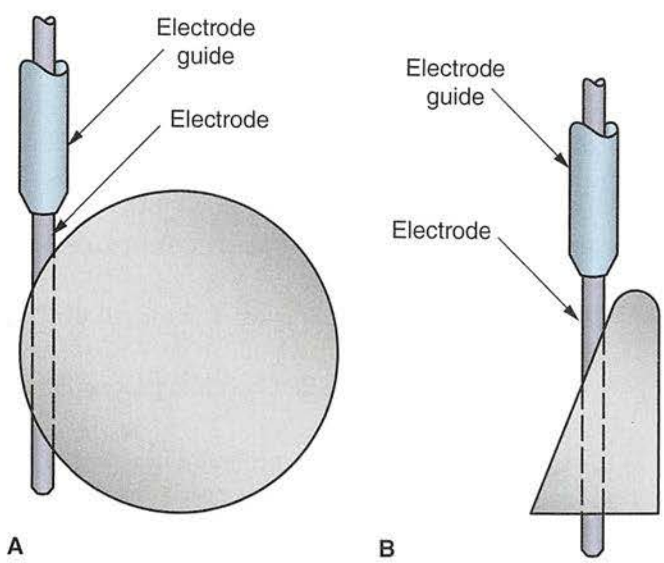
- It can drill holes in hardened metals.
- It can be used to accurately drill holes on highly curved or angled surfaces that would be difficult to drill by mechanical means, **Figure 31-11**, because the electrode never comes into physical contact with the workpiece.
- It can produce deep holes without the drift associated with mechanical drilling.
- There is no risk of breaking off a hardened drill bit inside the hole.
- The holes created by the EDM drilling process do not require deburring.

The small-hole EDM drilling process also has some drawbacks. The electrodes are consumed at nearly the same rate that the hole is drilled. For larger-diameter holes, the drilling rate is slower than can be achieved by mechanical drilling. A new or freshly dressed electrode must be used at the bottom of a blind hole, or the bottom of the hole will not be flat.



AMT—Association for Manufacturing Technology

Figure 31-10. Small-hole EDM drilling in progress.



Goodheart-Willcox Publisher

Figure 31-11. Because the electrode never comes into physical contact with the workpiece, small-hole EDM drilling can be used to drill holes in areas that would be problematic for mechanical drilling. A—Drilling a hole through the outer extents of a sphere. B—Drilling a hole through an angled surface.

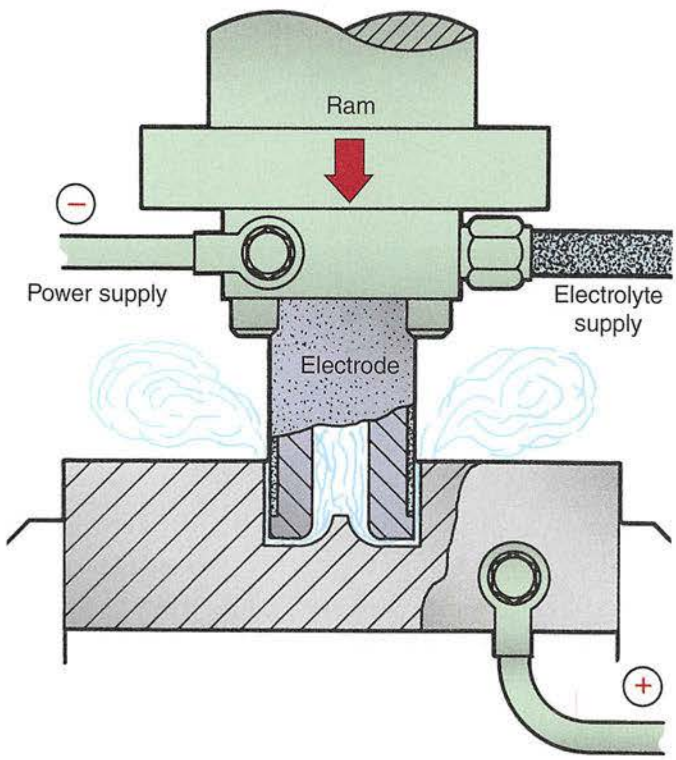
31.4 Electrochemical Machining (ECM)

Electrochemical machining (ECM) might be classified as electroplating in reverse. Like electroplating, the process requires DC electricity and a suitable electrolyte (an electrically conductive fluid). However, with ECM, the metal is removed from the work rather than deposited onto it, **Figure 31-12**.

In other ways, ECM is similar to EDM, but there are some significant differences. First, the fluid between the electrode and the workpiece is a conductive electrolyte rather than a dielectric fluid. As a result, there is no spark between the workpiece and the electrode. Metal is removed through electrolysis rather than through spark erosion.

The electrolyte for ECM is usually common salt (NaCl) mixed with water. A stream of electrolyte is pumped at high pressure through a gap between the positively (+) charged work and the negatively (–) charged tool (electrode). The current passing through the gap removes material from the work by electrolysis, duplicating the shape of the electrode tool as it advances into the metal. In some applications, tolerances as close as 0.0004″ (0.010 mm) can be maintained.

The work is not touched by the tool; therefore, no friction, heat, sparking, or tool wear occur. The machined surface is free of burrs and, in some instances, is highly polished. The operation of the machine is unique in that the only sound heard is the rush of liquid.



Goodheart-Willcox Publisher

Figure 31-12. The electrochemical machining (ECM) process.

31.4.1 Advantages of ECM

ECM offers many advantages:

- Metal is removed rapidly—up to 1 cubic inch per minute for every 10,000 amps of machining current.
- The kind of metal or its hardness does not affect the speed of material removal, with the exception of cast iron, which is machined by other processes.
- ECM is accurate. Difficult shapes can be machined easily.
- The machined metal is stress-free and will not warp or spring out of shape when removed from the machine.
- There is no tool wear.
- Several operations (milling, grinding, deburring, and polishing) often can be eliminated with ECM.

Advances are being made continually in ECM. The ability to produce highly complex shapes with simple tooling widens the range of application for this machining process.

31.4.2 Disadvantages of ECM

ECM also has some significant disadvantages:

- Equipment and tooling costs are high.
- The electrolyte is corrosive.
- Extremely high voltage must be applied to the workpiece.

Chapter Review

Summary

- Electrical discharge machining (EDM) can work metals that are difficult to work using conventional machining processes. It is capable of working tough, hard, fragile, or heat-sensitive metals to close tolerances.
- In the EDM process, a dielectric fluid fills the gap between the electrode and the workpiece. When voltage builds to a sufficient level, it creates an arc between the electrode and the workpiece. This arc erodes some metal from the workpiece.
- In electrical discharge wire cutting (EDWC), a small-diameter wire electrode removes material by spark erosion as it is fed through the workpiece.
- In small-hole EDM drilling, a spark is created between the end of a hollow, rotating electrode and the workpiece. As the electrode is fed into the workpiece, it bores a hole.
- Electrochemical machining is similar to electroplating, but in reverse. Unlike EDM processes, electrochemical machining removes material by electrolysis rather than spark erosion.

Review Questions

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

1. Unlike conventional machining processes, EDM does not produce ______ as metal is removed.
2. EDM stands for ______.
3. EDM can be used to work ______.
   A. metals that are too hard to be machined by conventional means
   B. nonconductive metals
   C. conductive and insulating materials
   D. All of the above.
   E. None of the above.
4. The ______ of an EDM machine maintains a very thin gap of about 0.001″ (0.025 mm) between the electrode and the work.
5. Explain the functions of the dielectric fluid used in EDM.
6. EDWC stands for ______.
7. How does EDWC differ from EDM?
8. Small-hole EDM drilling uses a ______, rotating electrode.
   A. nonconductive
   B. hollow
   C. flexible
   D. case-hardened
9. Which of the following statements regarding small-hole EDM drilling is true?
   A. A highly conductive electrolyte is used in the process.
   B. The electrode is in direct physical contact with the workpiece.
   C. The hole is created by arc erosion of the workpiece.
   D. All of the above.
   E. None of the above.
10. ECM stands for ______.
11. ECM is similar to ______ in reverse.
12. Metals to be machined by EDM and ECM must be able to conduct ______.
13. In ECM, metal is removed by ______.
14. In ECM, ______.
    A. the work is not touched by the tool
    B. there is no friction or heat generated
    C. there is no tool wear
    D. All of the above.
    E. None of the above.
15. What are five advantages that ECM offers?

CHAPTER **32**

Nontraditional Machining Techniques

Chapter Outline

Learning Objectives

After studying this chapter, you will be able to:

- Describe the chemical milling process and its advantages and disadvantages.
- Understand the purpose of hydrodynamic machining.
- Explain waterjet cutting and waterjet abrasive milling.
- Summarize the various ultrasonic machining processes.
- Explain electron beam machining.
- Describe the laser beam machining process.

Technical Terms

chemical blanking
chemical machining
chemical milling
electron beam machining (EBM)
etchant
hydrodynamic machining (HDM)
impact machining
laser
ultrasonic-assist machining
waterjet cutting

The metalworking industry is responsible for cutting, shaping, and fabricating both metals and nonmetals. As the use of existing materials has evolved and new materials have been developed, new machining techniques have been devised to keep pace. This chapter describes several of the new techniques that differ from the traditional chip-removal methods of the lathe, drill press, milling machine, grinder, and saw.

32.1 Chemical Machining

Chemical machining is a refinement of the process once used by photoengravers to prepare printing cuts and plates. Chemicals, usually in an aqueous (with water) solution, are used to etch away selected portions of a metal to produce an accurately contoured part. In general, chemical machining falls into two categories: chemical milling and chemical blanking.

32.1.1 Chemical Milling

Chemical milling, also called *chem-milling* or *contour etching*, is an accepted technique for machining metal to exacting tolerances through chemical action. The process makes it possible to remove metal selectively from relatively large surface areas, **Figure 32-1**. For example, chemical milling can be used to reduce the weight of sheet metal parts, which is critical to aerospace vehicle performance.

In the chem-milling process, masks (special coating materials) are applied to the areas of the part from which metal will not be removed. The prepared part is then immersed in an ***etchant*** (usually a strong alkaline solution). Where bare metal is exposed on the part, the resulting chemical action gradually erodes the metal. The areas that are masked do not react to the etching solution. The immersion time must be carefully controlled.

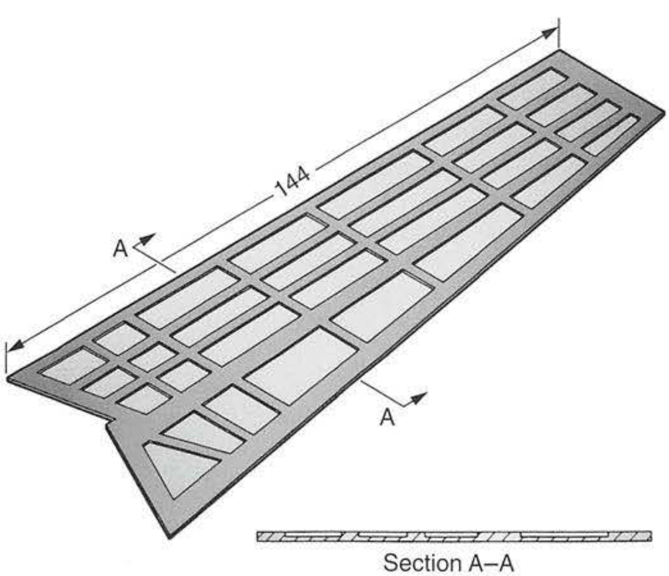
Chemical milling and conventional milling are complementary processes. Refinements in chemical milling make it possible to remove metal to form shapes or microscopic parts that would be difficult or impossible to achieve by conventional machining techniques, **Figure 32-2**.

Tapers and multiple-depth cuts can also be produced. Multiple-depth cuts are possible with chemical milling by masking shallower sections after they have been eroded to the desired depth. When the part is reimmersed in the etchant, additional metal is removed only from the unmasked areas. Tapers are produced by withdrawing metal from an etchant at a predetermined rate. See **Figure 32-3**.

Steps in Chemical Milling

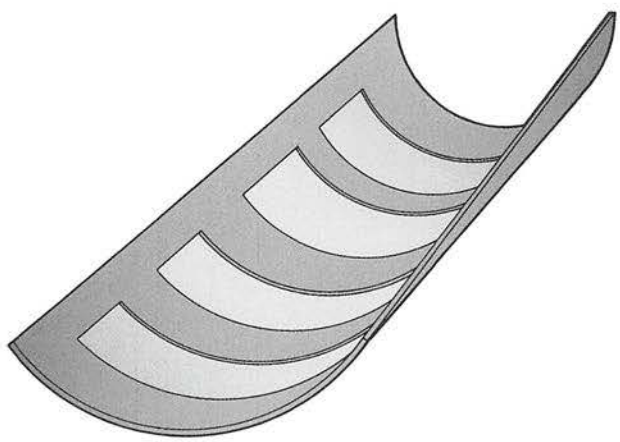
There are six major steps in the chemical milling process:

1. **Cleaning.** All grease and dirt that might affect the etching process are removed from the part.
2. **Masking.** The entire part is coated with a masking material, applied by brushing, dipping, spraying, or roller coating. The masked metal sheet is then baked to remove all solvents.



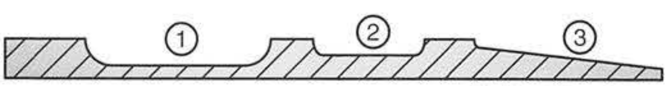
Goodheart-Willcox Publisher

Figure 32-1. Chemical milling is used to remove metal to close tolerances. This aircraft wing panel has been chemically reduced in sections where spars are not attached. This results in considerable weight reduction with no sacrifice of structural strength.



Goodheart-Willcox Publisher

Figure 32-2. With chemical milling, parts, such as the outer skin of an aircraft engine housing, can be milled after they are formed.



Goodheart-Willcox Publisher

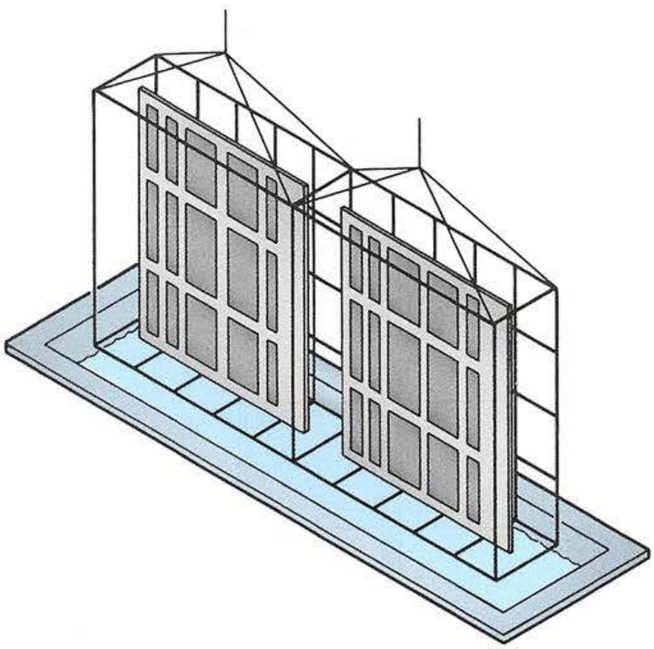
Figure 32-3. Surface 2 on this part is less deep than surface 1 because it was masked partway through the chemical milling operation. As a result, surface 1 was etched for a longer period of time. Surface 3 was created by dipping that end of the part in the etchant and withdrawing it gradually.

3. **Scribing and stripping.** A template is placed over the entire part. Then, the areas to be exposed are circumscribed, and the masking material is stripped away.
4. **Etching.** Parts are racked and lowered into an etchant for the milling operation. See **Figure 32-4**.
5. **Rinsing and solvent stripping.** After they are rinsed, the parts are lowered into a solvent tank, which releases the maskant bond. Maskant residue is stripped from the part.
6. **Inspection.** The accuracy of the chemical milling etch is measured with an ultrasonic thickness gage.

Advantages of Chemical Milling

Chemical milling offers many advantages:

- Tooling costs are low.
- Tolerances of ±0.003″ (0.08 mm) are possible on cuts up to 0.50″ (12.7 mm) deep.
- The size of the workpiece is limited only by the size of the immersion tank.
- Warping and distortion of formed sections is negligible.
- Contoured or shaped parts can be milled after they are formed, **Figure 32-5**.
- Many parts can be produced simultaneously.
- Unsupported pieces as thin as 0.015″ (0.38 mm) can be machined without danger of buckling.
- Both sides of the metal can be milled at the same time.
- Any metal, regardless of its state of heat treatment, can be machined chemically.
- No burrs are produced in the machined area.



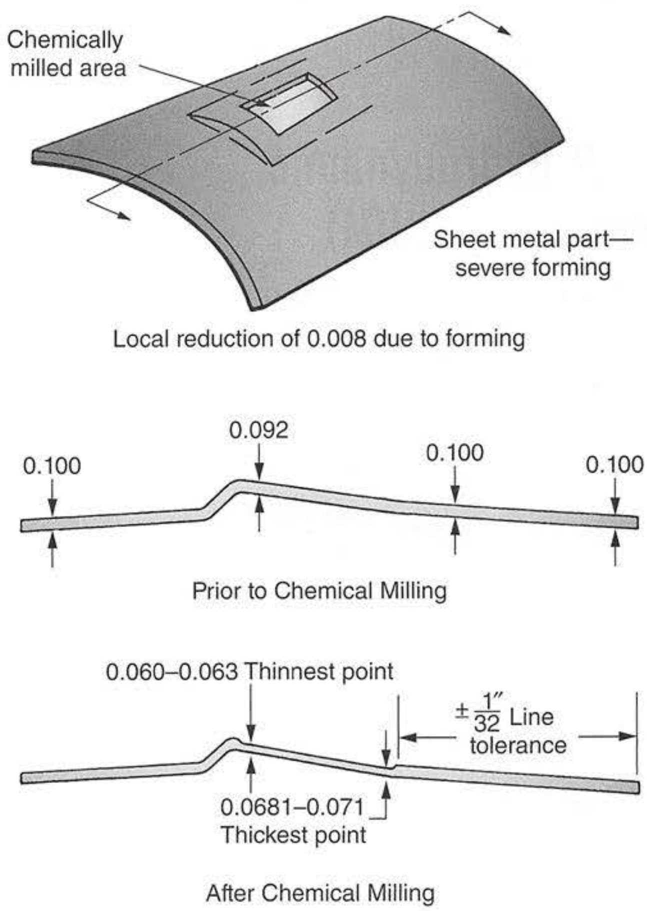
Goodheart-Willcox Publisher

Figure 32-4. Masked parts on a rack, ready to be lowered into a tank of etchant. The dark gray areas are unmasked.

Disadvantages of Chemical Milling

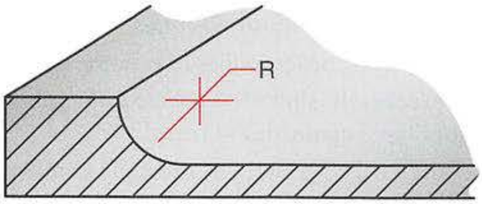
The chemical milling process does have some disadvantages:

- The process is slow. It takes considerable time to remove large quantities of metal.
- All surface imperfections must be removed before etching. Otherwise, these areas etch at a faster rate and are amplified on the finished surface.
- Chemical milling is not recommended for etching holes.
- Surface finishes on deep etches are not as fine as those on conventionally machined surfaces.
- Lateral dimensions are difficult to hold because the etchant works sideways as well as in depth. Typical lateral tolerances work on an etch factor of 3:1. This means that for every 0.003″ (0.08 mm) of etched depth, 0.001″ (0.03 mm) of undercut will occur. The top edges of the cavity will be sharp; however, the inside edges and corners will have a radius approximately equal to the cut depth, **Figure 32-6**.



Northrop-Grumman Corp.

Figure 32-5. This aircraft part was chemically milled after it was formed. Pieces even more severely formed than this part can be milled economically using this method.



Goodheart-Willcox Publisher

Figure 32-6. The inside edges of a chemically milled section have a radius equal to the depth of the etch.

32.1.2 Chemical Blanking

Chemical blanking, also called *chem-blanking, photoforming*, or *photoetching*, involves complete removal of metal from certain areas by chemical action. It is a variation of chemical milling. Chemical blanking is used by the aerospace and electronics industries to produce small, intricate, ultrathin parts. See **Figure 32-7**.

Metal foil as thin as 0.00008″ (0.002 mm) can be worked by chemical blanking. (By comparison, a sheet of copier paper is about 0.004″ or 0.1 mm thick.) This ultrathin metal is laminated to a plastic backing to protect it from damage during shipping.

This process is not recommended for metals thicker than 0.09″ (2.3 mm). However, almost any metal can be chemically blanked.

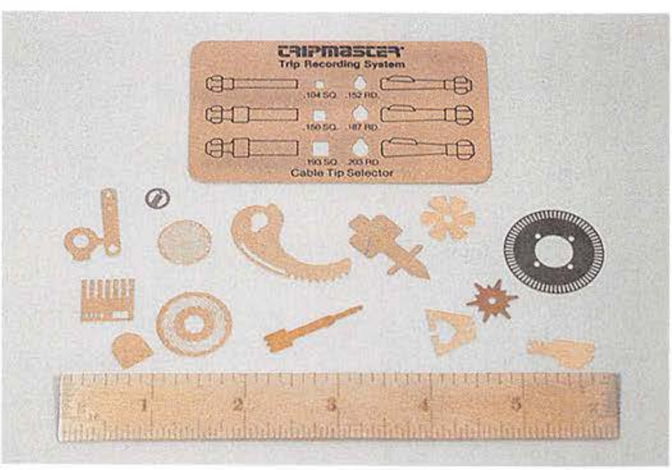
32.2 Hydrodynamic Machining (HDM)

Hydrodynamic machining (HDM), or ***waterjet cutting***, uses a high-velocity, high-pressure stream of water to cut through materials, **Figure 32-8**. It was developed to shape composites, which are layers of a tough, fabric-like material bonded together into three-dimensional shapes called *layups*. See **Figure 32-9**. A nontraditional cutting method was needed because the texture of composites quickly dulls conventional cutting tools. For shaping metals and hard nonmetallic materials, abrasives are added to the water jet, **Figure 32-10**. However, waterjet cutting can be accomplished with or without abrasives.

Depending on the material being cut, tolerances can be held to ±0.004″ (0.1 mm). No heat is generated that could damage the material being cut, nor is airborne dust produced. Instead of going into the air, particulates (fragments) are carried away by the water jet.

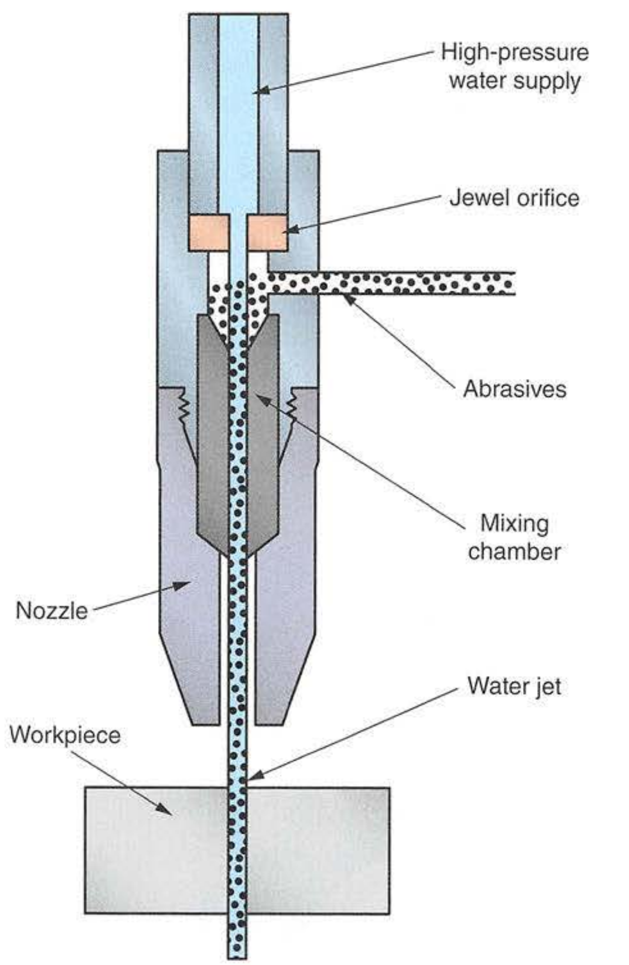
SAFETY NOTE

The speed of the water jet leaving the nozzle is approximately three times the speed of sound, which can cause serious injury. Keep your hands clear of the work area.



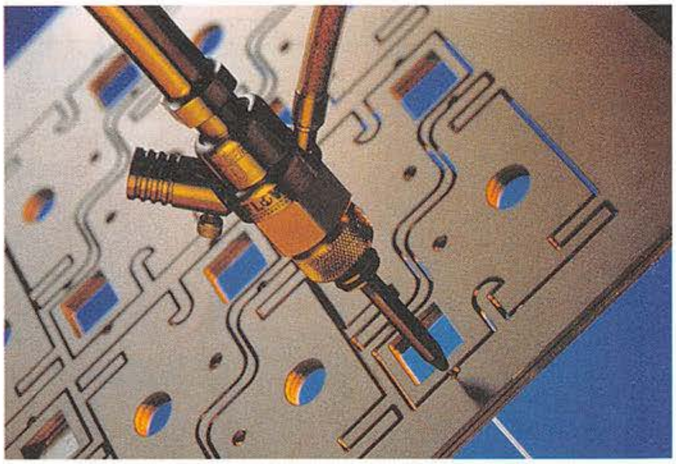
Microphoto, Inc.

Figure 32-7. Made by chemical blanking, these electronic parts range in thickness from 0.0001″ to 0.002″ (0.0025 mm to 0.05 mm).



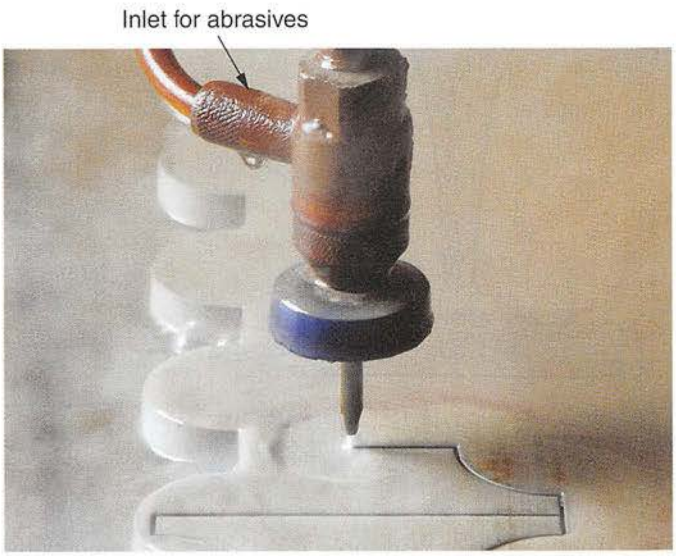
Goodheart-Willcox Publisher

Figure 32-8. Waterjet cutting head. Water is forced through an orifice made of a very hard material, such as diamond, ruby, or sapphire, under extremely high pressure. Abrasives are added to the high-pressure water stream by induction.



Flow International Corp.

Figure 32-9. A high-powered stream of water cuts intricate shapes in tough composite materials.



xtrekx/Shutterstock.com

Figure 32-10. Abrasives are added to cut hard materials, such as this stainless steel plate.

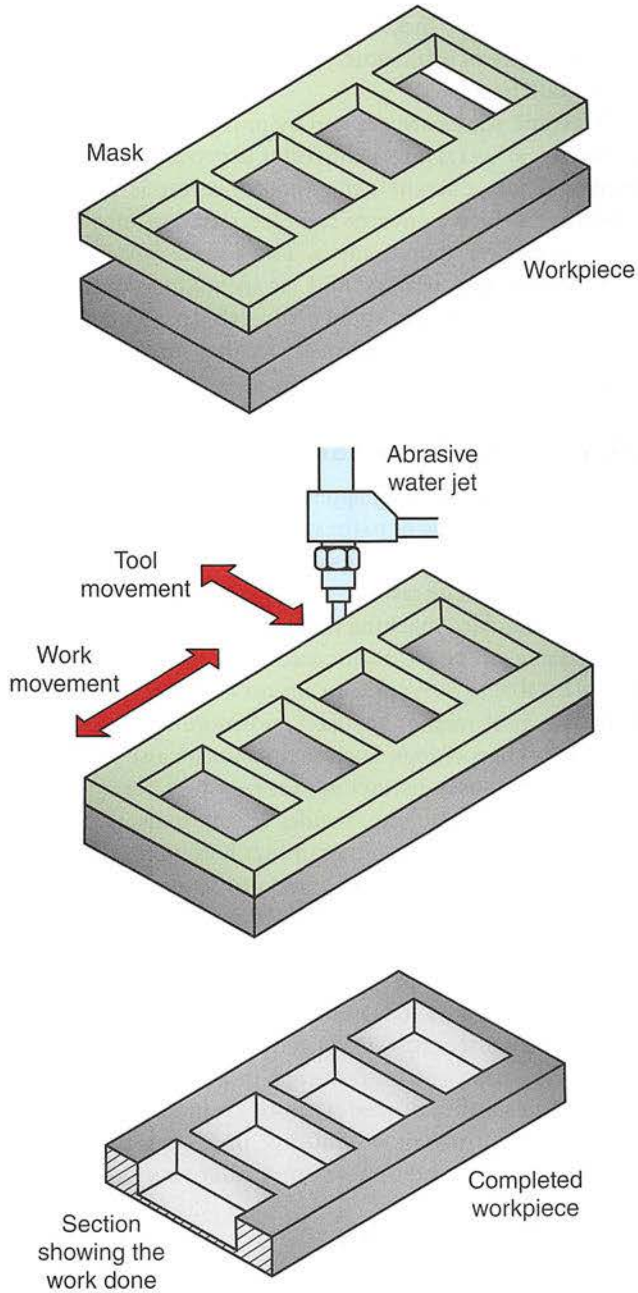
32.3 Waterjet Abrasive Milling

Originally, waterjet machining was used only in applications in which the jet passed all the way through the material being cut. An abrasive was added when metals and other hard materials were to be cut. Cut depth was not a consideration. Techniques and processes have since been developed to control cut depth.

Waterjet abrasive milling is similar to chemical milling. However, waterjet abrasive milling does not require expensive disposal of environmentally hostile by-products (etchants and material residue). A mask made from material that is resistant to the abrasive action of the water jet is placed over the workpiece. The mask, rather than variation in a tool path, controls the geometry of the area being machined. See **Figure 32-11**.

Each pass of the workpiece under the abrasive water jet removes a small quantity of material. Workpiece speed determines the amount of material removed on each pass.

Many difficult-to-machine aerospace materials with complex shaped areas can be machined by this technique. Tolerances of ±0.002″ (0.05 mm) can be maintained.



Goodheart-Willcox Publisher

Figure 32-11. The waterjet milling operation. A mask composed of an abrasive-resistant material can be used several times before it is worn away and must be replaced.

32.4 Ultrasonic Machining

The term *ultrasonic* is used to describe sound waves at frequencies higher than the human ear can detect. The science of ultrasonics has found applications in many areas, including machining, welding, quality control, and cleaning. Considerable research is being done to develop new uses and to improve existing techniques.

The average person can hear sounds that vibrate between 20 and 20,000 hertz (cycles per second). Below 20 hertz, sound waves are *infrasonic*. Above 20,000 hertz, sound waves are *ultrasonic*. Industrial ultrasonic machining applications make use of ultrasonic sound waves with frequencies up to 100,000 hertz.

Ultrasonic waves are created by passing an electric current (usually 60 hertz) into a converter to produce the desired frequency. Then, current from the converter is passed to a transducer, which converts the electricity to mechanical motion, producing sound waves. The sound waves may be used in conjunction with a fluid (as in quality control and cleaning applications) or applied directly to a cutting tool or metal as it is being machined, welded, or formed. See **Figure 32-12**.

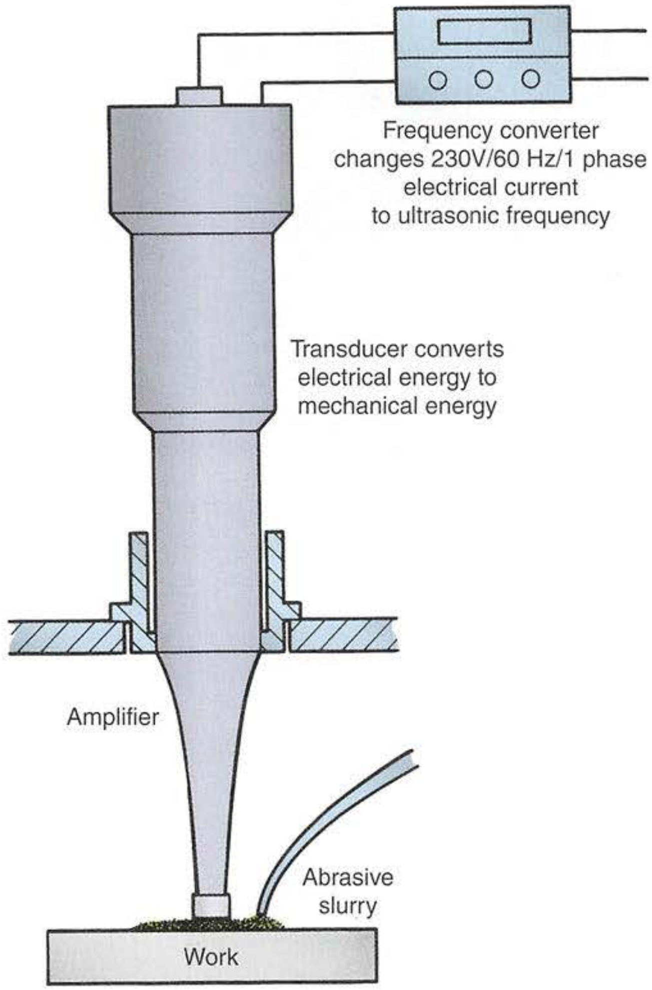


Figure 32-12. Ultrasonic machining setup.

32.4.1 Ultrasonic-Assist Machining

Ultrasonic-assist machining applies sound waves to a conventional tool as it is cutting or to the metal as it is being cut. In this way, the ultrasonic sound waves assist the conventional metal cutting processes. In ultrasonic-assist machining, a transducer is fitted to a standard machine tool to make the tool vibrate at a high frequency. The ultrasonic assist can reduce the needed tool force by 10% to 50% and can almost completely eliminate tool chatter. Tool wear is reduced and more cutting can be done between sharpenings. Surface finishes are also improved. Chatter reduction is a distinct advantage when boring operations require the use of long, slender boring tools. Ultrasonic assist can be used in drilling, reaming, honing, milling, and EDM (electrical discharge machining) processes.

Ultrasonic assist is also useful in grinding operations. Ultrasonic waves are passed through the grinding wheel. Particles and chips that normally become embedded in the wheel are vibrated loose and washed away by the coolant. Grinding temperatures are reduced, and wheel life is extended. Material can be removed more rapidly, and the surface finishes are improved without an increase in power consumption. Ultrasonic vibrations imparted to grinder coolant have been found to produce similar results.

32.4.2 Impact Machining

Impact machining, also called *slurry machining*, is a machining process that removes metal by using ultrasonic frequencies and a tool that forces abrasives against the work. With the exception of diamond tools, it is the only commercially feasible way to machine extremely hard, brittle, and fragile materials. In fact, the technique works best on hard, brittle materials (glass, quartz, silicon, and carbides). It is ineffective on soft materials, such as aluminum and copper.

Machining is done by a shaped cutting tool oscillating at about 25,000 times per second. This rapid oscillation pounds a slurry of fine, abrasive particles against the work. The tool stroke, at the vibrating end, is only 0.003″ (0.076 mm), **Figure 32-13**.

Ultrasonic vibrations directly from the transducer do not produce enough motion to produce the required tool movement. A solid, funnel-shaped horn amplifies and transmits the vibrations. The tool does not touch the work and no heat is generated, so there is no risk of heat-related distortion in the work. Microscopic portions of the work are chipped away to produce the desired shape. The machined section is a mirror image of the cutting tool, **Figure 32-14**.

CAREER CONNECTION

Mechanical Engineer

What does a mechanical engineer do?

Mechanical engineering is the broadest branch of engineering, and mechanical engineers work across many industries. These engineers study problems in order to solve them with mechanical and thermal devices. Batteries, escalators, engines, electric generators, and elevators have all been designed and developed by mechanical engineers.

What education and skills are needed to be a mechanical engineer?

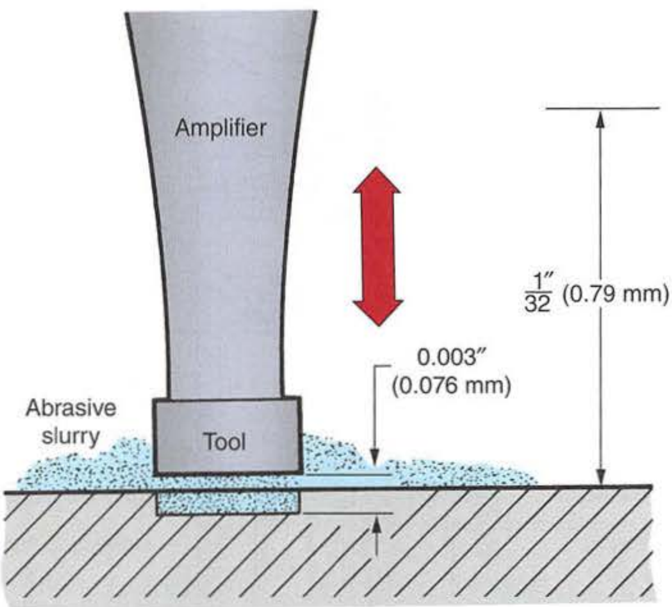
To become a mechanical engineer, students should study math, science, physics, and computer science. Like many engineers, mechanical engineers use computers extensively to model and test different designs.

Students should major in mechanical engineering or mechanical engineering technology from an ABET-accredited program. Advanced degrees are desirable for advancement and mobility within this field, and some universities offer five-year programs for a combined bachelor's/master's degree.

What is it like to be a mechanical engineer?

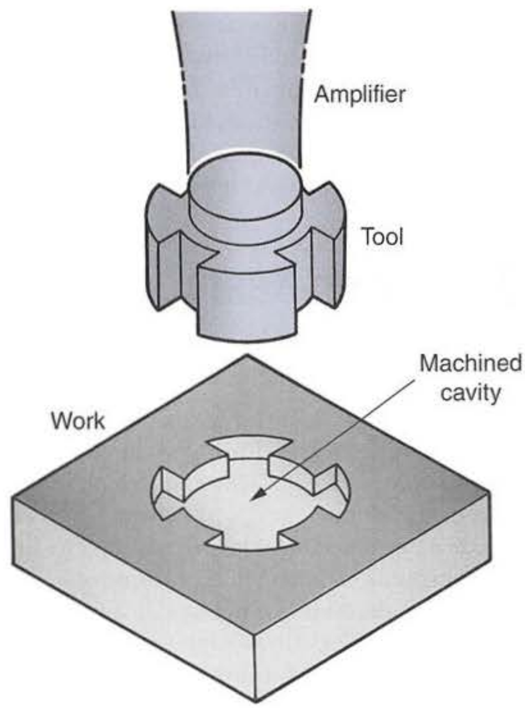
Mechanical engineers are a vital part of the design, testing, and manufacturing process. They work together with other engineers, technicians, and developers. Though they primarily work in offices, mechanical engineers may travel to worksites to give special attention to problems as they arise.

Mechanical engineers work in many industries, but most are employed by research and development teams in the physical, engineering, and life sciences. According to the *Occupational Outlook Handbook*, wages for mechanical engineers average $83,500 per year.



Goodheart-Willcox Publisher

Figure 32-13. Tool motion in ultrasonic (impact) machining is slight, only 0.003″ (0.076 mm). The 1/32″ (0.79 mm) measurement is used to indicate scale.



Goodheart-Willcox Publisher

Figure 32-14. The machined section is a mirror image of the tool.

Industrial applications of impact machining include the following:

- Slicing and cutting germanium and silicon wafers into tiny chips for transistors, diodes, and rectifiers for the electronics industry.
- Machining complex shapes in nonconductive and semiconductive materials that cannot be satisfactorily handled by EDM and ECM.
- Shaping virtually unmachinable space-age materials.

The disadvantages of impact machining are that it is slow, surface finish depends on the size of the abrasive grit used, and maximum cut depth is only about 1″ (25 mm). On the other hand, tolerances of 0.001″ (0.025 mm) can be maintained on hole size and geometry in most materials. Equipment cost is moderate. Machine operation does not require special skills, so the training period for machine operators is short.

32.4.3 Other Ultrasonic Applications

Industrial applications for ultrasonics vary. One common use is to improve the cleaning power of chemical solvents. As the ultrasonic waves pass through the cleaning solution, microscopic vapor pockets are created in the fluid. These "bubbles" (high-vacuum areas) form and collapse about 20,000 times per second. This creates local pressure as high as 10,000 psi and heat. The bubbles smash against the work and literally tear away the dirt, oil, grease, chips, flux, and other contaminants on the surface. The cleaning action is so thorough that the technique is used to decontaminate work that has been exposed to radioactive solutions and gases.

Other uses include automated, nondestructive ultrasonic testing. Sound waves are also used in the precision welding of dissimilar materials. For example, ultrasonic waves can weld aluminum wires to glass fibers on some electronic circuits.

32.5 Electron Beam Machining (EBM)

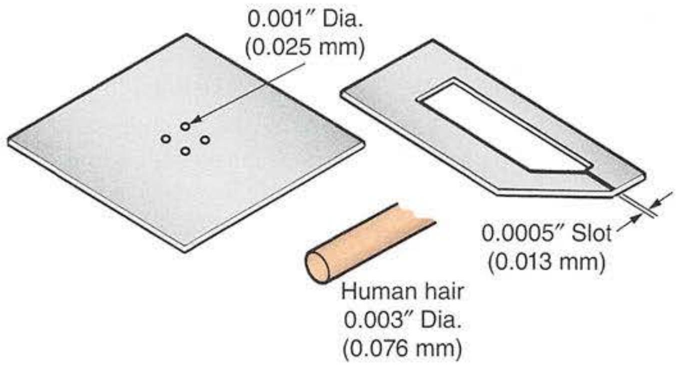
The electron beam microcutter-welder uses a high-energy, highly focused beam of electrons to weld or cut materials. ***Electron beam machining (EBM)*** was developed to meet specific needs of the atomic energy, electronics, and aerospace industries. In some respects, it is the most precise and versatile of the nontraditional machining techniques.

Because the electron beam can cut any known metal or nonmetal that can exist in a high vacuum, its microcutting capabilities are almost unlimited. The development of refined focusing systems has made it possible to control the cutting action with a high degree of precision.

Electron beam cutting action can be controlled so precisely that it is possible to drill holes as small as 0.0002″ (0.005 mm) in diameter and mill slots having widths of 0.0005″ (0.0127 mm). The finish of the completed work is similar to a very fine machined edge. See **Figure 32-15**.

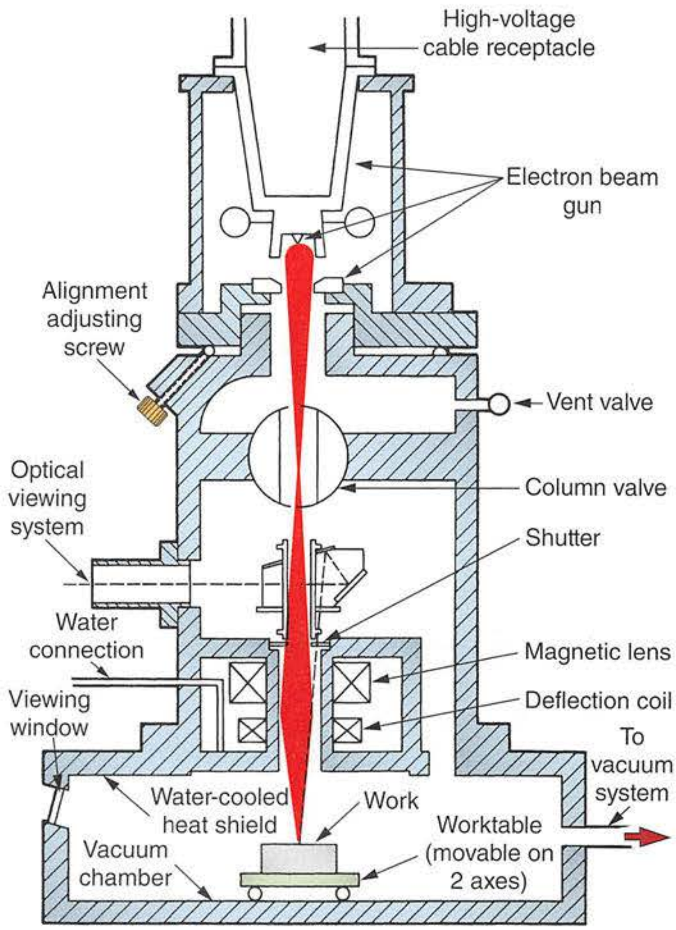
The electron beam machine is basically a source of thermal energy, **Figure 32-16**. The beam of electrons is focused to a very sharp point by the magnetic lens. The workpiece is placed on a worktable that can be moved under the electron beam.

Cutting is achieved by alternately heating and cooling the area to be cut. The heating and cooling must be controlled carefully so the material at the point of focus is heated



Goodheart-Willcox Publisher

Figure 32-15. Note the size of the work done using the electron beam technique. The parts and the shaft of human hair are drawn to the same scale.



Goodheart-Willcox Publisher

Figure 32-16. Cross-sectional view of an electron beam microcutter-welder.

to a temperature high enough to vaporize it, yet not enough to cause the surrounding area to melt. This is accomplished by pulsing the beam. The beam is on for only a few milliseconds and is off for a considerably longer period of time. The focal point of the beam can reach a temperature of 12,000°F (6649°C), but because the beam is on for only a short period of time, very little of the heat is transferred to the metal surrounding the cut.

Cut geometry (the shape of the cut) is controlled by moving the worktable in the vacuum chamber and by using the deflection coil to bend the beam of electrons to the desired cutting path. Initial hole diameter or cut width is controlled by the amount of power applied and the duration of the cutting time. See **Figure 32-17**.

32.6 Laser Beam Machining

The term ***laser*** is an acronym for Light Amplification by Stimulated Emission of Radiation. The laser produces a narrow, intense beam of light that can be focused optically onto an area only a few microns in diameter. See **Figure 32-18**.

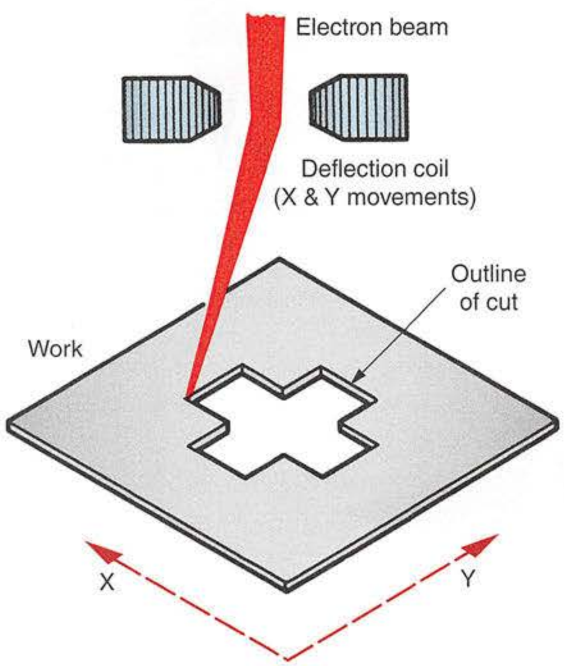
Depending on the initial energy source used to activate the laser, it is possible to instantaneously create temperatures up to 75,000°F (41,650°C) at the point of focus. This is almost seven times the average temperature of the sun. No known material can withstand such heat.

There has been a dramatic increase in the use of lasers in parts manufacturing. Lasers are used for cutting, drilling, slotting, scribing, heat-treating, and welding. See **Figure 32-19**.

The energy output of a laser is usually not continuous. It lasts only a fraction of a second. When used for machining and cutting, it operates at 1 to 5 cycles per second. The cycle can be controlled manually or electronically. The laser operates on the principle described in **Figure 32-20**.

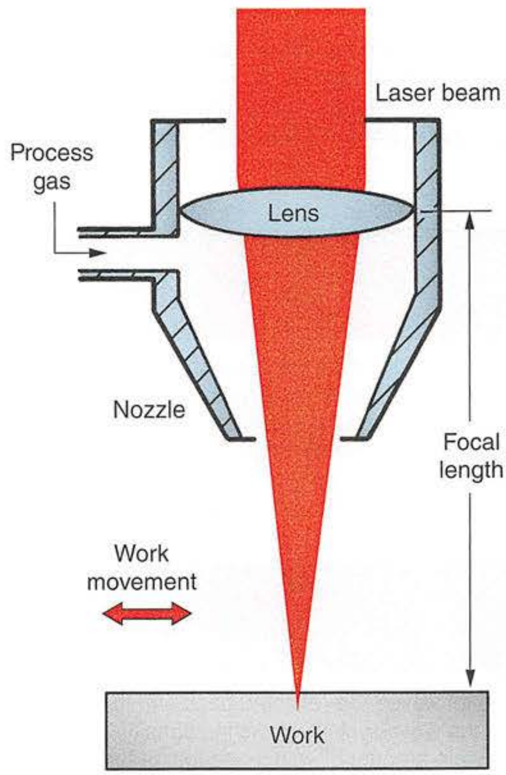
Some laser units can operate in either continuous or pulsed modes. With most materials, the edge quality of the cut is better with a continuous beam. Cutting speed is also much greater. The disadvantage of pulsing is that starting, stopping, and turning corners concentrates heat in localized areas on the work. This overheating causes the part to burn away from the programmed path, affecting cut quality and dimensional accuracy.

In the past, cutting aluminum, stainless steel, and titanium parts to shape using a laser required a time-consuming secondary operation to remove the oxide and dross that formed on the edge and surface of the cut. Titanium had a tendency to become brittle due to the oxygen that was absorbed during the cutting operation. These problems are solved by flooding the work surface with argon or nitrogen during the cutting operation. Since no oxygen can contaminate the metal, there is no oxide formation or oxygen absorption. Parts can be welded immediately after being cut because they require no cleaning.



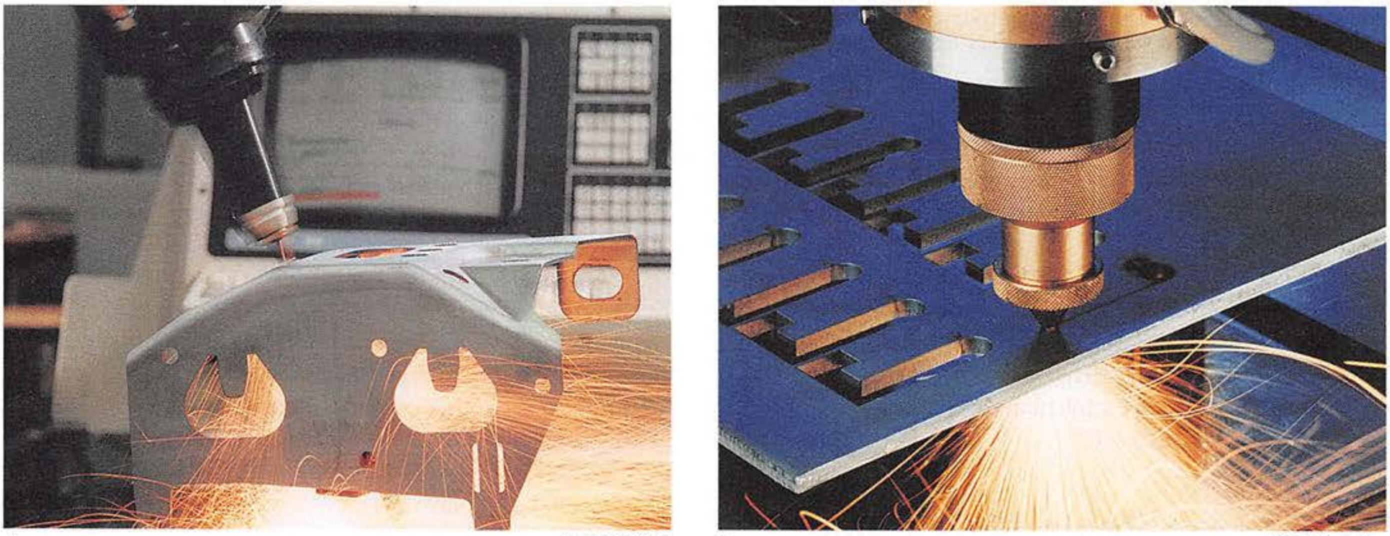
Goodheart-Willcox Publisher

Figure 32-17. The path of the cut is controlled by deflecting the electron beam or by moving the worktable. Cutting must be done in a vacuum.



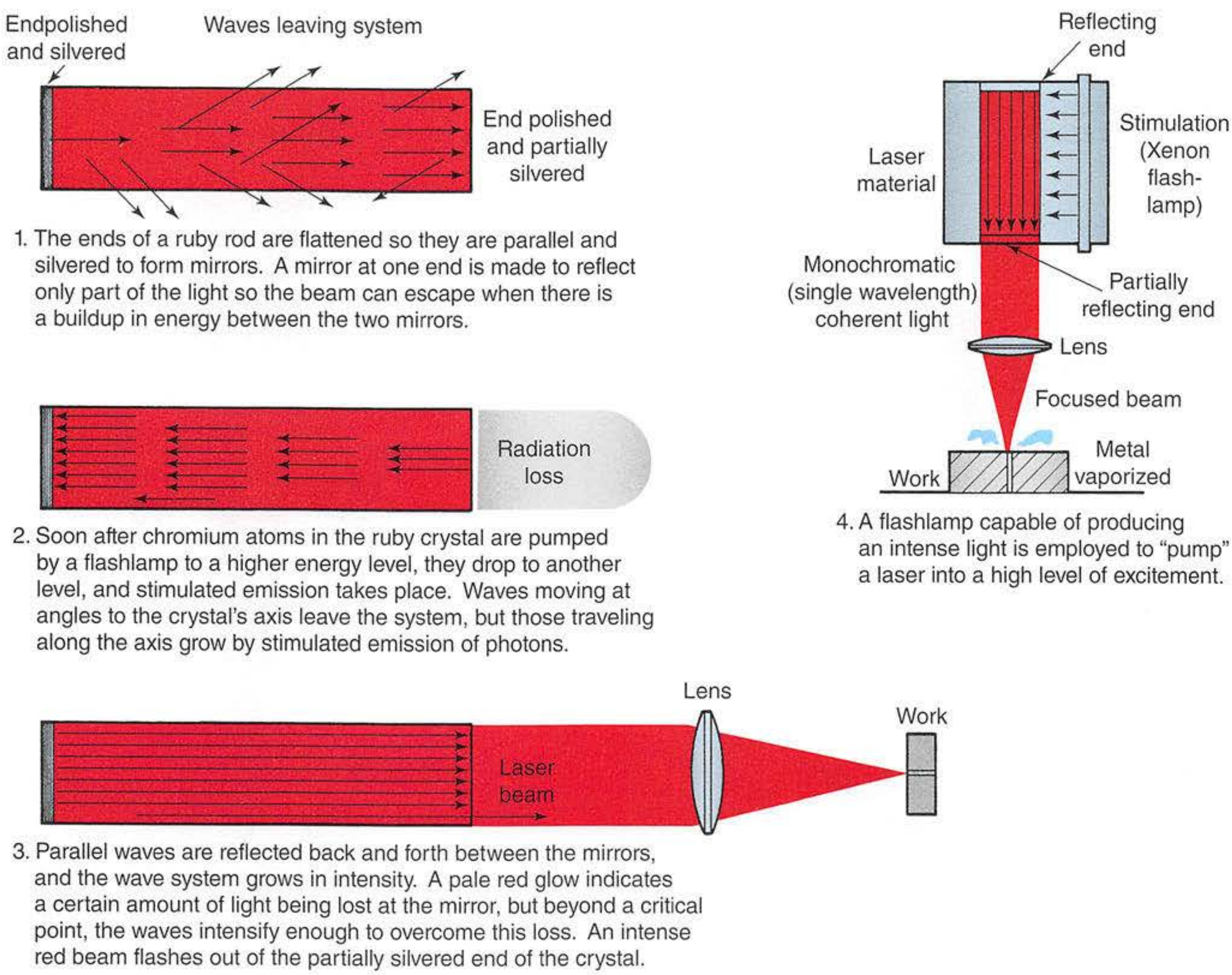
Goodheart-Willcox Publisher

Figure 32-18. A laser produces a narrow, intense beam of light. The beam can be concentrated to a point as small as 0.0002″ (0.05 mm) diameter.



A Lumonics Corp. B Rofin-Sinar, Inc.

Figure 32-19. Laser cutting. A—When used with CNC, a laser is capable of cutting 3D workpieces. B—A CNC CO_2 laser cutting slots in a stainless steel plate. The high-velocity gas jet aids in material removal by blowing out molten metal through the backside of the work. It also protects the lens from spatter ejected from the cut zone, especially during the piercing operation.



Goodheart-Willcox Publisher

Figure 32-20. Operation of a laser.

Chapter Review

Summary

- In chemical machining, chemicals are used to etch away selected portions of the metal to produce an accurately contoured part. Chemical machining falls into two categories: chemical milling and chemical blanking.
- Chemical milling is used to remove metal selectively from relatively large surface areas. Areas that are not to have metal removed are masked before the part is immersed in the etchant. Areas can be masked off in stages to produce multiple-depth cuts.
- The steps in chemical milling are cleaning the part, masking the entire part, scribing and stripping the mask away from the areas to be etched, etching, rinsing and solvent stripping, and inspection.
- Chemical blanking, or photoetching, is a process that uses chemicals to cut parts out of ultrathin materials.
- Hydrodynamic machining, or waterjet cutting, is a process that uses a high-velocity stream of water to cut through materials. Abrasives can be added to the water stream in order to cut through very hard materials.
- Waterjet abrasive milling works on the same principle as waterjet cutting. However, as with chemical milling, a mask is applied to protect certain areas of the workpiece. The water jet and the workpiece are moved so that the jet does not cut all the way through the material.
- Ultrasonic waves can be used to enhance the conventional cutting process. This is known as *ultrasonic-assist machining.* Ultrasonic waves can also be used with an abrasive slurry to wear away metal from the workpiece. This is known as *impact machining.*
- Electron beam machining focuses a beam of electrons to cut through materials. Because the electron beam is pulsed on and off, very little heat transfers from the cut into the surrounding metal.
- Laser beam machining uses the intense heat of a tightly focused beam of light to cut through materials. The cut area can be flooded with an inert gas to protect the metal being cut from oxidation and oxygen absorption.

Review Questions

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

1. Chemical milling is also known as chem-milling or ______.
2. Chemical machining falls into two categories. Briefly describe each of them.
3. During chemical milling, a mask protects the portions of a workpiece that are ______.
   A. made from nonferrous metal
   B. to be etched
   C. to be cleaned
   D. All of the above.
   E. None of the above.
4. List the six major steps in chemical milling, in order.
5. Briefly describe hydrodynamic machining (HDM).
6. What is the major difference between waterjet abrasive milling and chemical milling?
7. Sound waves below 20 hertz are called ______.
8. Sound waves above 20,000 hertz are called ______.
9. In impact machining, a tool forces ______ against the work to do the cutting.
10. Impact machining is one of the very few commercially feasible methods for machining which types of materials?
    A. Hard.
    B. Brittle.
    C. Fragile.
    D. All of the above.
    E. None of the above.
11. What are three disadvantages of impact machining?
12. With impact machining, tolerances of ______ can be maintained on hole size and geometry in most materials.
13. Holes as small as ______ in diameter can be drilled using electron beam machining.
14. The electron beam machine is basically a source of what type of energy?
    A. Thermal.
    B. Sonic.
    C. Fluid.
    D. All of the above.
    E. None of the above.
15. List two methods used to control the shape of the cut with EBM.
16. What does the term *laser* stand for?

CHAPTER **33**

Other Processes

Chapter Outline

33.1 Machining Plastics
- **33.1.1** Nylon
- **33.1.2** Delrin
- **33.1.3** Teflon
- **33.1.4** Acrylics
- **33.1.5** Laminated Plastics

33.2 Machining Ceramics
- **33.2.1** Determining Ceramic Hardness
- **33.2.2** Cutting and Forming Ceramics

33.3 Chipless Machining
- **33.3.1** Chipless Machining Processes
- **33.3.2** Intraform Machining

33.4 Powder Metallurgy
- **33.4.1** Powder Metallurgy Applications
- **33.4.2** Powder Metallurgy Process
- **33.4.3** Powder Metallurgy Costs

33.5 High-Energy-Rate Forming (HERF)
- **33.5.1** Types of HERF Operations
- **33.5.2** Explosive Forming
- **33.5.3** Electrohydraulic Forming
- **33.5.4** Magnetic Forming
- **33.5.5** Pneumatic-Mechanical Forming

33.6 Cryogenics
- **33.6.1** Cryogenic Applications
- **33.6.2** Cryogenic Treatment of Cutting Tools

Learning Objectives

After studying this chapter, you will be able to:

- Discuss the general machining characteristics of various plastics.
- Explain the determining factors for machinability of ceramic materials.
- Describe the five basic operations of chipless machining and their variations.
- Describe how powder metallurgy parts are produced.
- Compare the advantages and disadvantages of various HERF techniques.
- Explain how the science of cryogenics is used in industry, and list some applications.

Technical Terms

briquetting
chipless machining
cryogenic
electrohydraulic forming
explosive forming
high-energy-rate forming (HERF)
magnetic forming
Mohs hardness scale
powder metallurgy (P/M)
sintering

New materials and new processes are continually being introduced in the field of machining and its related areas. In addition to the nontraditional machining processes described in Chapter 32, processes such as chipless machining, high-energy-rate forming, and cryogenic treatment are being used today. The machining of various types of plastics and the fabrication of parts from powdered metals have become important. This chapter provides a basic introduction to these technologies.

33.1 Machining Plastics

Plastics are being machined in increasingly larger quantities each year, **Figure 33-1**. For this reason, it is important that the machinist be familiar with the machining characteristics of the common plastics, such as nylon, Teflon™, Delrin™, acrylics, and laminated plastics.

Special care must be taken when working with plastics. The dust and fumes given off by some plastics may be irritating to the skin, eyes, and respiratory system. Other plastics have fillers, such as glass fibers, that can be harmful to your health. Be sure you are aware of the safety precautions that must be observed before you attempt to machine any plastic. Always check the material's safety data sheet (SDS) before performing any machining operations.

33.1.1 Nylon

Nylon is the name for a group of polyamide resins. Nylon is tough and exhibits high tensile (pull), impact, and flexural (bending) strengths. This material is highly resistant to abrasion and is not affected by most common chemicals, greases, and solvents.

Chick Machine Tool, Inc.

Figure 33-1. More and more plastics are being machined. The fixtures shown here hold both plastic and metal parts for machining. CNC programs direct machine tools to change cutting speeds and feeds when different materials come into position for machining.

This plastic is excellent for bearings, gears, cams, and rollers. Another application for parts machined from nylon is in the preparation of experimental components. Research and development projects also make considerable use of machined nylon.

General Machining Precautions for Nylon

Most types of nylon can be machined using the same techniques used to machine soft brass. The use of coolants allows higher cutting speeds, but coolants are not necessary to produce good-quality machined surfaces. When used, coolants should be of the soluble-oil type.

Since nylon is not as rigid as metal, it must be well supported during machining operations. Otherwise, deflection of the unsupported stock will result in distortions and inaccurate dimensions. To ensure accuracy, bring parts machined from nylon to room temperature before checking dimensions.

Turning Nylon

Nylon can be turned on a standard metalworking lathe. Although tool bits sharpened to machine soft brass are satisfactory, best results are obtained using the tool bit shapes shown in **Figure 33-2**. Tools must be kept very sharp. The best finish on nylon is obtained with a high speed and fine feed.

Milling Nylon

Conventional milling cutters, providing they are kept very sharp, can be used for milling nylon. Climb milling minimizes the formation of burrs. Surface speeds in excess of 100 fpm (30 mpm) can be used.

Vertical milling is practical using fly cutters or two-lip end mills. Cutters must be kept very sharp to prevent plastic from melting or becoming gummy. Feeds of 1″ (25 mm) per minute or higher have proven satisfactory. Smoother surface finishes can be achieved using lighter feeds.

Drilling Nylon

Since drilling produces considerable heat, it requires extra care. Standard twist drills, when sharpened as shown in **Figure 33-3**, produce acceptable results. However, best results are obtained by using drills designed specifically for plastics. Drills for plastics have flutes that are highly polished and have a long lead.

Use heavy feeds to prevent the drill from scraping the plastic rather than cutting it. Scraping results in excessive heat, which melts the plastic. If a coolant is not used, the drill must be withdrawn from the hole frequently to clean out chips and prevent overheating. If the drill is kept cool, holes are drilled to size.

Nylon can be reamed using an expansion reamer that is adjusted to a few thousandths of an inch oversize. Holes

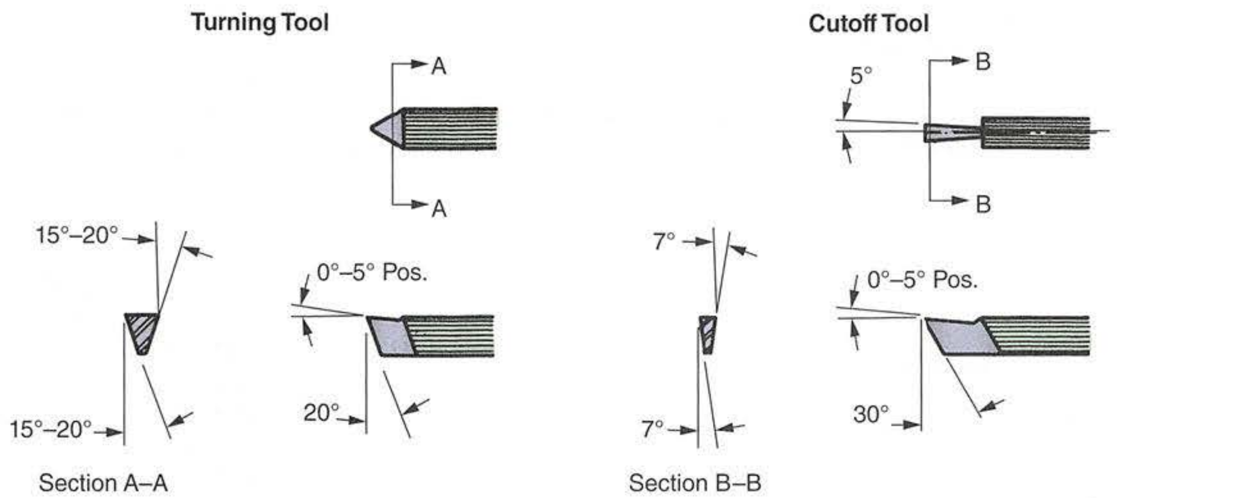


Figure 33-2. Typical cutting tools for machining nylon on a lathe.

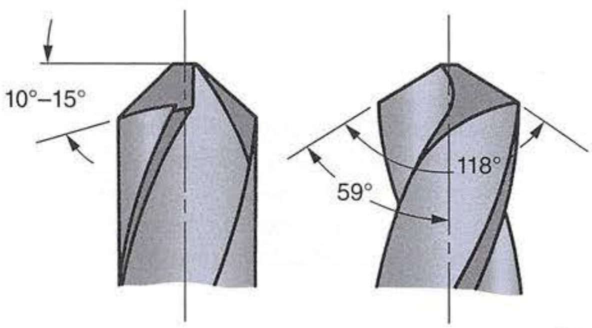


Figure 33-3. Drill point recommended for drilling nylon.

finished with solid reamers tend to be undersize because the material partially distorts, then returns to its original shape once the reamer is removed.

Threading and Tapping Nylon

Nylon can be threaded and tapped using conventional equipment. However, before being tapped, the hole should be chamfered to reduce the chance of the first few threads tearing. Production tapping requires a tap that is 0.005″ (0.127 mm) oversize, unless a self-locking thread is desired. The tap can be made oversize by chrome plating.

Nylon can be threaded on a lathe using a single-point cutting tool and the same procedures recommended for metal. However, the cutting tool must be kept very sharp and have plenty of clearance. Because of nylon's resilience, the finish cut should not be less than 0.005″ (0.127 mm). Support long work with a ball bearing follower rest.

Sawing Nylon

For sawing nylon and other plastics, good results can be obtained using a band saw. A band saw blade quickly dissipates the heat. Dry cutting is best accomplished with a skip-tooth metal-cutting blade that has 4 to 6 teeth per inch. However, the blade must be sharp to prevent the nylon from becoming gummy. If gumming occurs, the blade usually freezes in the cut.

Hollow-ground circular saw blades are also used extensively in cutting plastics. Blades with a slight set are recommended for cutting multiple layers of thin plastic or cutting thicknesses greater than 3/4″ (19 mm).

Annealing Nylon

Like metal, machined nylon parts require annealing to prevent dimensional changes. Annealing of plastics should be carried out in the absence of air, preferably by immersion in a suitable liquid. High-temperature boiling hydrocarbons, such as waxes and oils, are recommended for annealing nylon. A temperature of 300°F (149°C) is often used for general annealing.

An annealing time of 15 minutes per 0.125″ (3.175 mm) of thickness is normally required. Allow the part to cool slowly in a draft-free area. Placing the heated piece in a cardboard container is a simple way to ensure slow, even cooling.

33.1.2 Delrin

Parts manufactured from Delrin™ (the DuPont trade name for acetal resin) have an unusual combination of physical properties that bridge the gap between metals and plastics. These properties include excellent dimensional stability, high strength, and rigidity. This plastic is used extensively for parts in business machines (gears, cams, bearings, and printing wheels). Delrin is also replacing brass and zinc for many applications in the automotive and plumbing industries. The material has low friction, and thus requires a minimum use of lubricants. It is very quiet in operation.

In machining characteristics, Delrin is very similar to nylon. Recommended machining, cutting, and finishing operations are shown in **Figure 33-4**.

Machining, Cutting, and Finishing with Delrin Acetal Resin						
Equipment			Cutting Speed			
Operation	**Machines**	**Tools**	**RPM/FPM**	**Feed**	**Coolant Use**	**Remarks**
Sawing	Std.	Std. 14 T.P.I. slight set	100–300 FPM (30.5–91.5 M/min)	Med.	—	Coolant improves finish of cut
Drilling	Std.	Std. twist drills 118°	1500 RPM for 0.500″ (1.27 cm) drill	Med.	At med. and high speeds	On-size holes drilled without coolant
Turning	Std.	Std.	690–840 FPM (210–256 M/min)	0.002″–0.005″ (0.051–0.13 mm)	At high speeds	Depth of cut—0.016″–0.200″ (0.41–5.08 mm); support long lengths
Milling	Std.	Std. cutters; single fluted end mills	Similar to brass	Similar to brass	Not required	—
Shaping	Std.	Std.	Max.	Similar to brass	Not required	—
Reaming	Std.	Expansion type preferred	Similar to brass	Med.	At med. and high speeds	—
Threading and tapping	Std.	Std.	Similar to brass	Similar to brass	At med. and high speeds	Coolant facilitates cutting to dimensions
Blanking and punching	Std.	Std.	—	—	Not required	Primarily for 1/16″ (0.16 cm) thick stock
Filing and sanding	Std.	Std. file, std. abrasive paper and discs	—	—	Wet sanding	Finish will vary with type of file
Finishing	Std.	6″–12″ (15.2–30.4 cm) dia. muslin pumice and water polishing compound	1000–2000 RPM	—	Not required	Use light pressure and rotate part

DuPont Co.

Figure 33-4. Recommended machining, cutting, and finishing operations for Delrin acetal resins.

33.1.3 Teflon

Teflon™ (the DuPont trade name for its fluorocarbon resins) fills a wide range of needs in the electronic, electrical, chemical, and processing industries. It has a very low friction coefficient: rubbing two flat pieces of Teflon together generates about the same friction as rubbing together two ice cubes. This makes Teflon ideal for use as bearings and seals in food processing equipment, where lubricating oil would contaminate the food.

Teflon works well in both high-temperature and ***cryogenic*** (very low-temperature) applications. It is an expensive material, but it will do things that no other material can do as well.

Teflon General Machining Characteristics

Teflon has a high thermal expansion rate. It expands at a rate about 10 times that of steel. When tolerances are critical, measurements should be made at room temperature, or (where applicable) at the temperature at which the part will be used. Teflon should be stored at a temperature of 74°F (23°C) or higher for at least 48 hours before machining operations begin.

Teflon has a tendency to pick up metal shavings and chips. For this reason, no machining should be attempted until the equipment has been thoroughly cleaned of all metal particles.

Turning Teflon

Teflon is more flexible than many other machinable plastics. To prevent deflection of the material away from the cutting tool, take care to support the work properly.

Tools must be sharp and have generous clearances so that the cutting edge of the tool does not rub, **Figure 33-5**. Chips must not be allowed to accumulate around the work because they prevent the heat from dissipating. See **Figure 33-6**.

Cutting fluids are needed when tolerances are critical. Large amounts of water-based coolant help minimize thermal expansion in Teflon. The best surface finishes are achieved at cutting speeds of 200–500 fpm (60–150 mpm) with feeds of 0.0002″–0.010″ (0.005 mm–0.25 mm).

Drilling Teflon

Drills sharpened as shown in **Figure 33-7** provide satisfactory cutting action in Teflon. Teflon tends to swell slightly during the drilling operation, which results in a hole smaller than the drill. To compensate for this swelling, the machinist must use a drill that is slightly oversize. You should test-drill a piece of scrap material to determine the exact drill size needed. For close-tolerance drilling in Teflon, use a feed rate of 0.004–0.006 ipm (0.10–0.15 mpm).

Milling Teflon

Teflon is milled in much the same manner as the other plastics. Only newly sharpened and honed cutters should be used. The work must be solidly supported.

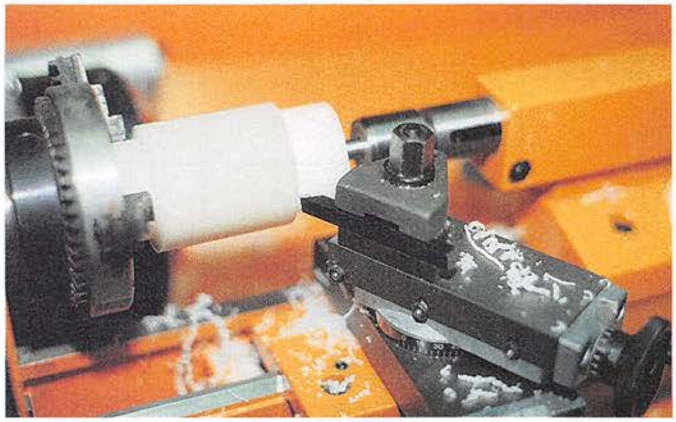
With shell mills, the milling head of the machine should be tilted slightly into the cut, as shown in **Figure 33-8**. This prevents cutter drag marks on the material.

Reaming Teflon

Reaming of Teflon is not advised. If close tolerances are specified, holes should be bored with a single-point tool.

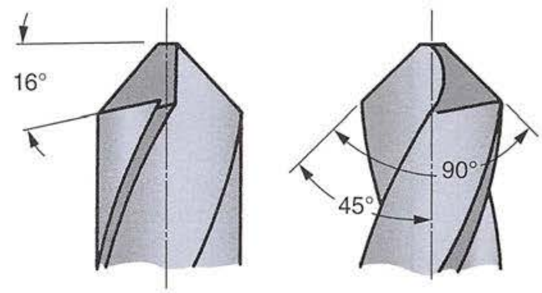
Threading and Tapping Teflon

Teflon is threaded and tapped using the same general techniques as nylon. Refer to the description earlier in this chapter.



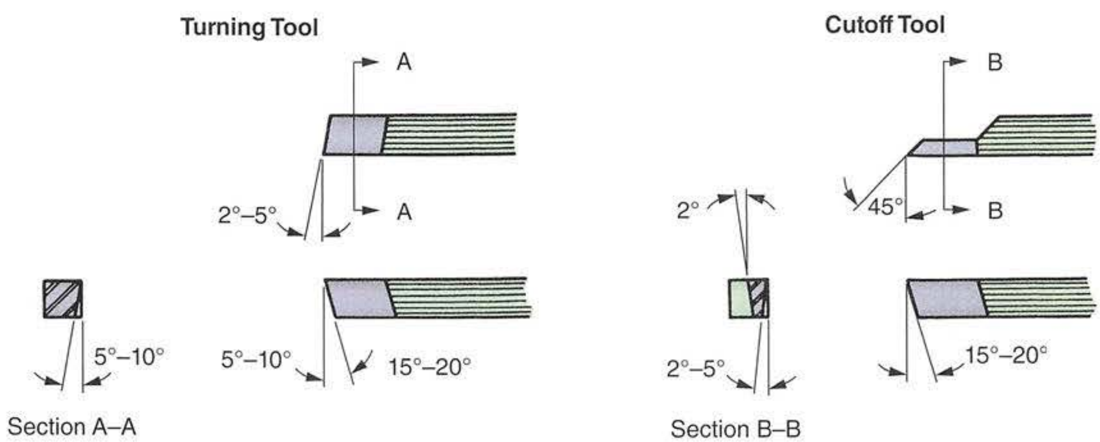
Goodheart-Willcox Publisher

Figure 33-6. When turning plastics, do not allow chips to accumulate around the work. Chips prevent heat from being dissipated, which may distort the work.



DuPont Co.

Figure 33-7. Drill point recommended for Teflon.



DuPont Co.

Figure 33-5. Typical lathe cutting tools for machining Teflon.

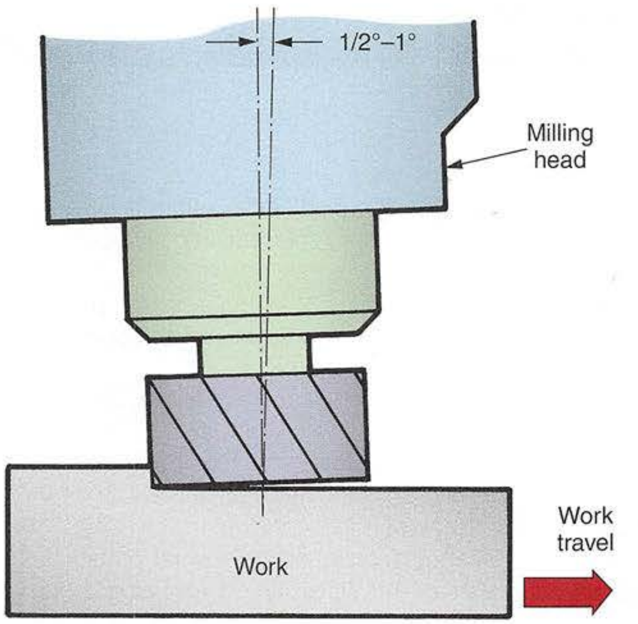


Figure 33-8. For shell milling, the cutter should be tilted slightly (1/2°–1°) into the cut. This ensures the cut is made with the leading edge of the cutter, eliminating cutter marks on the machined surface.

Sawing Teflon

To achieve a square cut in Teflon, a rigid machine with first-class saw guides is essential. No coolant is needed. Maximum machine speeds can be used with a skip-tooth blade that has 4 to 6 teeth per inch.

Annealing Teflon

To maintain dimensional stability, Teflon should be annealed. It is usually heated to a temperature above that to which the finished part will be exposed, but below 621°F (327°C). Above 621°F, Teflon turns to a gel. One hour of annealing for each 1″ (25 mm) of thickness is adequate. Allow the part to cool slowly. Heating is usually done in an oven.

The following oven annealing procedure for Teflon is recommended:

1. Anneal the rod or tubing from which the part is to be made.
2. Rough-machine the part to within 0.06″ (1.5 mm) of finished size.
3. Anneal again, but at temperatures slightly lower than that of the initial annealing.
4. Finish-machine after the Teflon has returned to room temperature.

33.1.4 Acrylics

The acrylic plastics Lucite™ (the DuPont trade name) and Plexiglas™ (the Rohm & Haas Co. trade name) have an unusual combination of desirable characteristics. They have good dimensional stability and high impact strength, even at temperatures as high as 200°F (93°C). Acrylics are also easy to machine, form, and polish. Because of their unusual "light piping" ability (they transmit light much like a hose carries water) and edge lighting qualities, acrylics are commonly used in light control and optical applications.

Acrylics are machined in much the same manner as other plastics. Generally, little difficulty will be encountered if sharp tools with adequate clearance are used. Drills should be sharpened as shown in **Figure 33-9**. Transparent acrylics present a unique machining experience. You can see the tool cutting inside the work. Poor cutting action can be spotted immediately and corrections made.

33.1.5 Laminated Plastics

Laminated plastics consist of layers of reinforcing materials (such as cotton fabric, paper, or glass fiber) that have been impregnated with synthetic resins. The resins are cured under heat and pressure.

Conventional machine tools are used to machine laminated plastics. Some laminated plastics, such as those containing glass fiber, are highly abrasive. Carbide tools are recommended.

SAFETY NOTE

A dust collection system and filtered dust mask or respirator must be used for operator safety when machining plastics that contain glass fiber.

Turning Laminated Plastics

A round-nose lathe tool produces the best surface finish. Speeds up to 4000 fpm (1220 mpm) can be used. Lathe work is usually done dry. However, internal threading may sometimes require use of a lubricant.

Drilling Laminated Plastics

Drilling operations are similar to those used for nylon. However, drilling parallel with the laminations should be avoided whenever possible because the material may split along the laminations. See **Figure 33-10**.

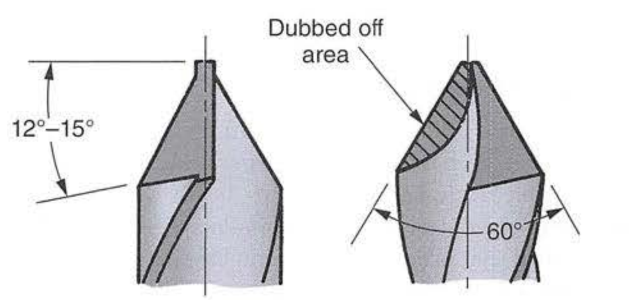


Figure 33-9. Drill point recommended for making through holes in acrylics. The tip angle should be increased to 118° for blind holes. Smoothly finished holes can be produced by first drilling a pilot hole and filling it with wax. The wax allows the chips to move up the flutes without sticking to the drill.

CAREER CONNECTION

Biomedical Engineer

What does a biomedical engineer do?

Biomedical engineers combine principles of engineering, biological sciences, and medicine in order to solve healthcare problems with technology. These engineers test artificial organs, develop replacement body parts, and design machines to diagnose medical problems.

Biomedical engineering is an emerging field with many developing specialties. These professionals may choose to specialize in bioinstrumentation, biomaterials, biomechanics, clinical engineering, rehabilitation engineering, or systems physiology.

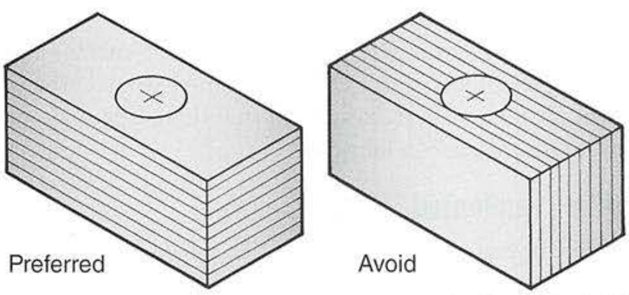
What education and skills are needed to be a biomedical engineer?

Biomedical engineers earn bachelor's degrees in biomedical engineering or in another engineering field with electives in biological sciences. Prospective biomedical engineering students should study chemistry, physics, and biology. They should also take advanced mathematics courses as well as coursework in drafting, mechanical drawing, and computer programming. Biomedical engineering spreads across several professional fields. Taking varied courses in high school and college will help students select future specialties.

What is it like to be a biomedical engineer?

Biomedical engineers serve many roles in different industries. They may work in manufacturing as developers and consultants, in universities as researchers and professors, and in hospitals with medical practitioners and their patients.

Recently, the *Occupational Outlook Handbook* reported that the highest 10% of those employed in this field earned more than $139,500 per year. Overall, biomedical engineers earned median annual wages of $86,200.



Goodheart-Willcox Publisher

Figure 33-10. Whenever possible, holes should be drilled at a right angle to the laminations. Holes drilled parallel to laminations tend to split some types of material.

Milling Laminated Plastics

Speeds up to 1000 fpm (305 mpm) are possible with good results. Feeds up to 20″ (508 mm) per minute have been used. For cotton fabric-based laminates, best results are obtained by using the cutter's highest spindle speed with the maximum feed that produces an acceptable surface finish. Climb milling is recommended to keep the work held tightly in the holding device and to prevent an edge from being raised.

Sawing Laminated Plastics

Band sawing is recommended for curved or straight cuts in laminated plastics when smooth edges and close tolerances are not specified. Blades with 5 to 8 teeth per inch and medium to high set can be operated at speeds up to 8000 fpm (2400 mpm). Feed the work as fast as it will cut without forcing the blade.

Threading and Tapping Laminated Plastics

Laminated plastics can be hand threaded using standard taps and dies. It is normally done dry. High-speed steel taps that are oversize by 0.002″ to 0.005″ (0.05 to 0.13 mm) should be used if available. A slight chamfer on the hole to be tapped, or on the rod to be threaded, improves the work quality by preventing the first few threads from tearing.

33.2 Machining Ceramics

The machinability of a ceramic depends on the material's hardness and on how tightly it is compacted. It is difficult to categorize the hardness of ceramics, partly because the processes used to manufacture ceramic parts can make the ceramic material more or less dense. A part that is more dense has more ceramic particles packed tighter together, and a part that is less dense has fewer particles packed into the same space. This affects the physical hardness of the part.

33.2.1 Determining Ceramic Hardness

The hardness of ceramic materials is usually categorized using the ***Mohs hardness scale***. This scale categorizes materials according to their resistance to being scratched by standard mineral substances. Higher numbers indicate higher scratch resistance. It is a crude, but effective, method for ranking the hardness of ceramic materials. The Mohs hardness scale assigns hardness values from 1 to 10 to reference minerals, as shown in **Figure 33-11**. On this scale, 1 (talc) is the softest and 10 (diamond) is the hardest. For example, a substance that can be scratched by topaz but cannot be scratched by quartz rates between 7 and 8 on the Mohs scale.

33.2.2 Cutting and Forming Ceramics

Raw ceramics, such as granite for countertops, must be cut using an abrasive process. Band machining using diamond blades is a common choice because diamond is harder than granite on the Mohs scale. Cutter failure is generally linked to the failure of the adhesive process holding the ceramic materials in place. Therefore, slower feed rates and cutting speeds and ample liquid coolant are required to extend cutter life.

Formed ceramic parts can be machined in either the green or fired state. Green ceramic parts are formed using various processes. Sometimes plastics are used to help the ceramic flow into dies or molds and to hold the ceramic together until the firing process fuses the ceramic particles together in a process known as *sintering*, which is described later in this chapter. Parts machined in the green state are less dense and can sometimes be machined using traditional machining processes, depending on the type of ceramic and the type of binding materials used to help form the ceramic into its desired shape.

Once fired, the ceramic is in its most dense form. It is this density, coupled with the hardness of the material, that determines whether the part can be machined using traditional machining processes or if a nontraditional process is required.

33.3 Chipless Machining

Chipless machining forms a metal wire or rod into the desired shape using a series of dies. This metalworking technique cannot replace conventional machining, but it does result in substantial cost savings for some jobs by reducing waste and increasing production speed. The process is sometimes called *cold heading* or *cold forming*.

33.3.1 Chipless Machining Processes

In chipless machining, a series of dies replaces conventional machine tools, such as the lathe, drill press, and milling machine. Material used in chipless machining is usually in coil form and is referred to as *wire*. Work is transferred from station to station, where the various dies are used to form the desired shapes. Shapes achieved using this method can be quite complex. In most cases, waste is totally eliminated.

Sementer/Shutterstock.com

Figure 33-11. The Mohs scale rates hardness according to whether a sample can be scratched by these standardized minerals of known hardness values.

Accuracy typically can be held to tolerances of 0.002″ (0.05 mm), and even closer tolerances can be achieved if required. However, cost increases in proportion to precision.

Five basic operations can be performed by the chipless machining process, **Figure 33-12**. Combinations and variations of these operations make possible a wide range of applications.

Chipless machining is an economical and efficient way to make bolts, nuts, screws, and other fasteners, **Figure 33-13**. It is also used to make the vast majority of spark plug bases, **Figure 33-14**.

Metals ranging from aluminum alloys to medium-carbon steel can be shaped using this technique. Stainless

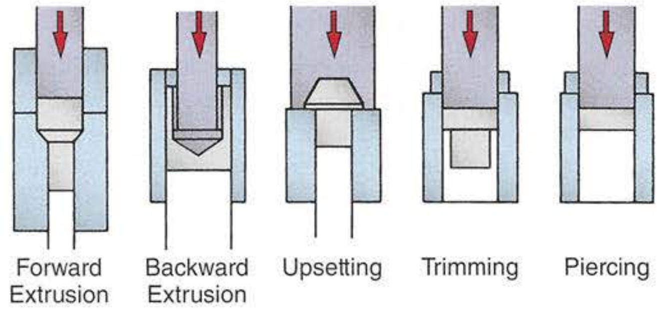


Figure 33-12. The five basic operations performed by machines designed for chipless machining.

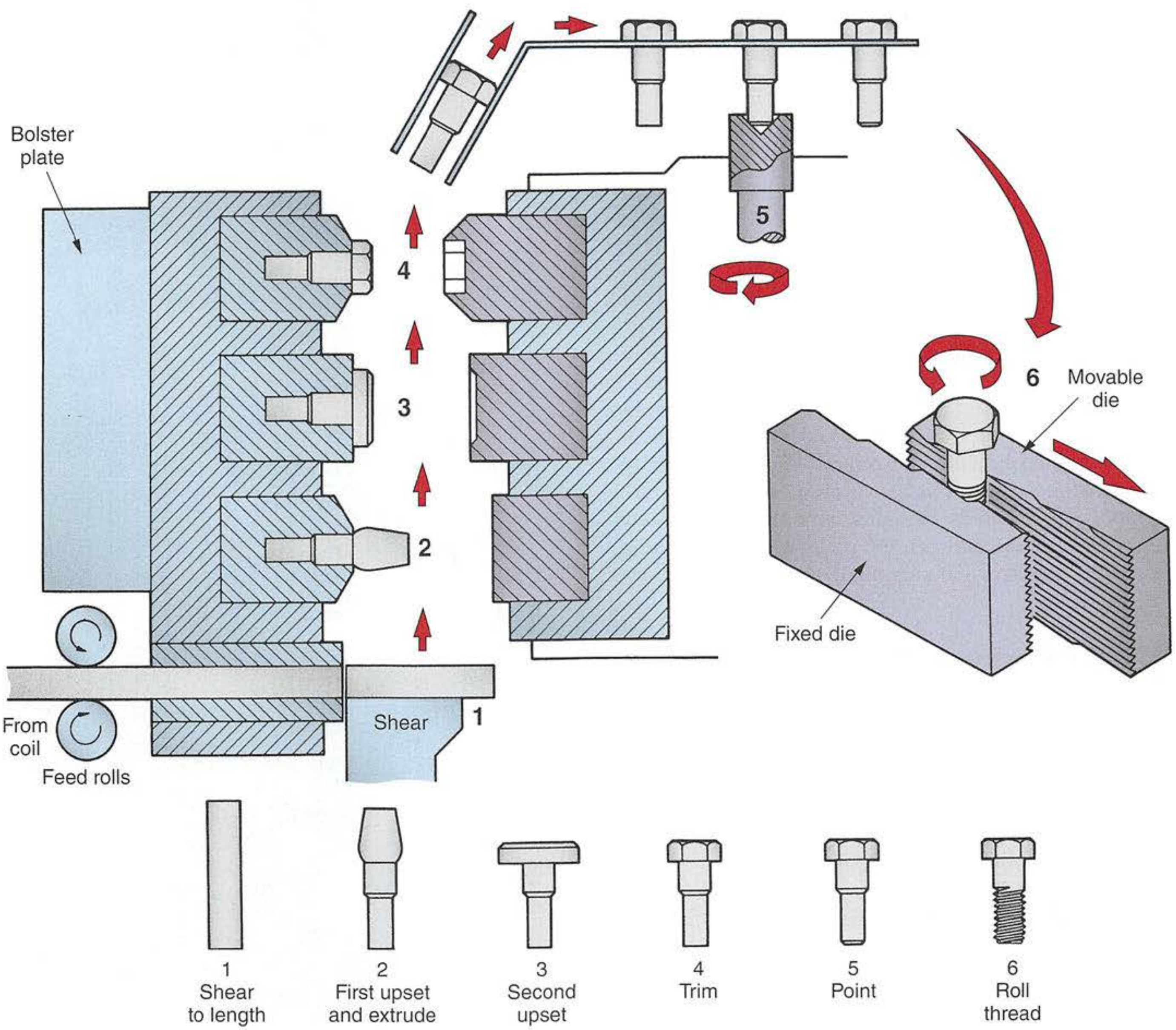
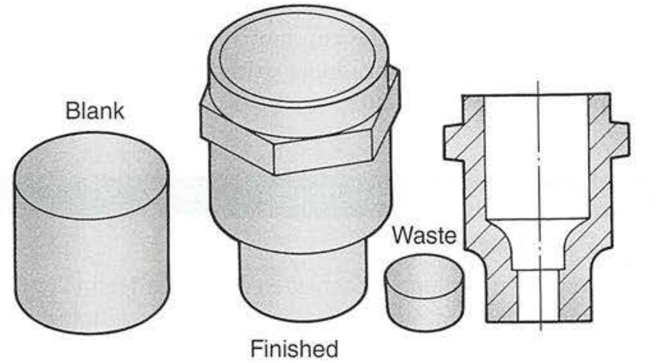


Figure 33-13. The sequence for producing bolts by chipless machining. Trace the part flow through the sequence of operations.



Goodheart-Willcox Publisher

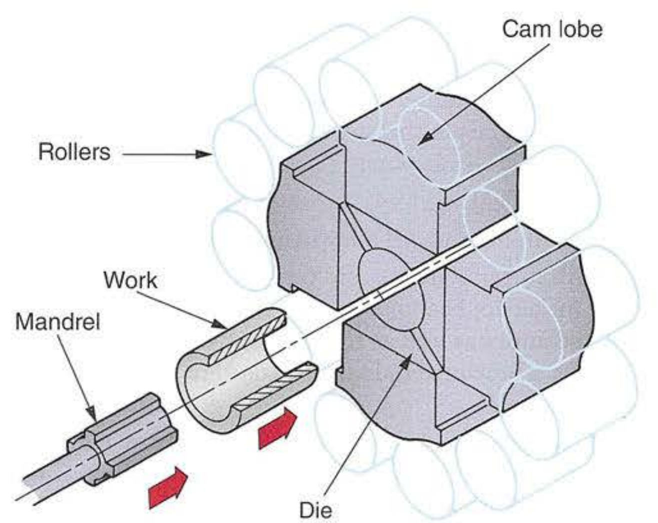
Figure 33-14. Almost all spark plug bases are made by chipless machining. The resulting scrap is minimal.

steel, copper, and nickel alloys can be cold formed, but the ease with which they are shaped depends on the part design. Material up to 1.5″ (38.1 mm) in diameter can be formed on some machines.

33.3.2 Intraform Machining

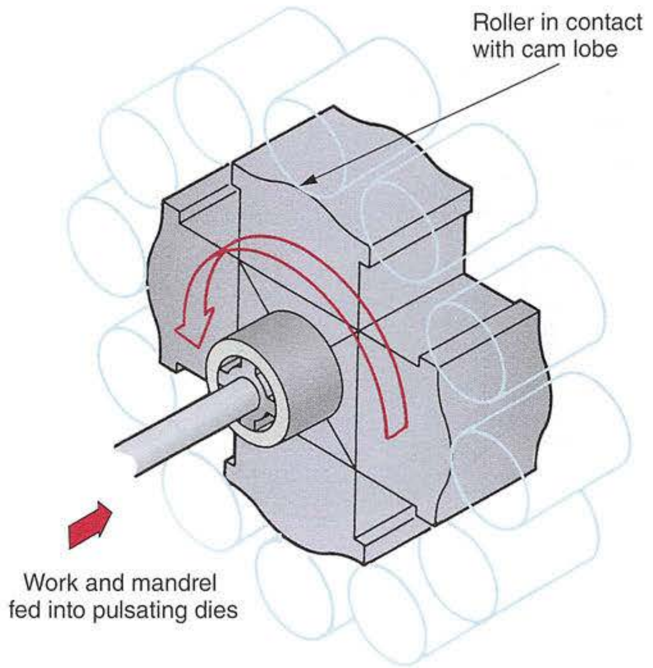
Intraform is a type of chipless machining used to form profiles on the inside diameter of a hollow cylindrical workpiece. Forming inside profiles would be extremely difficult and expensive to do by conventional machining techniques.

In this process, a section of hollow cylindrical stock is placed over a steel mandrel, **Figure 33-15**. It is then squeezed by rapidly pulsating dies, **Figure 33-16**. This reproduces the mandrel's profile on the inside diameter of the part, **Figure 33-17**.



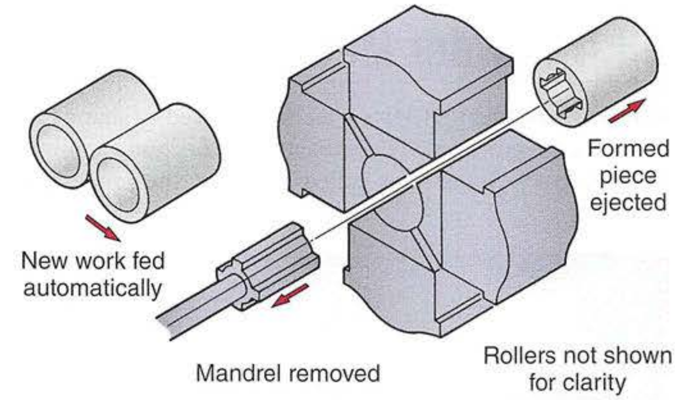
Goodheart-Willcox Publisher

Figure 33-15. A part ready to be shaped by the intraform process. The mandrel is inserted into the work. The work and mandrel are then inserted into the dies. Note that the cam lobes are positioned in the spaces between the rollers, so the pressure on the dies is minimal.



Goodheart-Willcox Publisher

Figure 33-16. As the dies rotate, the rollers apply increasing pressure as they ride up on the cam lobes. Contact with the rotating dies causes the work and mandrel to rotate at about 80% of the die rpm. The work feeds over the mandrel.

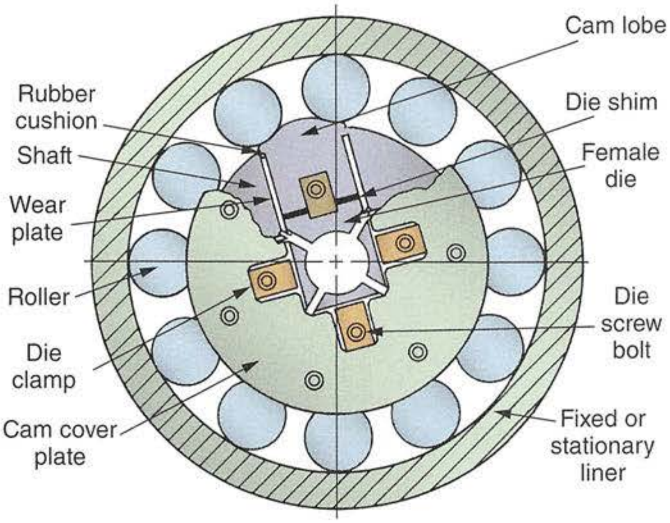


Goodheart-Willcox Publisher

Figure 33-17. When the operation is completed, the mandrel is retracted. The formed part is ejected and the next piece feeds into position automatically.

Figure 33-18 shows how fixed rollers cause the four dies to pulsate rapidly around the outside diameter of the work. The cam lobes are shaped to provide a smooth, continuous squeezing action of the dies. Even though the work is being squeezed by the dies more than 1000 times per minute, there is no excessive noise or vibration.

This technique provides a practical way to produce rifle barrels, for example. Predrilled steel blanks are fed into the machine, which forms the chamber and rifling. In addition to improving the surface finish of the bore, the operation



Goodheart-Willcox Publisher

Figure 33-18. An intraform machine die head in the open position (cam lobes between rollers). Interaction of the cams and rollers squeezes each die more than 1000 times per minute.

improves the physical characteristics of the metal. Approximately 80% of all barrels produced by this process are of target rifle quality, compared to only 10% shaped by conventional methods.

33.4 Powder Metallurgy

Powder metallurgy (P/M) is a technique used to shape parts from metal powders. These parts can be quite complex, **Figure 33-19**. Sometimes called *sintering*, the process was developed in the late 1920s to make self-lubricating electric motor bearings for the automotive industry. The steps involved in fabricating products using powder metallurgy are shown in **Figure 33-20**.

Metal Powder Industries Federation

Figure 33-19. An assortment of products made by the powder metallurgy process.

GREEN MACHINING

Powder Metallurgy

Powder metallurgy is considered a green technology in comparison to other metal-forming or machining processes because it typically produces less waste and requires less energy. For example, machining a part from a solid piece of metal might require cutting away up to 40% of the original material and discarding it. In contrast, powder metallurgy affords machinists the ability to shape any part they want without scrap. The metal powder used in powder metallurgy is created primarily from recycled materials. Furthermore, the energy required in the heating, or sintering, stage of powder metallurgy is less than the thermal energy required for melting processes, such as forging. Powder metal parts usually do not require finishing, which results in additional energy savings.

33.4.1 Powder Metallurgy Applications

The P/M process is widely used by industry for a variety of applications:

- Self-lubricating bearings and bearing materials.
- Precision-finished machine parts, such as gears, cams, or ratchets, with tolerances as close as ±0.0005″ (0.0127 mm).
- Permanent metal filters (sintered bronze fuel filters, for example).

Powder metallurgy is also useful for fabricating materials and components that are difficult to work. Examples of these include the following:

- Tough cutting tools (tungsten carbide), **Figure 33-21**.
- Aluminum-nickel alloy (Alnico) magnets.
- Mixtures of metals and ceramics (cermets) for jet and rocket applications that require the heat resistance of ceramics as well as the heat transfer qualities of metals. These materials are also used to make special cutting tools.
- High-density counterweights for aerospace instruments that require maximum weight concentration in a minimum space.
- Nickel-cadmium (NiCad) storage battery elements.

Wrought powder metallurgy tool steels are now available in mill forms (bars, rods, flats, wires, and plates). They offer improved machinability, wear resistance, increased toughness, and better dimensional stability than conventional cast

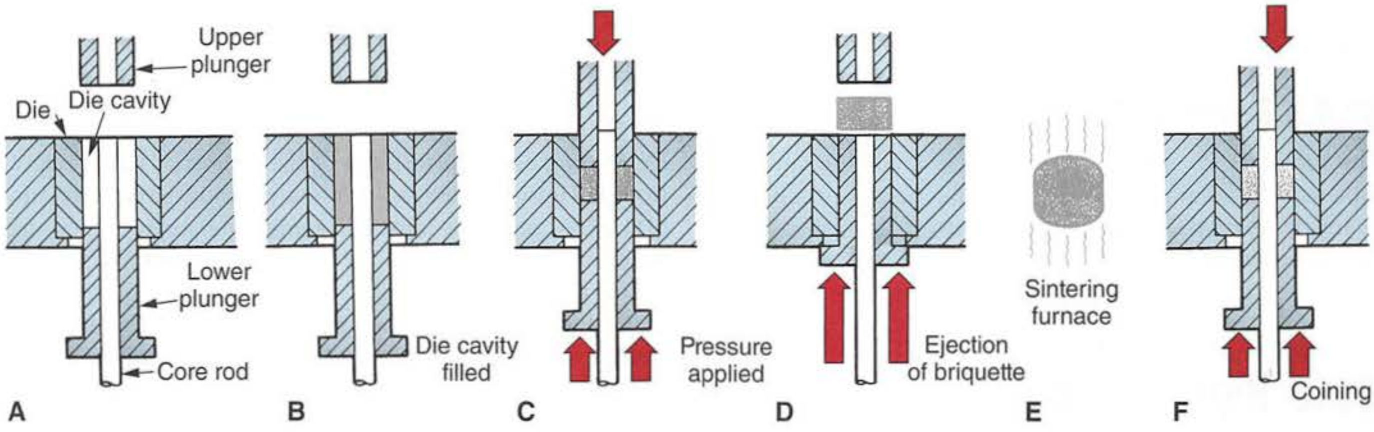


Figure 33-20. Steps in fabricating a part using the powder metallurgy process. A—The depth of the die cavity is determined by the thickness of the required part and by the amount of pressure that will be applied. B—The die cavity is filled with the proper metal powder mixture. C—Pressure as high as 50 tons per square inch is applied. D—The resulting briquette or "green compact" is pushed from the die cavity. E—Pieces are then passed through a sintering furnace to convert them into a strong, useful product. F—Some pieces can be used as they come from the furnace. Others may require a coining or sizing operation to bring them to exact size and to improve their surface finish.

Figure 33-21. A technician examines a tray of tungsten carbide cutting tool inserts following the application of a wear-resistant coating that extends tool life. The inserts were made using the powder metallurgy process.

or wrought tool steels. P/M tool steels are primarily used for metal-cutting and metal-forming operations.

33.4.2 Powder Metallurgy Process

The first phase in the manufacture of powder metal products is the careful mixing of high-purity metal powders. The powders are carefully weighed and thoroughly mixed into a blend of correct proportions. Many materials can be used, including iron, steel, stainless steel, brass, bronze, nickel, and chromium. Combinations of these metals and nonmetals can also be used.

Briquetting

The powder blend is then fed into a precision die. The die cavity has the shape of the desired part, but it is several times deeper than the thickness of the part. The powder is compressed by upper and lower punches. Pressures applied range from 15 to 50 tons per square inch. This portion of the operation is known as ***briquetting.***

The piece, as ejected from the die, appears to be solid metal. However, this briquette or "green compact" is brittle and fragile. It will crumble if not handled carefully.

Sintering

To transform the green compact into a strong, useful unit, it must be heated to a temperature of 1500°F to 2300°F (816°C to 1260°C) for 30 minutes to 2 hours, depending on the powder mixture. This process, known as ***sintering,*** is done in a controlled-atmosphere furnace. The atmosphere within the sintering furnace is carefully controlled to prevent oxidation or contamination of the parts being processed.

Finishing

Many parts can be used as they come from the furnace. However, because of shrinkage and distortion caused by the heating operation, the pieces may have to go through a sizing, coining, or forging operation. The part must be reheated just prior to undergoing these operations, which press the sintered pieces into accurate size dies to obtain precise finished dimensions, higher densities, and smoother surface finishes.

Powder metal parts can also be drilled, tapped, plated, heat-treated, machined, or ground.

33.4.3 Powder Metallurgy Costs

The tool cost for powder metallurgy is moderate, and the process is best suited to quantity production. If production quantities exceed several thousand units, the finished piece can be produced by the P/M method at less than the cost of rough sand castings.

33.5 High-Energy-Rate Forming (HERF)

The introduction of super-tough alloys for aerospace vehicles and the need to shape thin, brittle metal have been responsible for the development of new ways to do the work. One of these new techniques is ***high-energy-rate forming (HERF).*** HERF uses extreme pressures to form a material around a die. There is little similarity between this process and conventional metalworking processes, such as turning, drilling, and milling.

When super-tough alloys are shaped by conventional means, they exhibit spring-back. This means the metal tends to try to revert to its original shape, **Figure 33-22.** It is difficult and costly to shape these metals to acceptable tolerances by conventional means.

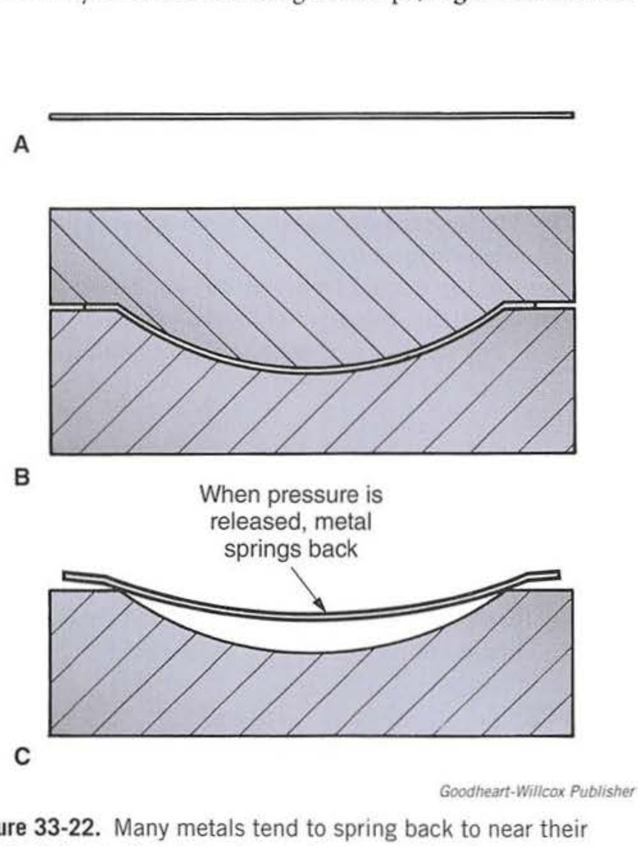


Figure 33-22. Many metals tend to spring back to near their original shape after being formed by conventional means. A—A flat sheet-metal blank ready for forming. B—The metal is formed between dies. C—The metal tries to return to its original shape when the male die is removed or pressure is released.

33.5.1 Types of HERF Operations

Depending on the placement of the explosive, HERF operations fall into two categories: stand-off and contact. In stand-off HERF, the charge is located some distance from the work. Its energy is transmitted through a fluid medium, such as water. This technique is used to form and size parts. In contact HERF, the charge touches the work and the explosive energy acts directly on the metal. Welding, hardening, compacting powdered metals, and controlled cutting are done with this technique.

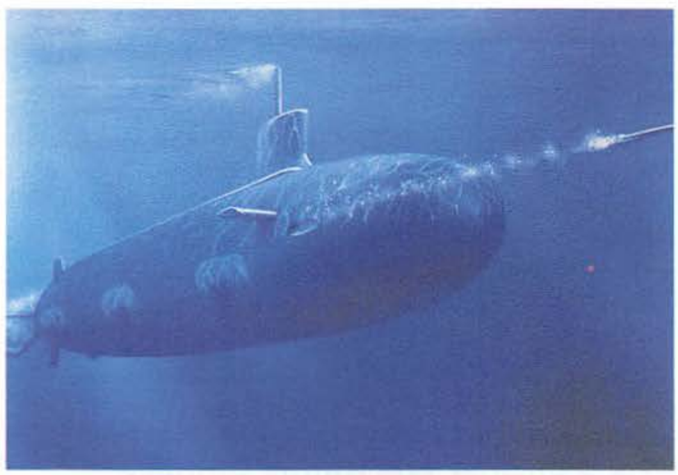
In HERF, the metal is shaped in microseconds with pressures generated by the sudden application of large amounts of pressure. The great pressures are generated by detonating explosives, releasing compressed gases, discharging powerful electric sparks, or applying electromagnetic energy. The metal, in most cases, is slammed against the die and shaped so rapidly that there is no tendency for the material to try to return to its original shape.

HERF offers many advantages. Tool costs are reduced, and there is usually no need for expensive machinery. There appears to be no limit to the size of the sections that can be formed.

33.5.2 Explosive Forming

Explosive forming uses the high-pressure wave of an explosive charge to form the metal. It is an older technique, originating in the late 1800s and used to shape ornate door knobs and similar products. In more recent years, it has been adapted to modern machining. For example, the heavy steel missile hatches on the Navy's submarines, **Figure 33-23,** are formed by this technique.

The size of many aerospace and marine components makes them impossible to form in existing presses. The presses are



U.S. Navy

Figure 33-23. Many parts used on nuclear submarines are shaped by HERF.

either too small or are not powerful enough to develop the pressure required to shape the high-strength alloys.

Explosive Forming Process

Explosive forming uses the pressure wave generated by an explosion in a fluid to force the material against the walls of the die, **Figure 33-24**. The fluid helps to distribute the pressure pulse generated by the detonation equally across the surface of the material.

In preparation for explosive forming, the metal is cut or fabricated to a shape determined by the contours of the finished part. This preform is placed in the die, the die is filled with water, and an explosive charge is suspended in the water, **Figure 33-25**.

A vacuum is necessary between the work and the die. Otherwise, the air trapped between the preform and die would become pressurized by the explosion, preventing the metal from seating in the die and assuming its proper shape. A large holding ring, clamped over the outer edge of the work, provides the seal necessary for drawing a vacuum in the die.

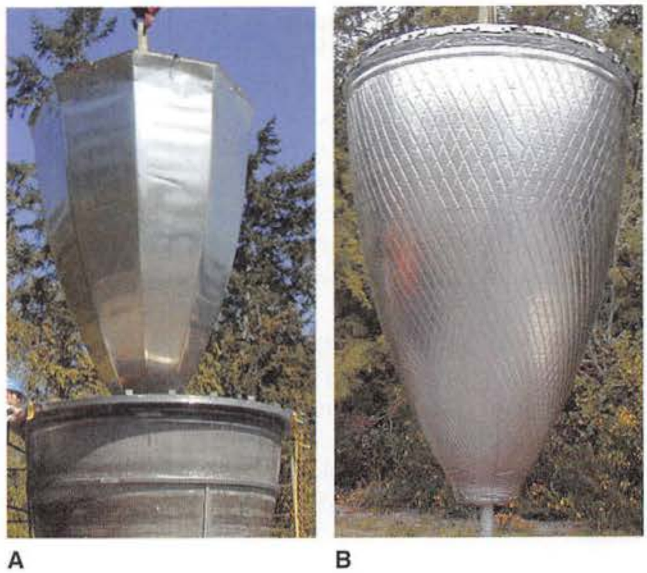
When the explosive is detonated, the resulting pressure slams the material against the die walls. The forming process is accomplished in microseconds.

Placement and quantity of the explosive is critical. The charge can range from a few ounces to form small parts, to the many pounds needed to form large sections of aluminum and steel up to 4″ (102 mm) thick. Many forms of explosives are used: rod, sheet, granules, liquid, stick, cord, and plastic.

Disadvantages of Explosive Forming

While explosive forming offers many advantages, there are also some drawbacks associated with the process:

- The technique has not been developed to the point that a part can always be formed properly on the first shot.
- Noise can be a problem because an explosion is used to do the forming.



PA&E Bonded Metals Division, Sequim, WA

Figure 33-25. Explosive forming was used to shape the skin for space shuttle engines. A—A preform is placed in the die. B—Explosive force shapes the component to take on the required shape.

- Strict laws prohibit the use of explosives in populated areas. As a result, it is usually necessary to locate the facility in an isolated site. This increases transportation and handling costs.
- Personnel must be highly skilled in the safe handling of explosives.
- Insurance rates are high.

33.5.3 Electrohydraulic Forming

Electrohydraulic forming, also called *capacitor discharge forming* or *spark forming,* is similar to explosive forming, but it uses electricity rather than an explosive charge to generate

Goodheart-Willcox Publisher

Figure 33-24. This diagram shows the principle of the explosive forming process. In some applications, a few dollars' worth of explosives will do the work of a press that may cost a million dollars or more.

the required pressure. See **Figure 33-26**. Many of the titanium parts on aerospace vehicles are formed using this technique. Titanium is a tough, light metal, but it is difficult to work by conventional methods.

In electrohydraulic forming, the work is submerged in water, as is done with explosive forming. High-voltage electrical energy is discharged from a capacitor bank (an array of devices used to store electrical energy) into a thin wire or foil suspended between two electrodes, which are also submerged.

As the wire or foil is vaporized by the electric current discharge, the vapor products expand, converting the electrical energy to hydraulic energy. The shock wave forms the metal against the die. Since the energy produced is less than that associated with explosives, it is usually necessary to repeat the operation several times to achieve the desired results.

A well-designed electrohydraulic forming facility can be adapted to automation. Because it generates high pressures without an explosion, the electrohydraulic forming process can be used in conventional industrial facilities.

33.5.4 Magnetic Forming

Magnetic forming, also called *electromagnetic forming* or *magnetic pulse forming*, uses an insulated induction coil wrapped around or placed within the work, **Figure 33-27**. As very high momentary currents are passed through the coil, an intense magnetic field develops. This causes the work to collapse, compress, shrink, or expand, depending on the design of the coil. The coil is shaped to produce the desired shape in the work. See **Figure 33-28**. Coil location depends on whether the metal is to be squeezed inward or bulged outward.

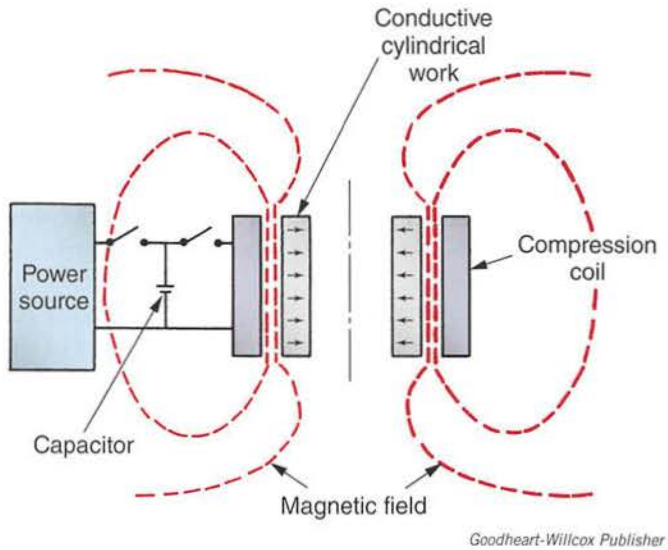


Figure 33-27. Magnetic pulse metal forming. A strong magnetic field is produced by discharging a capacitor through a coil. During the brief impulse, eddy currents in the work restrict the magnetic field to the surface of the workpiece. This creates a uniform force to form the metal.

The power source is basically the same as that used for electrohydraulic forming—a capacitor bank. The spark gap, however, is replaced by a coil. Energy is obtained by capacitor discharge through the coil. In fact, properly designed equipment can be used to perform either electrohydraulic or magnetic pulse operations.

The energy storage capacity and the ability to use that energy determine the size of the work that can be formed. Highly conductive metals can be formed easily by this process. Nonconductive or low-conductivity materials can be formed if they are wrapped or coated with a high-conductivity auxiliary material, such as aluminum.

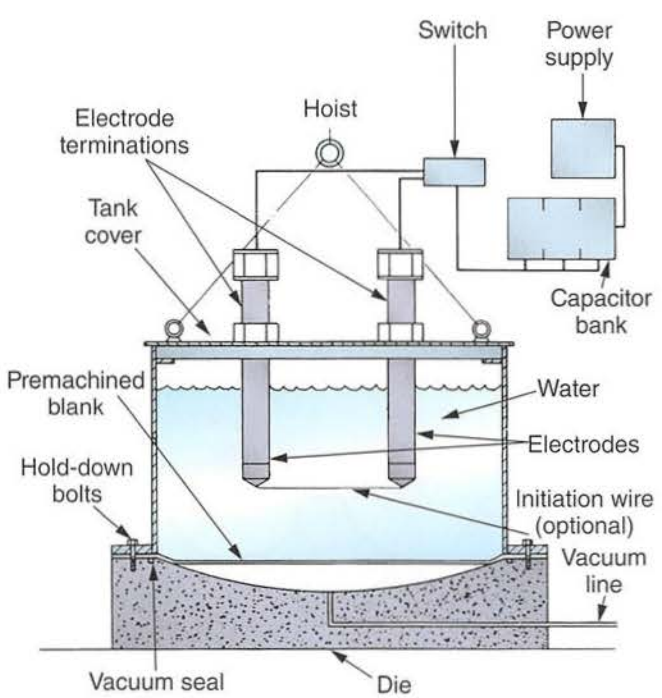


Figure 33-26. Setup for electrohydraulic forming, which uses electrical energy as a source of power for HERF operations.

33.5.5 Pneumatic-Mechanical Forming

Pneumatic-mechanical forming uses a punch and die operated by high-pressure gas. It was the first of the HERF techniques to become a standard production tool. The operation has much in common with conventional forging, since a punch and die are used. However, the machine requires less space than the conventional forging press, and the forces developed are many times more powerful and are sufficient to shape hard-to-work materials.

The metal blank is heated prior to the forming operation. Then a high-pressure gas is used to accelerate a punch into a die. The punch and die are mounted on opposed rams that meet with equal force, thus taking most of the strain off the frame of the machine. Some machines have a recoil mechanism similar to that used on an artillery piece. Because of

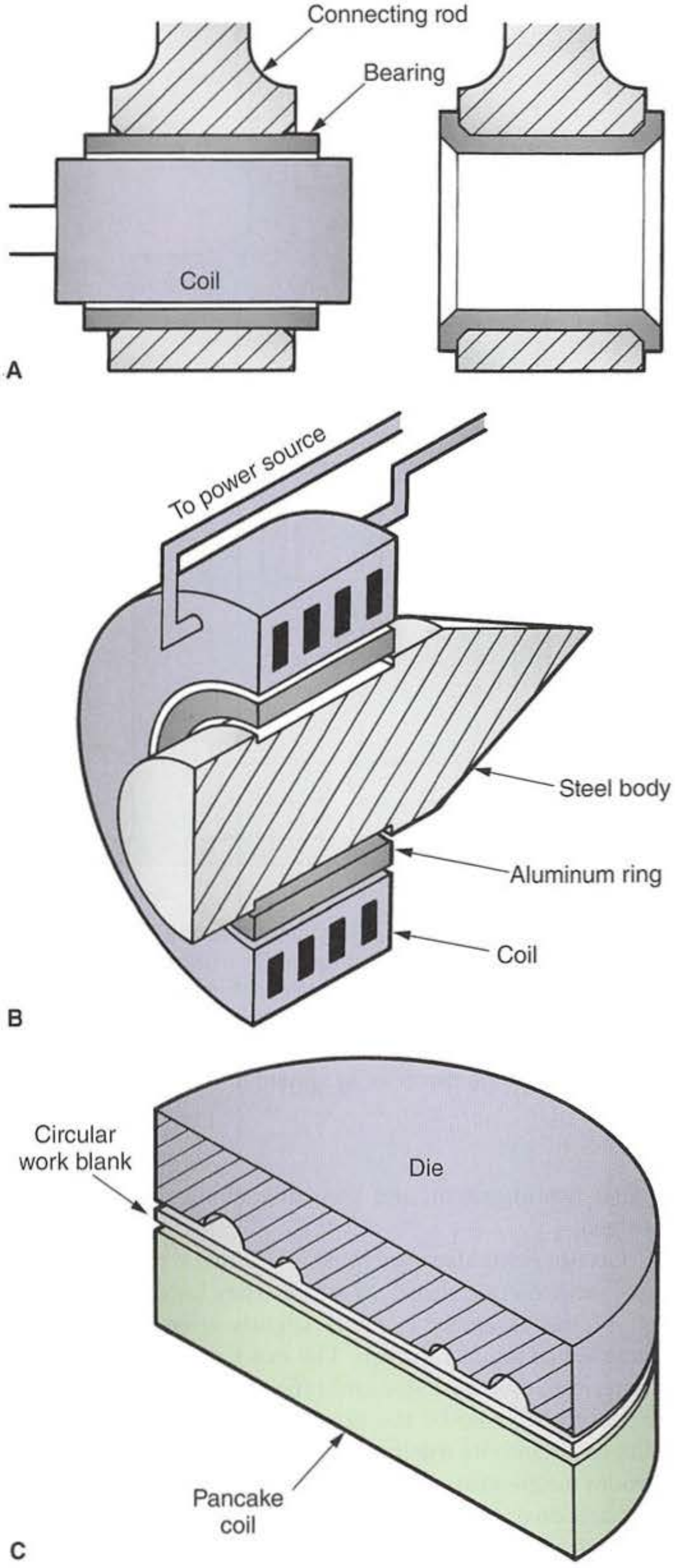


Figure 33-28. Magnetic pulse forming applications. A—Expanding a bearing sleeve into a connecting rod. B—Shrinking or squeezing parts together. C—Flat forming applications, which involve forcing sheet metal into a die, require a pancake coil to provide uniform magnetic pressure.

their self-reacting designs, most machines transmit no shock load to the floor. This makes it possible to use them in close proximity to conventional chip-making machines.

Pneumatic-mechanical forming can be accomplished by several means. The most common method uses a two-part cylinder, **Figure 33-29**. Gas is stored in one part of the cylinder at approximately 2000 psi (13,800 kPa). Gas is also stored in the second section of the cylinder, but at a much lower pressure, about 200 psi (1380 kPa). A plate with an orifice separates the two sections of the cylinder. The piston (ram), with a special seal, closes off the orifice. This causes the high-pressure gas to act on only a small section of the piston (area of the orifice), while the low-pressure gas acts on the entire area of the piston. This maintains a stable balance between the two sections.

To make the machine operate, the pressure is increased slightly in the high-pressure section. This upsets the balance and the piston starts to move. When the seal is disengaged, the high pressure instantly acts on the entire surface area of the piston, driving it forward at tremendous speed, **Figure 33-30**. Even hard-to-shape metals are usually formed with one blow. Some advanced materials can best be shaped by this technique.

With the pneumatic-mechanical method of forging, precise control is possible. Fewer operations are needed (most parts can be formed with one stroke of the press), so production is rapid. The parts are produced to close tolerances with smoothly finished surfaces that require a minimum of machining. Because less material is used, fewer operations must be performed to finish the part, and higher production rates are obtained; thus, substantial savings are possible.

33.6 Cryogenics

The science of cryogenics is one of the more recent additions to the field of machining and metalworking. Cryogenics is not a new or different way to work metal; rather, it is

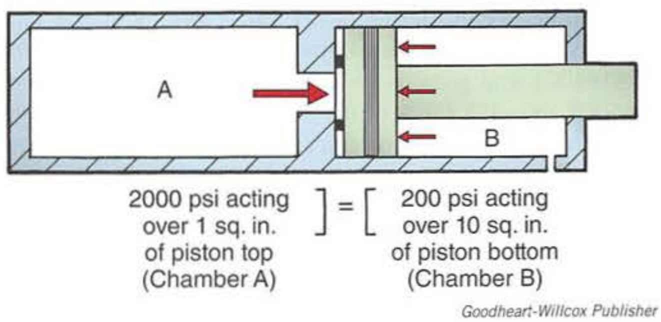
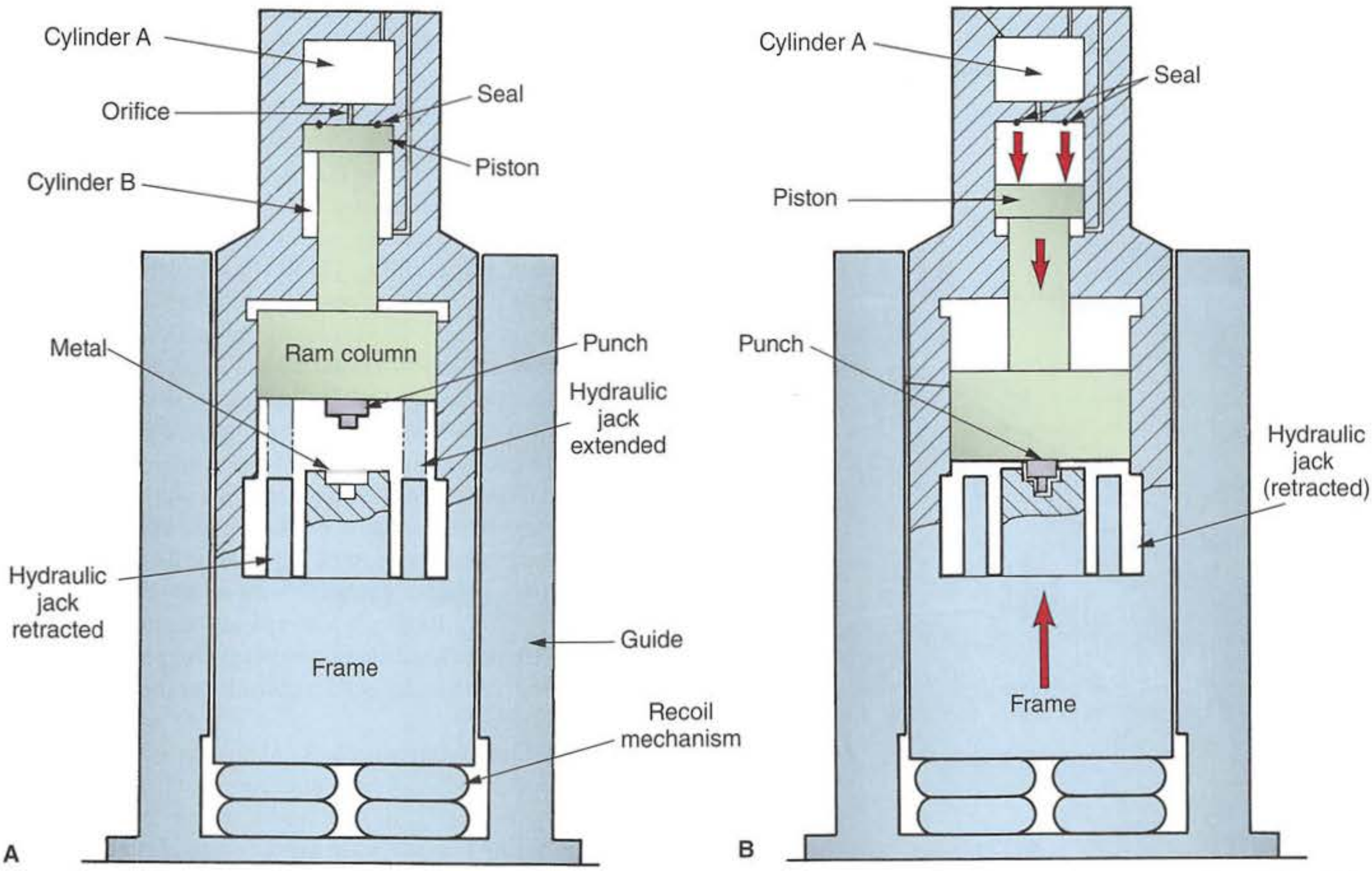


Figure 33-29. The high-pressure gas acts on only a small area of the piston top (area of the orifice), while the low-pressure gas acts on the entire area of the piston bottom. This maintains a balance between the two forces.



General Dynamics Corp.

Figure 33-30. Pneumatic-mechanical forming. A—Cross-sectional view of a pneumatic-mechanical forming press. The hydraulic jacks extend at the end of each operating cycle to lift the ram column back into position for the next cycle. B—The operation of the press is triggered when the pressure in Cylinder A is increased enough to break the seal. This slight movement allows high-pressure gas to act instantaneously over the entire area of the piston. The ram is driven downward at great speed. At the same time, the frame moves upward by reaction of gas pressure over a driven piston. The frame and ram are acted on with equal thrust, so each has equal momentum but in opposite directions. To reset for the next cycle, the jacks lift the ram column upward until it seats against the seal.

a technique used to improve or reinforce other metalworking techniques. The term *cryogenic* literally means "made icy cold." It combines the Greek word *kryos,* meaning "icy cold," and the Latin *generatus,* which means "to make or create."

The cryogenics temperature range begins at the temperature at which oxygen liquefies (approximately –300°F or –184°C), and goes down to approximately absolute zero (–460°F or –273°C). At this temperature, every element except helium freezes. Oxygen and nitrogen look something like white beach sand or table salt. Metal also acts strangely. Lead coils act like steel springs, some metals increase tremendously in strength, and others become superconductors of electricity.

33.6.1 Cryogenic Applications

Cryogenics is widely used in annealing and in the heat treatment of metals. The characteristics of a number of aluminum alloys and some space-age metals are greatly improved by first heating them, and then quenching them in liquid nitrogen.

Cryogenics is also used to shrink-fit metal parts together. Since most metals shrink in size as they become cold, one part of the assembly is made slightly oversize and then immersed in liquid nitrogen. The exact amount of oversize is determined by part size and type of metal. The diameter is reduced (shrunk) by the extreme temperature drop until it fits easily into its mating part. As the cooled part returns to room temperature, it expands and is thus locked in place. The parts do not become distorted as they would if they were mechanically pressed together or heated (expanded) until they could be fitted together.

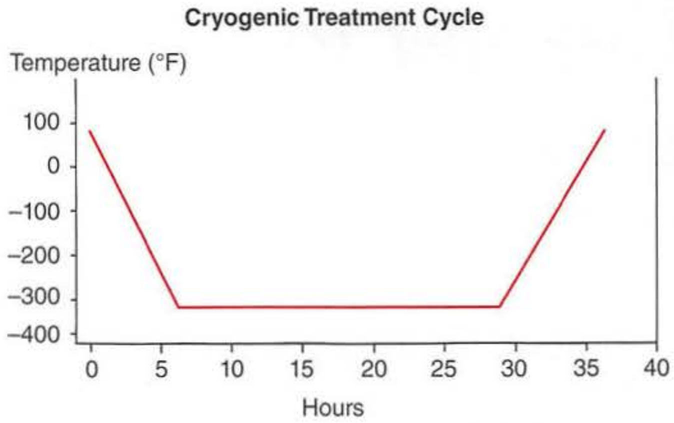
33.6.2 Cryogenic Treatment of Cutting Tools

Research has shown that cryogenic treatment of cutting tools can greatly extend their operating life. Cryogenic treatment

produces carbide inserts that last two to eight times longer than untreated inserts. Similar treatment of copper spot welding electrodes increases their useful life by 300%. Other types of cutting tools have shown increases of up to 400% in tool life.

During cryogenic treatment, liquid nitrogen enters the cooling chamber as a gas, at a carefully controlled rate. The cutting tools are not immersed in liquid nitrogen. Cooling occurs at a slow rate to avoid a sudden change in temperature that would cause damaging thermal shock.

The full cooling cycle takes about 40 hours. Lowering the temperature to about –320°F (–196°C) requires about 4 to 6 hours. The tools "soak" at this temperature for 24 hours or more. Another 4 to 6 hours are required to return the tools to room temperature. See **Figure 33-31**. For larger or more massive items, the time required to lower and raise temperature in the cooling chamber must be increased.



Goodheart-Willcox Publisher

Figure 33-31. To achieve the maximum benefit from the cryogenic treatment, cutting tools require a gradual lowering of temperature, an extended soak period, and a slow return to room temperature.

Chapter Review

Summary

- Most types of nylon can be machined with the same techniques used to machine soft brass. Since nylon is not as rigid as metal, it must be well supported during machining operations.
- Parts made from Delrin have dimensional stability, high strength, and rigidity. Delrin is machined in the same way as nylon.
- Teflon has a high thermal expansion rate. It should be stored at a temperature of 74°F (23°C) or higher for at least 48 hours before the machining operations. After being machined, Teflon should be annealed.
- Acrylics have good dimensional stability and high impact strength, even at elevated temperatures. They are easy to machine, form, and polish. Sharp tools with adequate clearances should be used.
- Laminated plastics consist of layers of reinforcing materials that have been impregnated with synthetic resins. Carbide tools may be required for machining some laminated plastics. Drilling parallel with the laminations should be avoided whenever possible.
- Chipless machining is a process in which a metal wire or rod is forced into the desired shape using a series of dies. It is an economical and efficient way to make bolts, nuts, screws, and other fasteners.
- Intraform is a type of chipless machining used to form profiles on the inside diameter of a hollow cylindrical workpiece.
- Powder metallurgy is a method of making parts from metal powders. Metal powders are pressed in a die to form a green compact. The green compact is fragile when it emerges from the die and is placed in a controlled-atmosphere furnace to strengthen.
- High-energy-rate forming (HERF) uses extreme pressures to form a material around a die. The pressures can be generated by explosives, the release of compressed gas, electrical discharge, or electromagnetism.
- Cryogenics is the application of extremely cold temperatures to change the physical properties of a material. Cryogenic temperatures are used to shrink-fit parts. The durability of cutting tools is also commonly improved by cryogenic treatment.

Review Questions

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

1. Give a common trade name for each of the following types of plastics:
   A. Polyamide resins.
   B. Acetal resins.
   C. Fluorocarbon resins.
   D. Acrylic resins.
2. If plastics are to be machined with any degree of accuracy, the cutting tools must be _____.
3. Like metal, machined nylon parts require _____ to prevent dimensional changes.
4. When turning Teflon on a lathe, care must be taken to prevent _____ from accumulating around the work. If this is not done, heat will build up and cause the plastic to become distorted.
5. When drilling laminated plastics, avoid drilling holes _____.
   A. parallel to the laminations
   B. perpendicular to the laminations
   C. all the way through the material
   D. All of the above.
   E. None of the above.
6. What system is commonly used to categorize the hardness of ceramic materials?
7. Chipless machining is also known as _____ or cold forming.
8. In chipless machining, metal is formed into the desired shape using a series of _____.
9. List the five basic operations that can be performed by the chipless machining process.
10. Intraform is a chipless machining technique that can form profiles on the inside of hollow _____ workpieces.
11. The powder metallurgy process is used to make _____.
    A. self-lubricating bearings
    B. precision machine parts
    C. permanent metal filters
    D. All of the above.
    E. None of the above.

12. What is a briquette?
13. List the steps, in proper order, for making a part by the powder metallurgy technique.
14. The abbreviation HERF means _____.
15. In HERF, metal is shaped _____.
    A. by the slow application of great pressure
    B. with pressure generated by the sudden application of large amounts of energy
    C. by conventional forging methods
    D. All of the above.
    E. None of the above.
16. List three methods of generating the pressures needed for a HERF operation.
17. _____ is a HERF technique that uses an induction coil to shape metal.
18. In pneumatic-mechanical forming, _____ is used to accelerate the punch into the die.
    A. pressurized oil
    B. pressurized gas
    C. explosives
    D. All of the above.
    E. None of the above.
19. The science of cryogenics deals with temperatures ranging from −300°F (−184°C) to _____.
    A. 32°F (0°C)
    B. 0°F (−18°C)
    C. −100°F (−73°C)
    D. −460°F (−273°C)
20. Why is it better to use cryogenics, rather than heat, to shrink-fit parts together?
21. Why must the cooling of treated cutting tools be done at a slow rate?

APPENDIX A

Math Review

Why Math

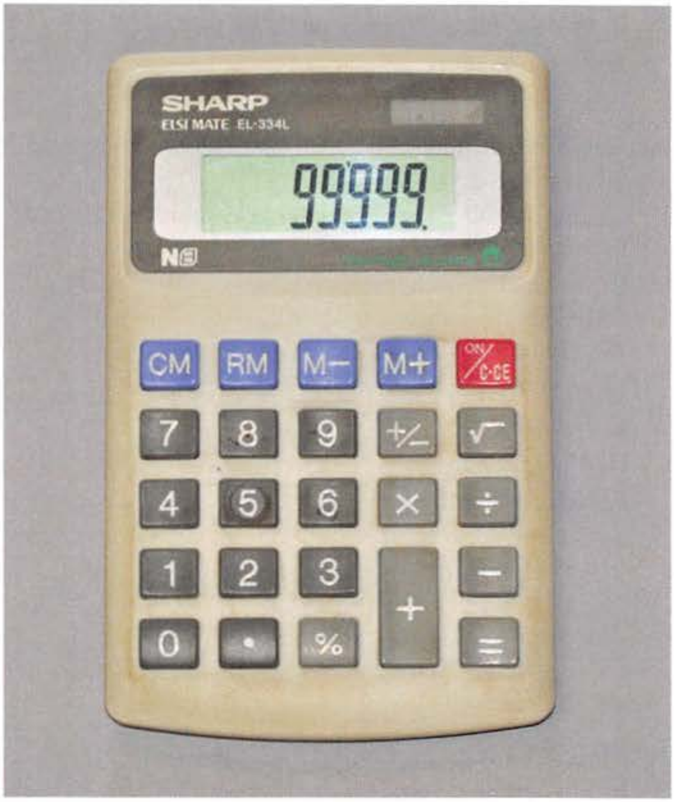
All mechanical trades involve using math for numerous tasks. Machinists use math to make precise measurements and convert units. They must calculate speeds and feeds for machining operations including drilling, turning, milling, and tapping. Proper machine setup and job planning require using math to determine the right sizes and locations for materials and tools. Some trades require use of other specialized or more advanced math, but all trades require an understanding of what is presented here.

Calculators

Calculators can be a great time-saver, but you should not rely on a calculator to replace your knowledge of basic math. There will be times in the field when a calculator is not available and you will want to do some basic math. However, a calculator can be a handy tool when one is available. Knowing some basic math and using common sense observation can help prevent big errors when using a calculator.

It is important to do operations in the correct order when using a calculator. For example, the formula for the area of a circle requires multiplying 3.1416 by the radius of the circle squared (multiplied by itself). Let's say the radius is 5″. The area is found by multiplying 5×5 (5 squared), then multiplying the result by 3.1416. If you know that 5×5 (written 5^2) is 25, and 3 times 25 is 75, you will know the answer is slightly more than 75. Now you can check to see if the answer you got on your calculator is close. The correct answer is 78.54 in^2. As stated earlier, it is important to know which order to multiply the numbers. Using this example, if you multiply 3.1416×5 and then multiply that by itself, you'll get 246.7413—not even close! Knowing the correct order to do the operations on paper will help you enter numbers in the correct order on your calculator.

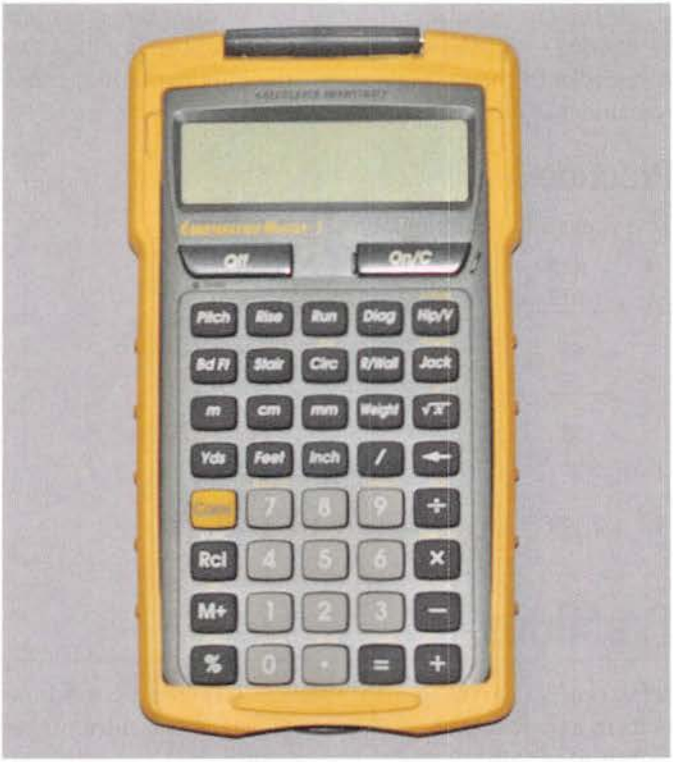
There are several types of calculators one might consider. The most familiar type is a *general calculator*, which allows the user to perform basic math functions. See **Figure A-1**. *Basic calculators* have a memory function that allows the user to store the results of a calculation so that number can be recalled and used in a subsequent calculation. Most also have a % key. When a number is shown in the display, pressing the % key allows the user to then press a number key to calculate that percent of the displayed number. The √ key is for finding the square root of a number. (Squares, roots, and exponents are covered later in this supplement.) The ± key changes the value of a displayed number from positive to negative or negative to positive. Scientific calculators are more advanced, having the ability to do many trigonometric functions and other advanced operations. Scientific calculators are not necessary for most trades math.



Goodheart-Willcox Publisher

Figure A-1.

Another type of calculator that can be very useful for trades workers is the *construction calculator*. See **Figure A-2**. Construction calculators can be used to convert inches to feet and feet to inches. Most allow the user to convert between metric and US Customary measurements. Construction calculators vary from one manufacturer and model to the next, so it is not possible to give a detailed description or instructions here. All calculators come with instructions that explain their functions.



Goodheart-Willcox Publisher

Figure A-2.

PRO TIP

Always look at your answer and compare it with what might be a reasonable answer. This is the most important tip for calculator use. If the answer does not look reasonable, it is probably not correct.

Whole Numbers

Whole numbers are simply numbers without fractions or decimal points, numbers such as 1, 2, 3, 4, etc. Adding, subtracting, multiplying, and dividing whole numbers primarily requires memorizing a few math facts.

Adding and Subtracting Whole Numbers

Example: Adding this column of whole numbers requires memorizing the *sum* of 3 + 5 and the sum of 8 + 2.

$$\begin{array}{r} 3 \\ 5 \\ +\quad 2 \\ \hline 10 \end{array}$$

The same type of memorization of math facts is required to subtract whole numbers. We know that the result of subtracting 12 from 37 is 25 because we know that 2 from 7 is 5 and 1 from 3 is 2.

$$\begin{array}{r} 37 \\ -\quad 12 \\ \hline 25 \end{array}$$

The key to both addition and subtraction is to line up the columns of digits correctly. The whole numbers should be aligned on the right.

In subtraction, if the number being subtracted (the number on the bottom) is larger than the number it is being subtracted from (the number on the top), borrow 10 from the next digit to the left and add it to the one on the right. Write small numerals above the column to help you keep track.

Example:

$$\begin{array}{r} {\scriptstyle 2\ 16} \\ \not{3}\not{6} \\ -\quad 19 \\ \hline 17 \end{array}$$

Multiplying Whole Numbers

Multiplication of whole numbers requires memorization of a multiplication table. The only way to get $6 \times 5 = 30$ is to know that multiplication fact or to add 6 + 6 + 6 + 6 + 6. That becomes way too tedious for bigger multiplication problems. To multiply numbers whose values are 10 or more (those with more than one digit), align the digits representing 0 through 9 (the 1s) in the right-hand column. Then multiply the top row by the 1s digit in the second row:

Example:

10s
1s

$$\begin{array}{r} 31 \\ \times\quad 15 \\ \hline 155 \end{array}$$

Next, multiply the top row by the 10s digit in the second row. Because you multiplied by the 10s digit, the *product* (the result of multiplication) is written with its right-most digit in the 10s column:

$$\begin{array}{r} 31 \\ \times\quad 15 \\ \hline 155 \\ 31 \end{array}$$

If the problem has more digits in the second row, the above steps are repeated for each digit with the products being written in rows beneath one another, with the right-most digit in each row being written in the column for the place it represents: 100s, 1000s, etc.

When all of the multiplication is complete, add the products just as you would for a simple addition problem. The result of this addition is the product (answer) of the multiplication problem.

$$\begin{array}{r} 31 \\ \times\quad 15 \\ \hline 155 \\ 31 \\ \hline 465 \end{array}$$

Dividing Whole Numbers

Division of whole numbers is the reverse of multiplication, but the problem must be set up differently. The *dividend* (the number being divided) is written inside the division symbol. The *divisor* (the number the dividend will be divided by) is written to the left of the symbol:

divisor ⟶ $7\overline{)28}$ ⟵ dividend

From the multiplication table, we know that $7 \times 4 = 28$. So, if 28 is divided into 7 parts, each part will have 4, or $28 \div 7 = 4$. 4 is the *quotient* (the answer to a division problem) and it is written above the division symbol and above the 1s place of the 28:

$$\begin{array}{r} 4 \\ 7\overline{)28} \end{array}$$ ⟵ Quotient

When the divisor is more than 9, the process is divided into steps as follows:

$$4\overline{)320}$$

4 goes into 32 8 times. Write the 8 above the 2 (the right column of the 32). Now multiply 4×8, which is 32. Write the 32 beneath the 32 in the division symbol.

$$\begin{array}{r} 8 \\ 4\overline{)320} \\ \underline{32} \end{array}$$

Subtract the product of your multiplication (32) from the number above it in the symbol (32). Because 4 goes into 32 exactly 8 times, the numbers are the same, so the result of your subtraction is 0.

$$\begin{array}{r} 8 \\ 4\overline{)320} \\ \underline{32} \\ 0 \end{array}$$

Drop the next digit to the right in the dividend (0 in this case) down beside the result of your subtraction. That makes the number at the bottom 00. 4 will not go into 0 (or 00), so the quotient of that step is 0. The quotient (answer) of the division problem is 80.

$$\begin{array}{r} 80 \\ 4\overline{)320} \\ \underline{32} \\ 00 \end{array}$$

320 can be divided by 4 80 times. If there are more places under the division symbol, just keep doing the same division, multiplication, subtraction, and drop down for each digit moving to the right.

Example:

$$\begin{array}{r} 102 \\ 6\overline{)616} \\ \underline{6} \\ 01 \\ \underline{0} \\ 016 \\ \underline{12} \\ 4 \end{array}$$ ⟵ remainder

If the last number produced by the drop-down cannot be divided evenly by the divisor, that number is called the *remainder*. In the example above, the quotient is 102 with a remainder of 4.

Practice A-1

Test your skills with the following problems.

A. $342 + 16$ B. $79 + 29$

C. $68 - 13$ D. $124 - 35$

E. 18×4 F. 213×24

G. $3\overline{)36}$ H. $7\overline{)214}$

Fractions

A fraction is a part of something larger. If there are 3 machine bolts in a pound, each machine bolt weighs one-third of one pound. One-third can be written as follows:

$$\frac{1}{3}$$ 1 ⟵ numerator, 3 ⟵ denominator

The number above the fraction bar is called the *numerator*. It indicates how many parts are in the fraction, in this case 1 machine bolt. The number below the fraction bar is the *denominator*. The denominator indicates how many parts are in the whole, in this case 3 machine bolts in the whole pound. If there are 50 machine bolts in a carton and we take 7 of them out, we have taken

$$\frac{7}{50}$$

of the machine bolts.

Equivalent Fractions

If two fractions represent the same value, they are said to be *equivalent fractions*. For example, 1/3 and 2/6 are equivalent fractions because they both represent one-third of the whole. If both the numerator and the denominator of a fraction are multiplied by the same amount, the result is an equivalent fraction.

Example:

$$\frac{1 \times 2 =}{3 \times 2 =} \; \frac{2}{6}$$

Adding Fractions

Fractions must have *common denominators* in order to be added. If the denominator of one of the fractions is 8, the other fraction must be written as an equivalent fraction with

a denominator of 8. For example, to add 3/4 and 1/8, write 3/4 as an equivalent fraction with a denominator of 8.

Example:

$$\frac{3}{4} \times \frac{2}{2} = \frac{6}{8}$$

When both fractions have the same denominator, add the numerators.

$$\frac{6}{8} + \frac{1}{8} = \frac{7}{8}$$

6 eighths plus 1 eighth is a total of 7 eighths.

A common denominator can be found by multiplying all of the denominators in a problem.

Example:

$$\frac{1}{3} + \frac{2}{5} + \frac{9}{14}$$

$$3 \times 5 \times 14 = 210$$

$$\frac{70}{210} + \frac{84}{210} + \frac{135}{210} = \frac{289}{210}$$

In this example, the numerator is larger than the denominator. This is because the value of the fraction is greater than 1. Two hundred eighty-nine is 79 parts larger than the whole of 210 parts. The same number could be written as:

$$1\,\frac{79}{210}$$

This is called a *mixed number* because it is made up of a whole number plus a fraction.

Subtracting Fractions

Subtracting fractions is similar to adding them. Both fractions must have a common denominator, then the numerators are subtracted just like whole numbers.

Example:

$$\frac{2}{3} - \frac{1}{4} =$$

Find common denominators and subtract the numerators.

$$\frac{8}{12} - \frac{3}{12} = \frac{5}{12}$$

Multiplying Fractions

To multiply fractions, multiply the numerators. Then multiply the denominators.

Example:

$$\frac{3}{4} \times \frac{2}{5} = \frac{3 \times 2}{4 \times 5} = \frac{6}{20}$$

To make the result easier to work with, always reduce it to its lowest terms. That means to write it as an equivalent fraction with the lowest possible denominator. For example, both 6 and 20 can be divided by 2 to make an equivalent fraction with a smaller denominator.

$$\frac{6 \div 2}{20 \div 2} = \frac{3}{10}$$

In this case, $\frac{3}{10}$ is the lowest terms of $\frac{6}{20}$.

To multiply a fraction by a mixed number, first change the mixed number to a fraction. Then multiply as common fractions. Reduce the product to its lowest terms.

Example:

$$\frac{2}{3} \times 4\,\frac{1}{2} = \frac{2}{3} \times \frac{9}{2}$$

$$\frac{2}{3} \times \frac{9}{2} = \frac{18}{6}$$

$$\frac{18}{6} = \frac{3}{1} = 3$$

Dividing Fractions

To divide fractions, invert the divisor (swap the numerator and denominator) then multiply as common fractions. It may help you to remember "keep it, change it, flip it"—keep the first fraction as it is, change the division to multiplication, and flip the second fraction.

Example:

divisor

$$\frac{3}{4} \div \frac{2}{3} = \frac{3}{4} \times \frac{3}{2}$$

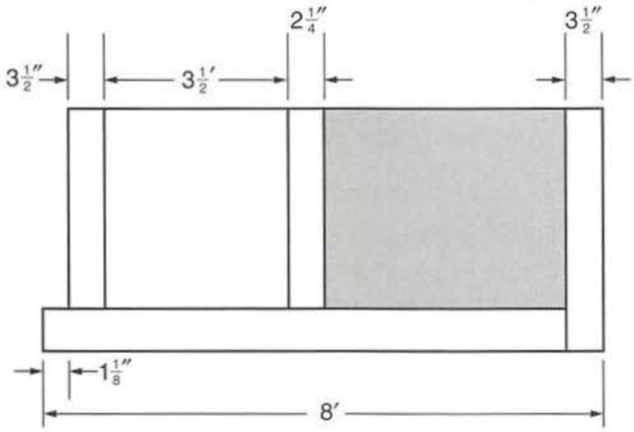
$$\frac{3 \times 3}{4 \times 2} = \frac{9}{8}$$

In this example, the quotient is a fraction with a larger numerator than its denominator. This indicates that the value is greater than 1. To express this in its simplest form, convert it to a mixed number.

$$\frac{9}{8} = \frac{8+1}{8} = 1\,\frac{1}{8}$$

Sometimes it is necessary to combine operations in a single problem. Some such problems also involve more than one type of unit, such as inches and feet. The first step in solving such problems is to decide which operation should be done first. It often helps to write the problem in such a way that it states the order of operations. Next, convert everything to the same unit(s). Where the operations will include adding or subtracting fractions, convert them to their least common denominators. Now solve the problem, doing all operations in the planned order. Finally, convert the units to whatever makes sense for the problem. For example, you would not write fine measurements in square feet.

How long is the shaded space in this drawing?



1. Convert the overall length to inches. 96″
2. Subtract $1\frac{1}{8}''$. $95\frac{7}{8}''$
3. Convert $3\frac{1}{2}'$ to inches. 42"
4. Add dimensions at top. $3\frac{1}{2}'' + 42'' + 2\frac{1}{4}'' + 3\frac{1}{2}'' = 51\frac{1}{4}''$
5. Subtract $51\frac{1}{4}''$ from $95\frac{7}{8}''$. $95\frac{7}{8}'' - 51\frac{2}{8}'' = 44\frac{5}{8}''$

Practice A-2

A. $\frac{1}{4} + \frac{5}{8}$

B. $\frac{3}{4} - \frac{1}{3}$

C. $\frac{1}{2} \times \frac{3}{4}$

D. $\frac{1}{3} \div \frac{1}{2}$

E. $\frac{2}{3} + \frac{7}{12}$

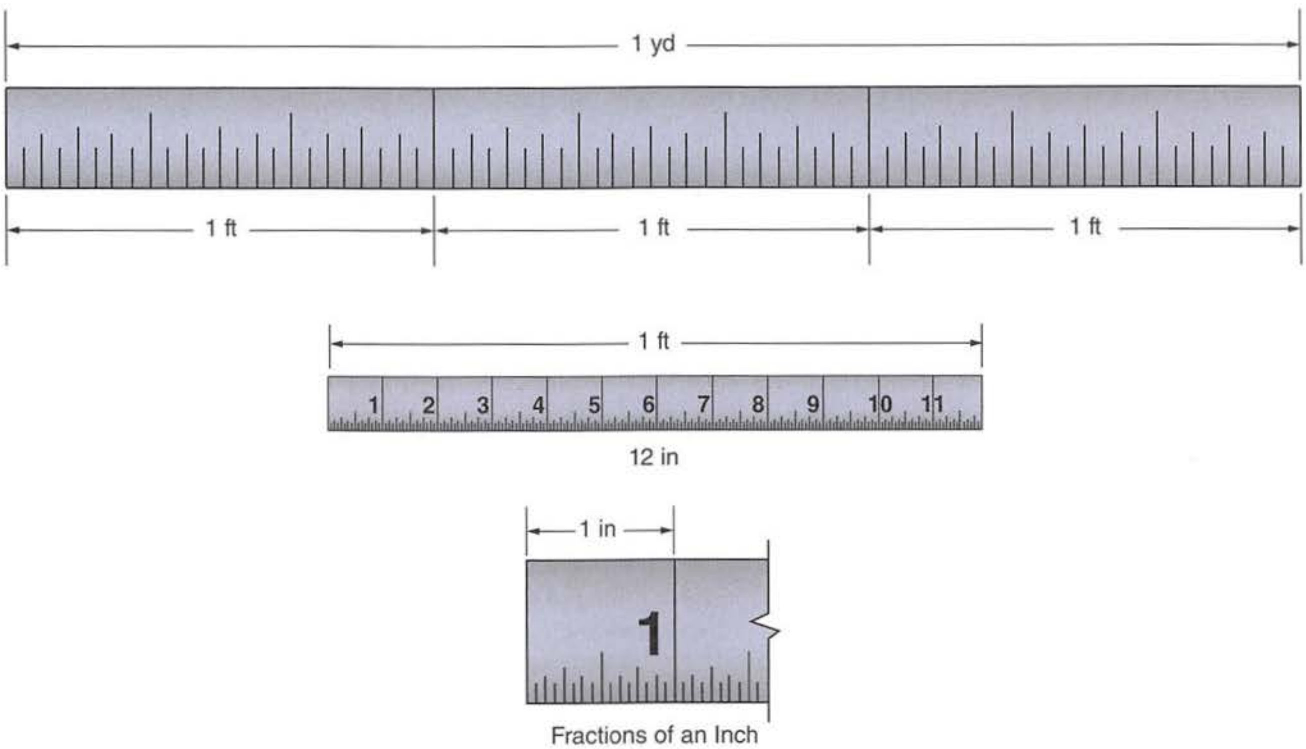
F. $\frac{13}{16} - \frac{2}{5}$

G. $\frac{2}{3} \times 2\frac{1}{2}$

H. $1\frac{1}{4} \div \frac{2}{5}$

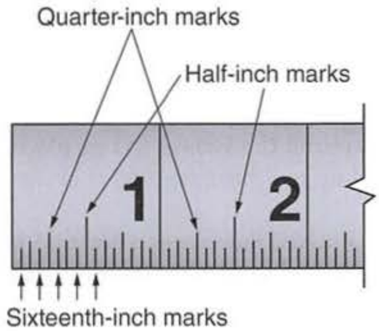
Reading a Ruler

Measuring devices, such as rulers and tape measures, may be marked for measuring inches and fractions of an inch; meters, centimeters, and millimeters; feet, inches and tenths of an inch; or by any other system. The measuring system used to divide the spaces on a measuring device is called the *scale*. The most common linear (in a line) scale in construction uses yards, feet, inches, and fractions of an inch. There are three feet in a yard, 12 inches in a foot, and the inches are most often divided into halves, fourths, eighths, and sixteenths. See **Figure A-3**. The longest marks on the scale indicate inches. The inches on a measuring device may be divided into eighths, sixteenths, or even thirty-seconds. The second longest marks on the scale represent halves, the next longest represent fourths, and so on. See **Figure A-4**. The first step in reading the scale is to determine what the



Goodheart-Willcox Publisher

Figure A-3.



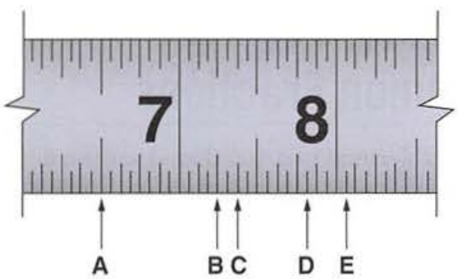
Goodheart-Willcox Publisher

Figure A-4.

smallest marks on the scale represent. Count down from the whole inch to the halves, then the quarters, the eighths, sixteenths, and thirty-seconds, if they are used. Then count the number of marks from the last inch mark to the mark you are reading.

Practice A-3

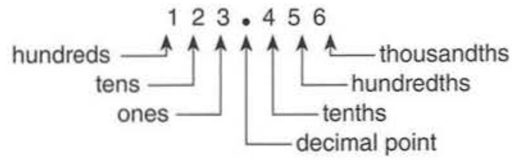
What measurements are represented by the letters on this figure?



Decimal Fractions

Decimal fractions are commonly called simply *decimals.* The decimal system uses increments of 10. Decimal fractions are fractions whose denominators are multiples of 10. If the denominator is 10, the fraction is tenths. If the denominator is 100, the fraction is so many hundredths.

Decimal fractions are often written on a single line with a dot separating digits representing one or more from the decimal fraction. The dot between the whole number and the decimal fraction is the *decimal point.* Every place to the left of the decimal point increases the value of the digit in that place tenfold. That is why the second place to the left of the decimal point is called the tens place and the third place to the left is the hundreds place, etc. Moving to the right of the decimal point the place values decrease tenfold. A decimal fraction of $\frac{5}{10}$ can be written as 0.5. A decimal fraction of $\frac{12}{1000}$ can be written as 0.012.



The value of the number in the example above is one hundred twenty-three and four hundred fifty-six thousandths.

Adding and Subtracting Decimals

To add decimals, line up the decimal points in a column, add the numbers, and put the decimal point in the result in the decimal point column.

Example:

```
   1.4
  19.2
+ 31.7
------
  52.3
```

Subtracting decimals is very similar. Line up the decimal points in the problem and the answer and subtract as usual.

Example:

```
  27.74
-  2.23
-------
  25.51
```

If there are more decimal places in the number being subtracted than there are in the number it is being subtracted from, zeroes can be added to the right without affecting the value of the number.

Example:

```
  5.70  <-- added zero
- 2.02
------
  3.68
```

Multiplying Decimals

Decimals are multiplied the same as whole numbers, except for the placement of the decimal point in the product (answer). Add the number of decimal places to the right of the decimal point in both the number being multiplied and the number it is being multiplied by. The decimal point should be placed that many places to the left in the product.

Example:

```
    12.25  <-- two decimal places to the right
  ×  3.75  <-- two decimal places to the right
  -------      (total of four decimal places)
     6125
    8575
   3675
 --------
  45.9375  <-- decimal point is four places
               to the left
```

Dividing Decimals

Dividing decimals is also much like dividing whole numbers, except for keeping track of the placement of the decimal point. As a reminder, the number being divided is the

dividend, the number it is divided by is the divisor, and the answer is the quotient. To start the division problem, move the decimal point in the divisor all the way to the right. Move the decimal point in the dividend the same number of places to the right. Add zeroes to the right of the dividend, if necessary. Divide as you would for whole numbers.

Example:

divisor ⟶ .4 ⟌20 ⟵ dividend

4. ⟌200. move decimal points

$$\begin{array}{r} 50 \\ 4\,\overline{)\,200} \\ \underline{20} \\ 00 \end{array}$$

Practice A-4

Test your skills with the following problems.

A. $\begin{array}{r} 2.12 \\ 17.01 \\ \underline{+\ 9.05} \end{array}$ B. $\begin{array}{r} 34.09 \\ 12.125 \\ \underline{+\ 2.899} \end{array}$

C. $\begin{array}{r} 18.48 \\ \underline{-\ 12.25} \end{array}$ D. $\begin{array}{r} 134.02 \\ \underline{-\ 8.14} \end{array}$

E. $\begin{array}{r} 5.25 \\ \underline{\times\ 5} \end{array}$ F. $\begin{array}{r} 15.34 \\ \underline{\times\ 6.25} \end{array}$

G. $35 \div .07$ H. $2.25 \div .25$

Converting Common Fractions to Decimal Fractions and Rounding Off

To change a common fraction to a decimal fraction, divide the numerator by the denominator.

Example: Change 1/4 to a decimal fraction.

$$\begin{array}{r} .25 \\ 4\,\overline{)\,1.00} \\ \underline{8} \\ 20 \end{array}$$

$$\frac{1}{4} = .25$$

Sometimes the division yields a number with a repeating decimal.

Example: Change 2/3 to a decimal.

$$\begin{array}{r} .666 \\ 3\,\overline{)\,2.000} \\ \underline{1\,8} \\ 20 \\ \underline{18} \\ 20 \end{array}$$

These numbers should be rounded off to the desired number of places. When rounding off, the last digit should be increased by 1 if the next digit is 5 or more. If the next digit is less than 5, the last digit used stays the same. In the above example, round the answer to two places. The second digit is rounded up to 7 because .666 is closer to .67 than it is to .66.

To convert a mixed number as a decimal, keep the whole number as is and convert the fractional part as above.

Express $12\frac{3}{4}$ as a decimal.

$$12 + \begin{array}{r} .75 \\ 4\,\overline{)\,3.00} \\ \underline{2\,8} \\ 20 \end{array} = 12.75$$

Practice A-5

Convert these fractions to decimals and round the answers to three places.

A. $\frac{1}{3}$ B. $\frac{22}{7}$ C. $\frac{10}{15}$

Converting Decimals to Common Fractions

To convert a decimal to a fraction, drop the decimal point and write the given number as the numerator. The denominator will be 10, 100, 1000, or 1 with as many 0s as there were places in the decimal number.

Example:

$$.42 = \frac{42}{100} \text{ or } .125 = \frac{125}{1000}$$

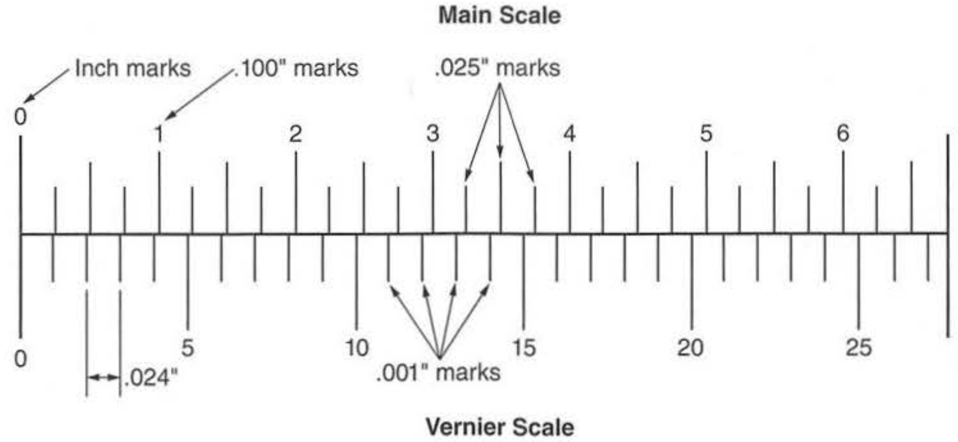
Reading a Vernier Scale

As you learned in Chapter 5, *Measurement*, the vernier scale is named after French mathematician Pierre Vernier. It uses two scales that slide past one another to allow accurate measurements beyond what would be possible with a single scale. The vernier scale principle can be used with inches or the metric system.

Reading an Inch-Based Vernier Scale

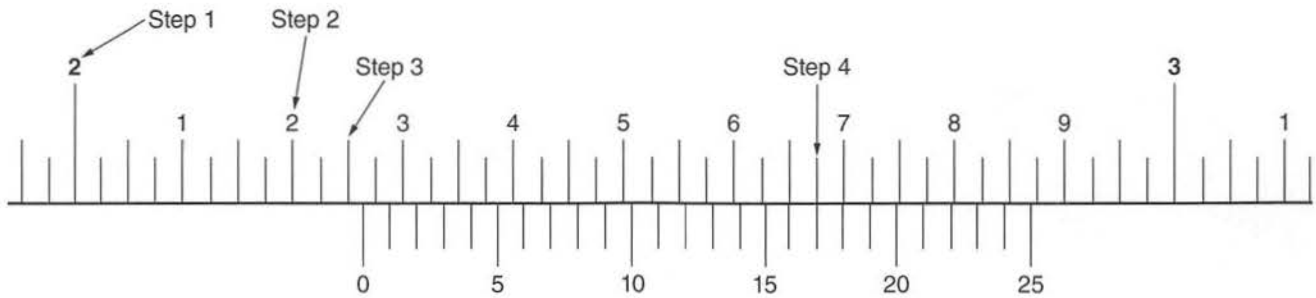
The main scale is divided into inches, tenths of an inch, and 25 thousandths of an inch. The vernier scale is divided into slightly smaller increments, so that only one mark on the vernier scale can be aligned with a mark on the main scale. See **Figure A-5**.

To read the vernier scale, start with the location of the 0 on the scale. See **Figure A-6**. Find the first whole inch mark



Goodheart-Willcox Publisher

Figure A-5.



Goodheart-Willcox Publisher

Figure A-6.

to the left of the vernier scale 0 mark. In **Figure A-6**, that mark is 2″. We will be adding the numbers found in each step, so write down "2″." Next, find the largest 0.1″ mark to the left of the 0 on the vernier scale. In **Figure A-6**, that is the 0.2″ mark. Write that under "2″," with their decimal points aligned for addition later.

2.0
0.2

Add the decimal point after the top 2 and 0s as necessary to help keep things aligned for later addition. The third step is to find the largest 0.025″ mark on the main scale to the left of the 0 on the vernier scale. In this case, that is the second one after the 0.2″ mark, so it represents 0.025″ + 0.025″ or 0.050″. Write that number in your addition column, adding 0s as necessary.

2.000
0.200
0.050

The final reading is on the vernier scale. Find a line on the vernier scale that lines up with any mark on the main scale. In **Figure A-6**, that is the mark representing 17, or 0.017″. Write that in your addition column, adding any necessary 0s, and do the addition.

2.000
0.200
0.050
0.017
2.267

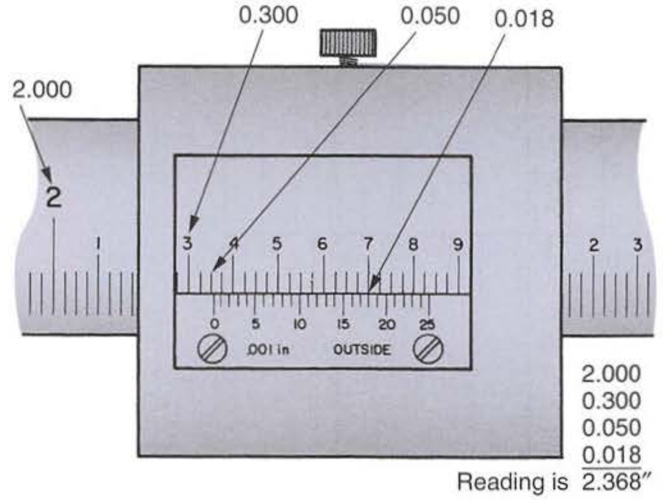
The vernier scale in **Figure A-6** is reading 2.267″.

Review two additional examples of reading a US Conventional vernier scale in **Figure A-7** and **Figure A-8**. In **Figure A-7**, 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″

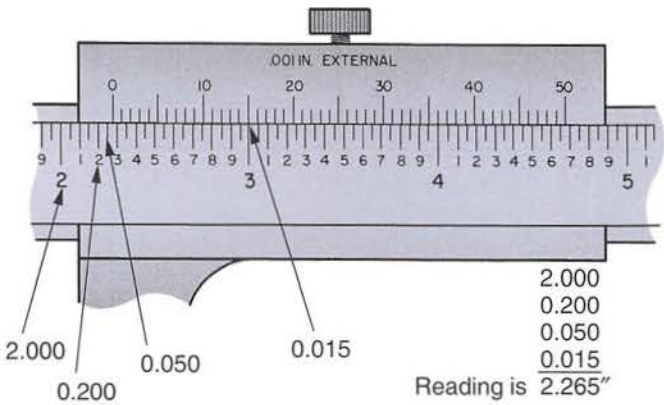
In **Figure A-8**, the "0" line on the vernier plate is

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″



Goodheart-Willcox Publisher

Figure A-7.



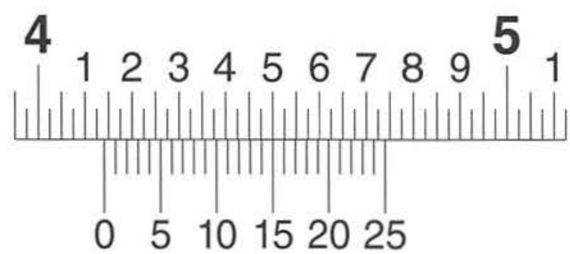
Goodheart-Willcox Publisher

Figure A-8.

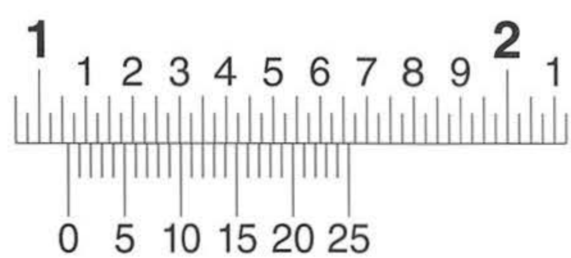
Practice A-6

Take readings from the following vernier scales:

A.



B.



Reading a Metric Vernier Scale

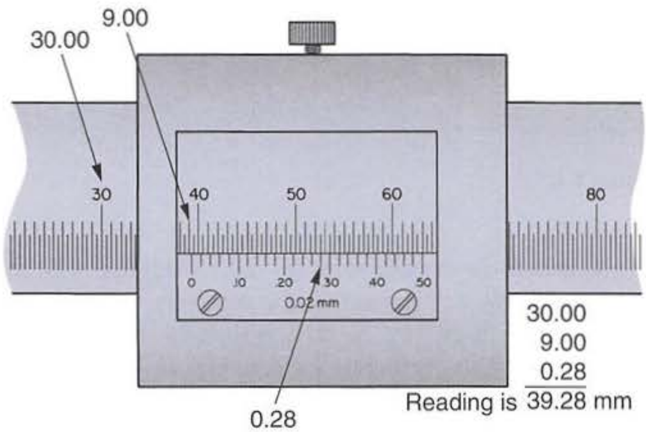
The principles used in reading metric vernier measuring tools are the same as those used for US Conventional measure. However, the readings on the metric vernier scale have 0.02 mm precision. A 25-division metric vernier scale is illustrated in **Figure A-9**, and a 50-division metric vernier scale is shown in **Figure A-10**. Note that each division on both scales corresponds to two-hundredths of a millimeter (0.02 mm).

Reading a Micrometer Caliper

The micrometer caliper is a precision measuring 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.

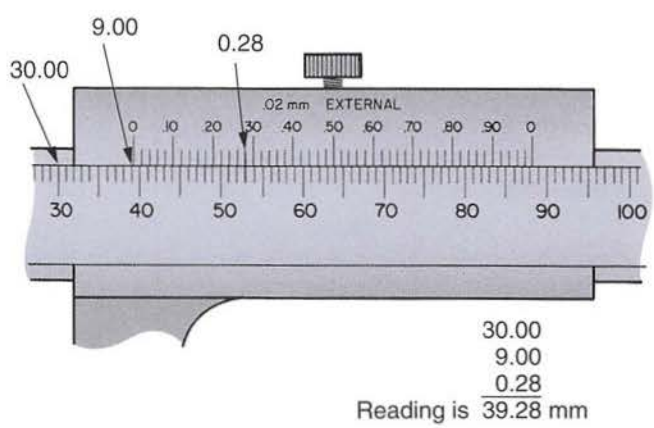
Reading an Inch-Based Micrometer

The micrometer is read by recording the highest number on the sleeve (1 = 0.100, 2 = 0.200, etc.). To this number, add the



Goodheart-Willcox Publisher

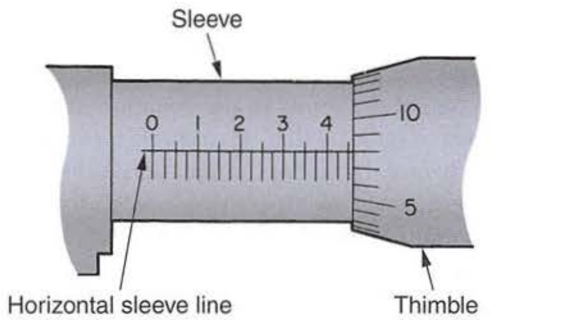
Figure A-9.



Goodheart-Willcox Publisher

Figure A-10.

number of vertical lines visible between the number on the sleeve and the thimble edge (1 = 0.025, 2 = 0.050, etc.). To this total, add the number of thousandths of an inch indicated by the line that corresponds with the horizontal sleeve line. Add the readings from the sleeve and the thimble in **Figure A-11**:

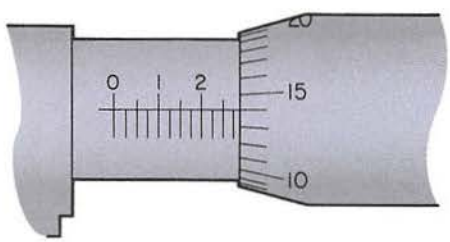


Goodheart-Willcox Publisher

Figure A-11.

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

Another example is presented in **Figure A-12**. As before, add the readings from the sleeve and thimble to determine the total reading:

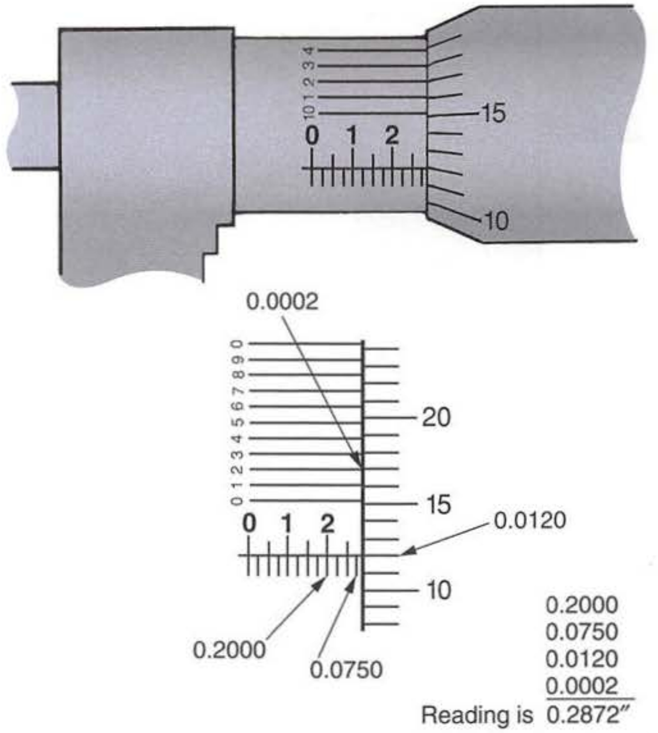


Goodheart-Willcox Publisher

Figure A-12.

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

Vernier micrometers have an additional set of lines on the sleeve, **Figure A-13**, to allow even more precision. As before, add the readings from the sleeve and thimble, then add the 1/10,000″ (0.0001″) reading to find the total.

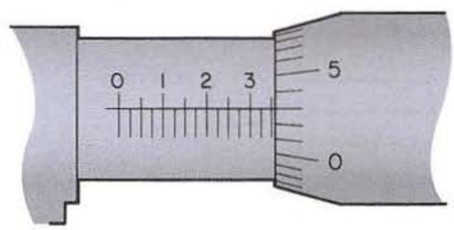


Goodheart-Willcox Publisher

Figure A-13.

Practice A-7

Take the readings from these micrometers:

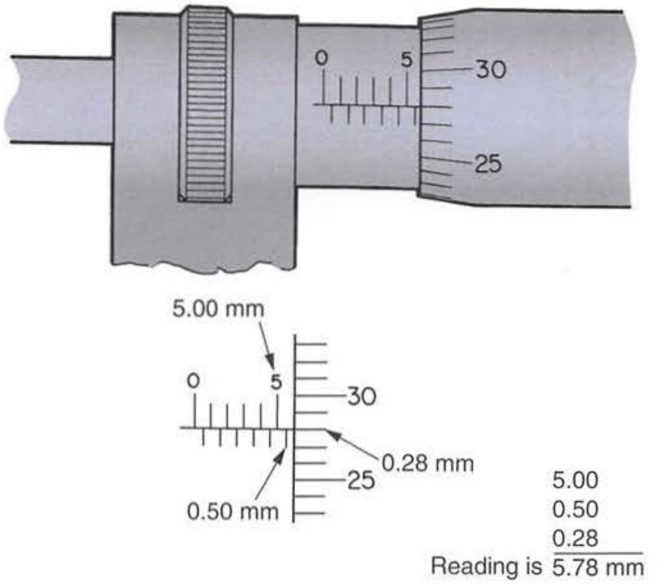


A.

B.

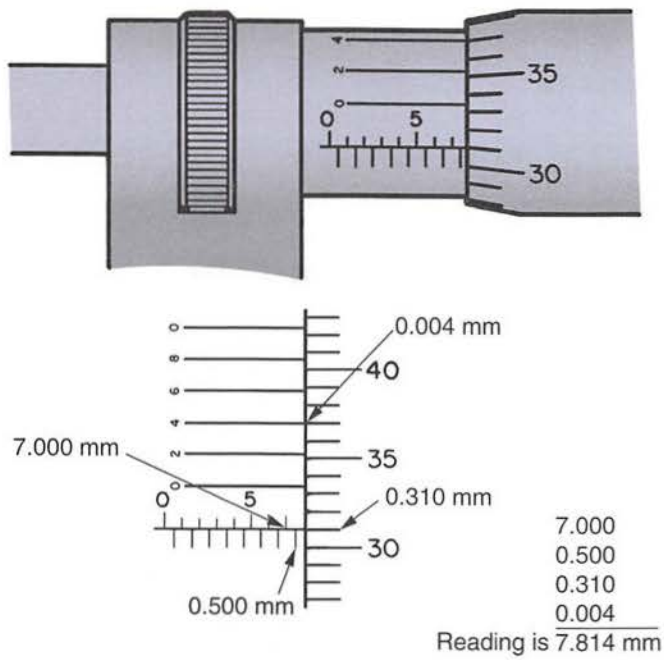
Reading a Metric Micrometer

Metric micrometers, **Figure A-14**, and vernier micrometers, **Figure A-15**, are read much like inch-based micrometers. The only difference is that you will read tenths, hundredths, thousandths, and two-thousandths of a millimeter, instead of fractions of an inch.



Goodheart-Willcox Publisher

Figure A-14.



Goodheart-Willcox Publisher

Figure A-15.

Equations

An *equation* is a mathematical statement that two things have the same or equal value. An equation can be thought of as a mathematical sentence. The words of the sentence are mathematical values called *terms*. An equation is always written with an equal sign (=). For example, $3 + 4 = 7$. In that statement 3, 4, and 7 are terms. The statement says that 3 plus 4 has the same value as 7. We work with equations quite often in doing machining math. Many useful formulas are stated as equations, as is evident in the next section. Equations can be used to find the value of one unknown term when the other values in the equation are known. For example, if you know that a truck is loaded with 10 bundles of shingles weighing 80 lb each, an unknown weight of sheet metal, and the total load is 1000 lb, you can find the weight of the metal with the following equation:

$$(80 \text{ lb} \times 10) + \text{weight of metal} = 1000 \text{ lb}$$

The (80 lb × 10) represents the total weight of the shingles. It is one of the terms in the equation. It is enclosed in parentheses to indicate that it is a single term that should be computed before the rest of the equation. Whenever a mathematical term, such as (80 lb × 10), is enclosed in parentheses, that computation should be done first. Now write the equation with the shingle weight computed:

$$800 \text{ lb} + \text{weight of metal} = 1000 \text{ lb}$$

When a mathematical operation is done on one side of an equation, the equation remains a true statement if the same thing is done on the other side of the equal sign. If we subtract 800 lb from both sides of our equation, it is still a true equation:

$$800 \text{ lb} + \text{weight of metal} - \mathbf{800\ lb} = 1000 \text{ lb} - \mathbf{800\ lb}$$
$$\text{Weight of metal} = 200 \text{ lb}$$

Practice A-8

Find the unknown value in each equation.

A. $\text{cost} = (\$.60 - \$.04) \times 5$

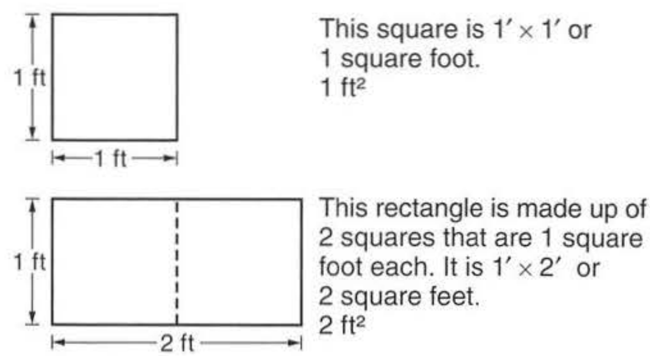
B. $X = \dfrac{3}{4} + 20$

C. $240 \text{ lb} = 2 \times \text{weight of crate}$

D. $\dfrac{1}{4} \div \dfrac{2}{3} = Y$

Area Measure

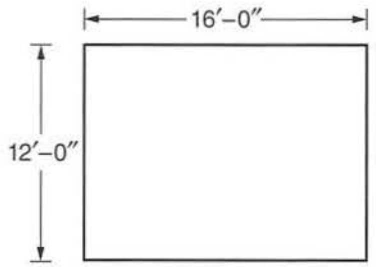
The area of a surface is always measured in units of square inches, square meters, square feet, etc. When a number is squared, that means it is multiplied by itself. For example, 3 squared is 9. Square units are written with a superscript 2, indicating that it is units × units.



Finding the Area of Squares and Rectangles

The area of a square or a rectangle is the number of units it is wide multiplied by the number of units it is long.

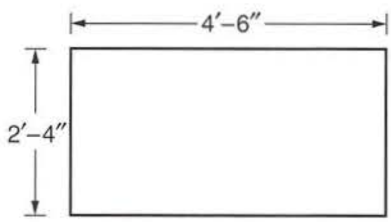
Example: Find the area of this rectangle.



$$12 \text{ ft} \times 16 \text{ ft} = 192 \text{ ft}^2$$

The width and length must be expressed in the same units.

Example: To find the area of this rectangle, convert all feet to inches, then multiply.



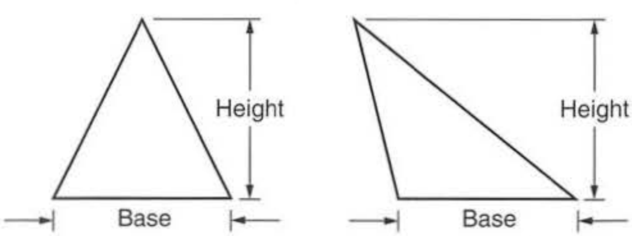
$$2'\text{–}4'' = 24'' + 4'' = 28''$$
$$4'\text{–}6'' = 48'' + 6'' = 54''$$
$$28 \text{ in} = 54 \text{ in} = 1512 \text{ in}^2$$

A square foot is 12 inches × 12 inches, or 144 square inches. If an area is given as a large number of square inches, it can be converted to square feet by dividing it by 144.

Example: In the example above, the area of the rectangle is 1512 in^2. If that is divided by 144, we find that it is 10.5 ft^2.

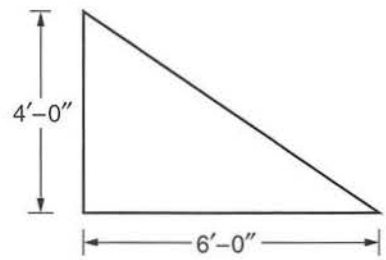
Finding the Area of Triangles

To find the area of a triangle, it is necessary to know the names of two parts of a triangle.



To find the area of any triangle, multiply the height times 1/2 the base.

Example: Find the area of this triangle.

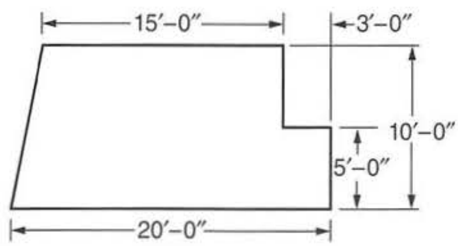


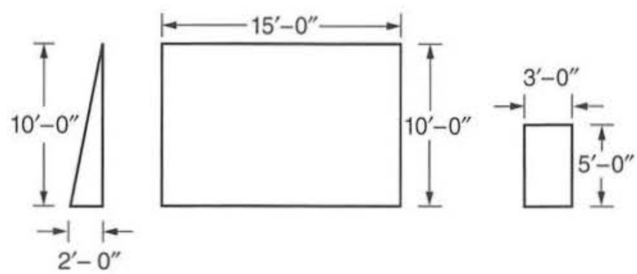
$$4 \text{ ft} \times \frac{6 \text{ ft}}{2} = 4 \text{ ft} \times 3 \text{ ft} = 12 \text{ ft}^2$$

Another way to achieve the same results is to multiply the base times the height, then divide that by 2.

$$4 \text{ ft} \times 6 \text{ ft} = 24 \text{ ft}^2$$
$$24 \text{ ft}^2 \div 2 = 12 \text{ ft}^2$$

Some figures may be made up of squares, rectangles, and triangles of varying sizes. To find the area of such a figure, break it into its various parts and find the area of each part, then add those areas.



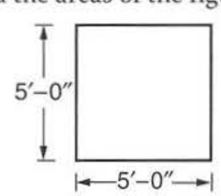


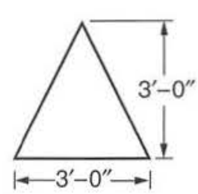
Triangle	Rectangle	Rectangle
$10 \times 2 = 20$	$10 \times 15 = 150 \text{ ft}^2$	$3 \times 5 = 15 \text{ ft}^2$
$20 \div 2 = 10 \text{ ft}^2$		

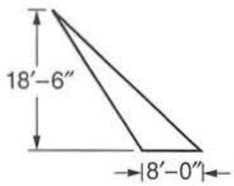
$$10 \text{ ft}^2 + 150 \text{ ft}^2 + 15 \text{ ft}^2 = 175 \text{ ft}^2$$

Practice A-9

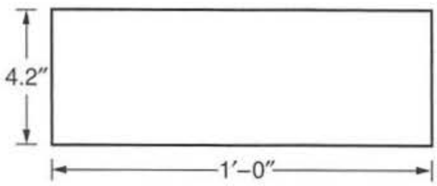
Find the areas of the figures.

A.


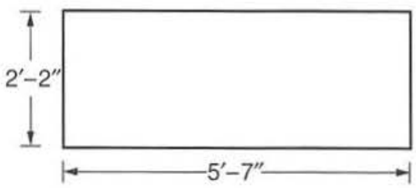
B.




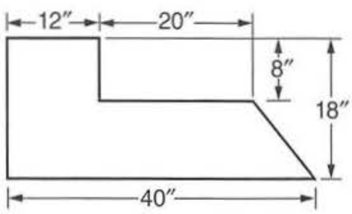
C.



D.



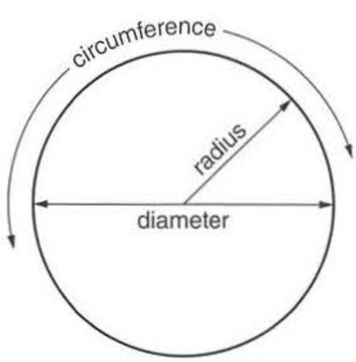
E.



F.

Finding the Circumference and Area of a Circle

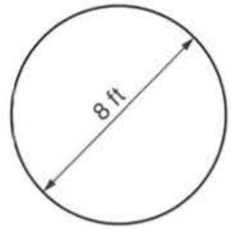
The distance from a circle's center point to its outer edge is its *radius*. The total distance across a circle through its center point is its *diameter*.



Many calculations involving circles or parts of circles use a constant of approximately 22/7, or 3.1416. The Greek letter π (pronounced "pie") is used to represent this constant. It is a constant because it never changes, regardless of the dimensions of the circle.

The *circumference* of a circle is its perimeter. To find the circumference of a circle, multiply the diameter by π. This is the same as multiplying the radius times 2 and multiplying that product times π.

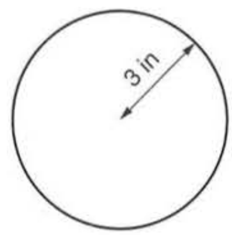
Example: Find the circumference of a circle with a diameter of 8′.



Circumference $= \pi \times$ diameter $= 3.1416 \times 8$ ft
3.1416×8 ft $= 25.1328$ ft
or
Circumference $= 2 \times 4$ ft $\times \pi = 8$ ft $\times 3.1416$

The area of a circle is found by multiplying π times the radius squared (the radius times the radius).

Example: Find the area of a circle with a radius of 3″.

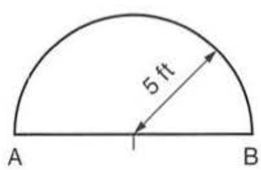


Area $= \pi \times$ radius^2 (or radius $\times$ radius)
Area $= 3.1416 \times 3$ in $\times 3$ in $= 3.1416 \times 9$ in^2
Area $= 28.2744$ in^2

Notice that in both of the examples using π the answer is rounded off to four decimal places. That is because π was rounded to four places. So the answer cannot be accurate to more than that many places.

Many of the shapes encountered in the trades are semicircles or even quarters of a circle. The areas and perimeters of these shapes can be found using the formulas for circles and dividing the result in half for a semicircle or by 4 for a quarter circle.

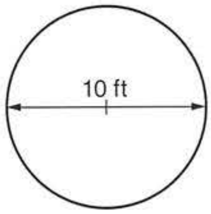
Example: Find the perimeter of the semicircular shape.



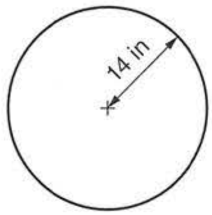
Diameter $= 2 \times 5$ ft $= 10$ ft
Circumference of circle $= \pi \times 10$ ft $= 31.416$ ft
Circumference of circular portion of figure $= 15.708$ ft
Length of line AB is 10 ft
Add 15.708 ft $+ 10$ ft $= 25.708$ ft

Practice A-10

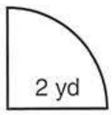
Find the perimeter and the area of each of these figures.



A.



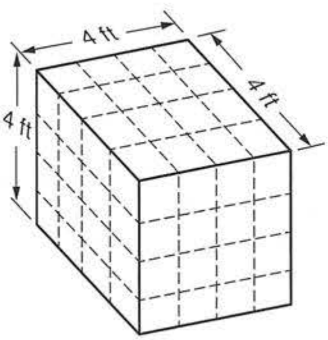
B.



C. Quarter circle

Volume Measure

The volume of a solid is always measured in units of cubic inches, cubic meters, cubic feet, etc. When a number is cubed, that means it is multiplied by itself, then by itself again. For example, 3 cubed is 27 ($3 \times 3 \times 3$). Cubic units are written with a superscript 3, indicating that it is units × units × units.

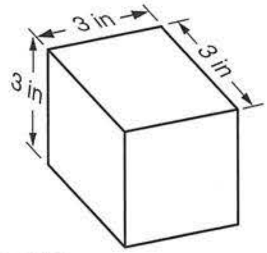


$4\text{ ft} \times 4\text{ ft} \times 4\text{ ft} = 64$ cubic ft or 64 ft^3

This cube is made up of 64 individual cubes, each measuring 1 foot by 1 foot by 1 foot.

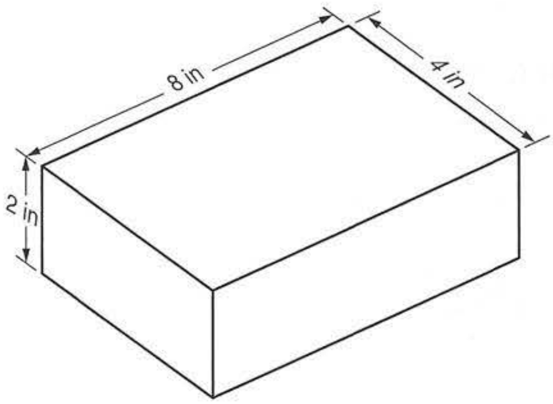
As long as a solid (a three-dimensional shape) has the same size and cross-section shape throughout its depth, its volume can be found by multiplying the area of one surface by the depth from that surface. To find the volume of a *cube* (all edges are the same size) or a rectangular solid (a rectangle with a third dimension) multiply the length, width, and height.

Example:



3-inch cube

Volume $= 3\text{ in} \times 3\text{ in} \times 3\text{ in} = 27\text{ in}^3$

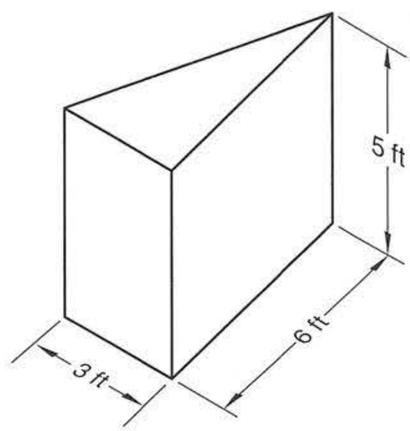


$4\text{ in} \times 8\text{ in} \times 2\text{ in}$ rectangular solid

Volume $= 4\text{ in} \times 2\text{ in} \times 8\text{ in} = 64\text{ in}^3$

A solid with two opposite triangular faces is called a *triangular prism*. A solid with two circular faces is a *cylinder*. The volume of a triangular prism or a cylinder is found by multiplying the area of its face by its height.

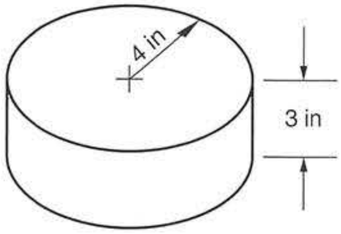
Example:



Triangular prism

Area of face $= \frac{1}{2} \times 3\text{ ft} \times 6\text{ ft} = 9\text{ ft}^2$

Volume $= 9\text{ ft}^2 \times 5\text{ ft} = 45\text{ ft}^3$



Cylinder

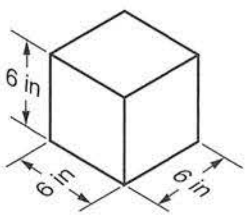
Area of face = $\pi \times 4 \text{ in} \times 4 \text{ in} = 50.2656 \text{ in}^2$

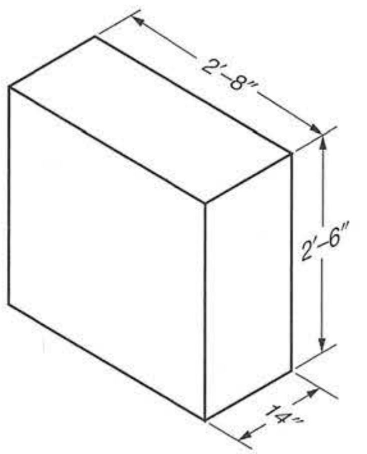
Round to 50.27 in^2

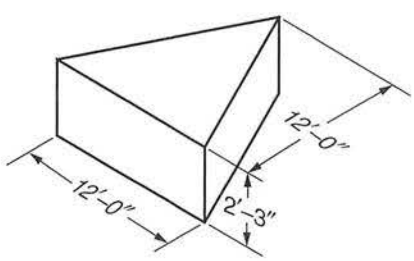
Volume = $50.27 \text{ in}^2 \times 3 \text{ in} = 150.81 \text{ in}^3$

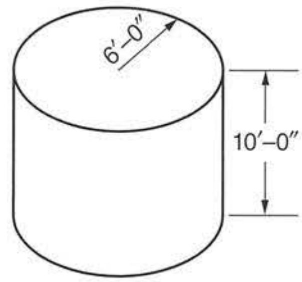
Practice A-11

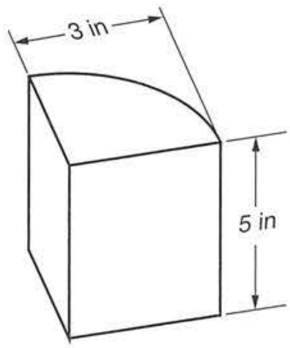
Find the volume of each solid.

A.


B.


C.


D.


E.


Quarter cylinder

Exponents

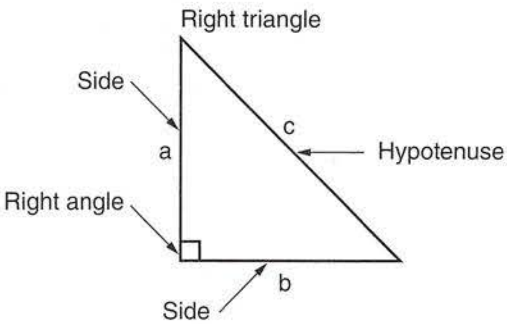
When a number is squared or cubed, the little superscript number written to the right is called an *exponent*. For example, in the number 10^2, the exponent is 2, indicating that the number is 10 multiplied by itself. Another way of saying this is "10 to the second power." If the number were to be $10 \times 10 \times 10$, it could be written as 10^3 and it could be called "10 cubed" or "10 to the third power." Exponents of 2 and 3 have names—squared and cubed, respectively—because they are the exponents used with area and volume measure. Higher exponents are only referred to as powers. For example, 10^5 is read as "10 to the 5th power." It is easier than saying "$10 \times 10 \times 10 \times 10 \times 10$." Both forms of that number equal 100,000.

Practice A-12

A. What is 12 squared?
B. What is 8.5 cubed?
C. What is 4 to the 6th power?
D. What are two other ways to write $3 \times 3 \times 3 \times 3$?

Working with Right Angles

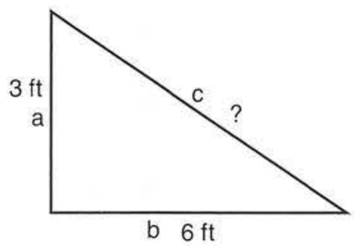
A *right angle* is 90°. A triangle having a right angle is called a *right triangle*. It will be helpful to know a few terms associated with right triangles.



A right triangle can only have one 90° corner. The total of all three corners is always 180°, so if one is 90°, the other two must add up to 90° together.

The *Pythagorean Theorem* is a principle that makes right triangles convenient to work with. Named after the Greek mathematician Pythagoras, the Pythagorean Theorem states that the sum of the squares of the sides of a right triangle is equal to the square of the hypotenuse. To help keep track of the Pythagorean Theorem, it is common to label the two sides *a* and *b* and the hypotenuse *c*. Then the theorem can be stated as an equation: $a^2 + b^2 = c^2$. If the lengths of the two sides of a right triangle are known, the Pythagorean Theorem can be used to find the length of the hypotenuse.

Example:



$$a^2 + b^2 = c^2$$
$$3 \times 3 + 6 \times 6 = c^2$$
$$9 + 36 = c^2$$
$$45 = c^2$$

The square root of 45 = the square root of c^2
(This is one time when a calculator will be a great help.)

$$6.7082 = c$$

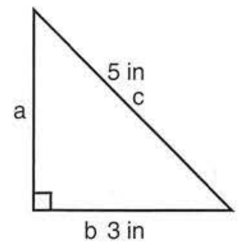
Rounded to 1 decimal place, the hypotenuse is 6.7 ft.

The Pythagorean Theorem can be used to find the length of any side of a right triangle if the other two are known. For example, if side *b* and the hypotenuse are known, $a^2 + b^2 = c^2$ can be rearranged to $a^2 = c^2 - b^2$.

Explanation:

- $a^2 + b^2 = c^2$
- The equation stays in balance if you do the same thing on both sides of the equal sign.
- Subtract b^2 from both sides of the equation.
- $a^2 + \cancel{b^2} - \cancel{b^2} = c^2 - b^2$
- $a^2 = c^2 - b^2$

Example:



$$a^2 = c^2 - b^2$$
$$a^2 = 25 - 9$$
$$a^2 = 16$$
$$a = 4$$

The same can be done to find side *b* when side *a* and the hypotenuse are known.

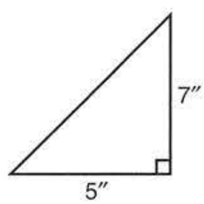
Special Right Triangles

The Pythagorean Theorem can be used to verify that a corner is 90° by measuring along the two sides, then checking the length of the hypotenuse between them. This is usually simplified by using 3 and 4 units as the sides. If 3 and 4 units are used, the hypotenuse of a square corner is 5 units. This is called a 3-4-5 triangle. If 6 and 8 units are used, the hypotenuse is 10 units. This is a multiple of the 3-4-5 triangle. These right triangles are preferred for easy calculations. Note that all sides are whole numbers in a 3-4-5 triangle or any of its multiples.

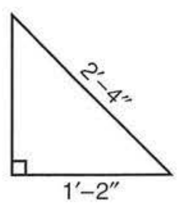
Practice A-13

Find the length of the unknown side to the nearest $\frac{1}{10}$ of an inch.

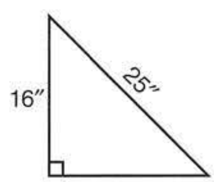
A.



B.



C.



Computing Averages

An average is a typical value of one unit in a group of units. For example, if 4 windows have areas of 12.0 ft^2, 11.2 ft^2, 11.2 ft^2, and 14.5 ft^2; the average size of one of those windows is 12.2 ft^2. The average is computed by adding all of the units and dividing that sum by the number of units in the group.

Example:

12.0 ft^2
11.2 ft^2
11.2 ft^2
14.5 ft^2
48.9 ft^2

```
     12.22
4 ) 48.90
    4
    08
     8
     09
      8
      10
```

The quotient should be rounded off to the same number of decimal places as is used in the problem.

Average window size is 12.2 ft^2

Practice A-14

Compute the averages of these groups.

A. 14, 14.4, 14.5, 15
B. 80 lb, 83 lb, 88 lb, 79.5 lb, 81.6 lb, 84 lb
C. 11 cubic yards, 13 cubic yards, 11.5 cubic yards, 12 cubic yards, 12.8 cubic yards

Percent and Percentage

A *percent* is one part in a hundred. One penny is 1% of a dollar. Twenty-five cents is 25% of a dollar. On the other hand, 25 cents is 50% of a half-dollar because if the half-dollar were divided into 100 parts of 1/2 cent each and the quarter were also divided into 1/2-cent increments, the quarter would equal 50 of those 1/2-cent increments.

Think of percent as hundredths. To find a given percentage of an amount, multiply the amount times the desired number of hundredths.

Example:

Find 12% of $4.40.

12% is 0.12 times the whole.

$4.40 × 12 = $0.528, or 53 cents.

Percent is sometimes interchanged with *percentage* in usage, but there is a slight difference. *Percent* should be used when a specific number is used, such as 15% of the labor force. *Percentage* should be used when no specific number is used with the term, such as a large percentage of our homes are green.

Using what was covered in the section on equations and solving for an unknown, it is possible to calculate the whole if you know the percentage.

Example:

What was the total spent on tools if $22.00 was spent on sales tax and the tax rate is 8%?

1. Write an equation with the facts you know: $22.00 = 8% × total.
2. Write percent as hundredths: $22.00 = 0.08 × total.
3. Divide each side of the equation by 0.08: $275 = total.

Practice A-15

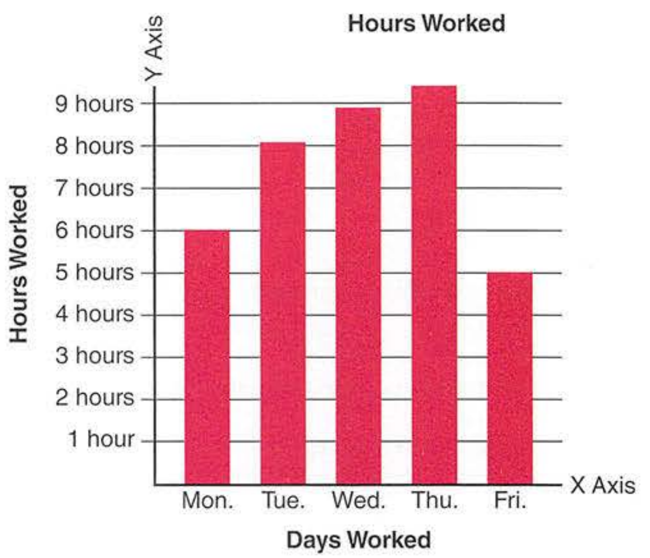
A. What is 10 percent of 225?
B. What is 65 percent of $350.50?
C. If merchandise cost $22.25 and the bill comes to $23.14, what percentage was added for sales tax?
D. If a 500-gallon tank is 60 percent full of water, how much water does it contain?

Graphs

Graphs are frequently used to show mathematical data in a more visual way than simply displaying numbers. There are many kinds of graphs. The most common ones are bar graphs (also called bar charts), line graphs, and circle graphs (also called pie graphs). Graphs are sometimes called charts.

Bar Graphs

As their name implies, *bar graphs* use bars to show data. See **Figure A-16**. Bar graphs have two dimensions. The horizontal line representing the starting point is called the *X axis*.



Goodheart-Willcox Publisher

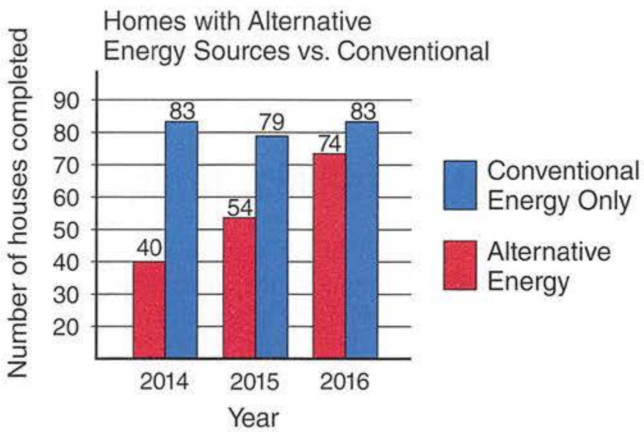
Figure A-16.

The vertical line with graduations indicating the height of the bars is called the *Y axis*. Every bar graph must have four parts: title, labels, scale, and bars. The *title* tells what is graphed. The *labels* on both the X and Y axes tell what kind of data is being shown. The *scale*, often only on the Y axis, tells what the units of measure are. The *bars* show the data numbers.

The tops of the bars might not align with the graduations on the Y axis. For example, the Wednesday bar in **Figure A-16** is between the 8-hour and 9-hour graduations. In this case, the value represented by the bar must be extrapolated. To *extrapolate* a value means to estimate it based on where it falls between two known values. We know that the line above the bar represents 9 hours and the line below the bar top represents 8 hours. The bar is about 3/4 of the way up to the 9-hour line, so the bar represents about 8 3/4 hours.

Some bar graphs have more than one bar for each point on the X axis. The bars generally show related data, but with some difference. This is a good way to compare two sets of data.

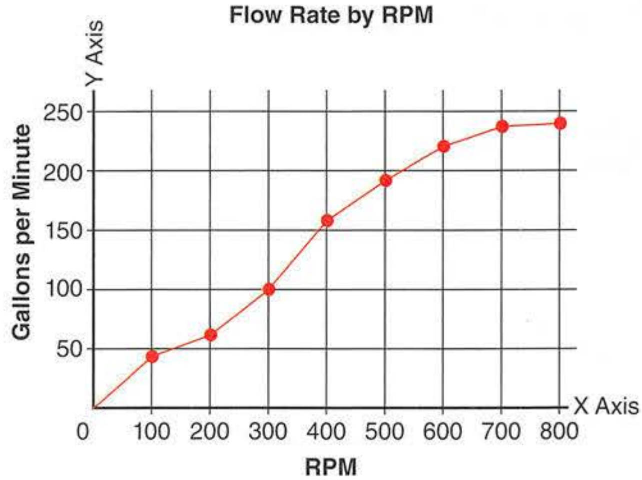
Example:



It is easy to see that the builder in the example completed more than twice as many conventionally powered homes as those with alternative energy sources in 2014. By 2016, although the total number of homes increased greatly, the number of homes with alternative energy sources nearly caught up with the number of homes with only conventional energy sources.

Line Graphs

Line graphs are similar to bar graphs, except the data points are connected by a line. The line graph in **Figure A-17** shows the rate of output from a pump running at various speeds. The flow increases as the speed of the pump increases, but at somewhere around 600 rpm the rate of increase in the flow begins to drop off. By extrapolation (explained above in the discussion on bar graphs), we can see that at 600 rpm the flow is about 220 gallons per minute. At 700 rpm the flow has only increased to about 235 or 240 gallons per minute.

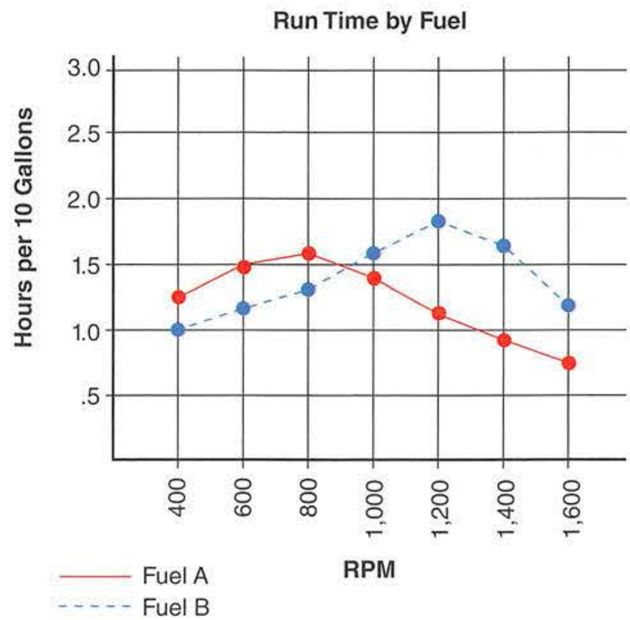


Goodheart-Willcox Publisher

Figure A-17.

Because this is a line graph, it is easy to see the trend by the slope of the line.

A line graph might use two or more lines (based on separate data points) to show a comparison. The line graph in **Figure A-18** compares the fuel efficiency of two different fuels in engines running at different speeds. From this graph it can be seen that if the engine is to be run only at 900 rpm, both fuels have the same run times. If the engine is to be run only at speeds below 900, fuel A is more efficient. If it is to be run only at speeds above 900, fuel B is more efficient.

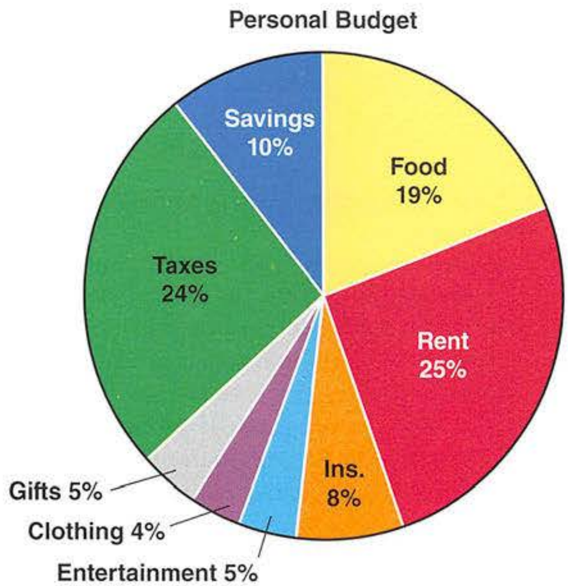


Goodheart-Willcox Publisher

Figure A-18.

Circle Graph or Pie Chart

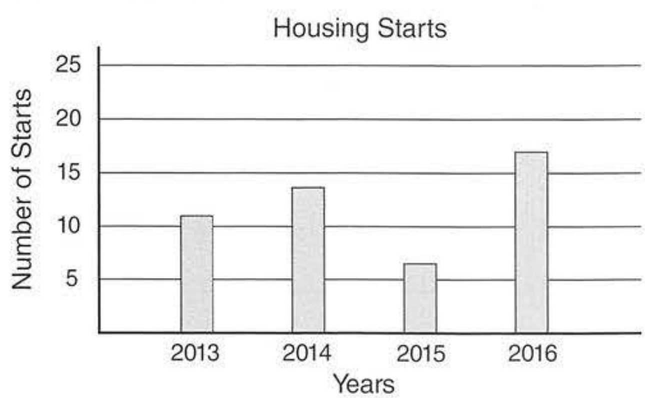
Circle graphs are most commonly called *pie charts* because they resemble a pie divided into slices. A pie chart is useful to show the sizes of the various parts of a whole. See **Figure A-19**. A pie chart has three essential parts: title, key, and circle. The *title* tells what is being graphed or charted. The *key* identifies each of the individual bits of data. The *circle* and its "pie slice" parts make up the foundation of the chart. If any one of these is missing, the chart is meaningless. For example, without the title, the pie chart in **Figure A-19** could be a record of actual spending or a planned budget.



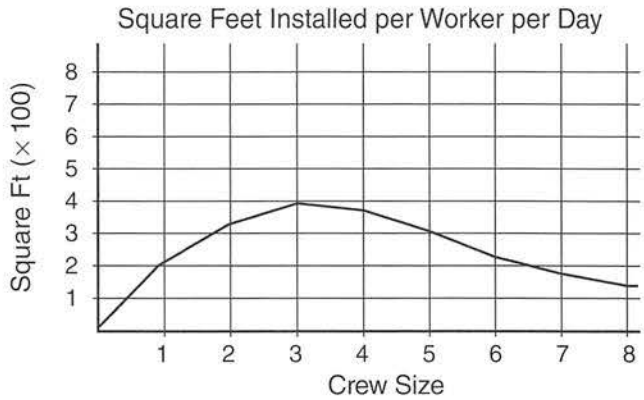
Goodheart-Willcox Publisher

Figure A-19.

Practice A-16



A. What type of graph is shown above?
B. In what year were the smallest number of houses started?
C. How many houses were started in 2014?



D. What size crews installed the most square feet per worker?
E. With two workers, how many square feet can the crew install in one day?
F. Is a person working alone more or less productive than each person in a seven-worker crew?
G. What is shown on the X axis of this graph?
H. Sketch a pie chart showing the following data:
- Building material $4,000
- Labor $3,000
- Overhead $2,000
- Profit $1,000

Mathematics for Machining

The following sections compile the math required for specific topics covered throughout this text. The material is collected here for easy reference and additional practice if desired.

Drilling Speeds

When setting the speed of a drill press, you will need to convert between drill speed, which is typically given in revolutions per minute (rpm), and cutting speed (CS), which is given in feet or meters per minute (fpm or mpm). Use the following formula to determine drill speed when given cutting speed for a drill of a given diameter (D) in inches:

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

Example: At what speed (rpm) must a 1/2″ diameter drill rotate when drilling aluminum? Note that the recommended cutting speed for aluminum is 250 fpm.

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

$$= \frac{4 \times 250}{0.5}$$

$$= 2000 \text{ rpm}$$

Metric problems are solved in a similar way using the following formula:

$$\text{rpm} = \frac{CS \times 1000}{D \times \pi}$$

Where:

CS = Cutting speed (mpm)
D = Drill diameter (mm)
π = 3 (rounded)

Refer to Chapter 12, *Drills and Drilling Machines*, for additional information on this topic.

Practice A-17

A. What rpm should you use when drilling cast iron (recommended CS is 100 fpm) with a 1/4″ drill?

B. What rpm is appropriate for medium-carbon steel with a cutting speed of 25 mpm on a 3 mm diameter drill?

Cutting Speeds on the Lathe

As in drill presses, cutting speed (CS) on the lathe is given in feet per minute (fpm) or meters per minute (mpm). The speed of the work (spindle speed) is given in revolutions per minute (rpm). So, the speed at the outside edge (periphery) of the turning work must be converted to rpm to determine spindle speed.

The formulas and processes for determining rpm on lathes are identical to those used for drills. There is one important distinction to note: when turning work on a lathe, the diameter (D) refers to the diameter of work itself.

Inch-based:

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

Where:

rpm = Revolutions per minute.
CS = Cutting speed recommended for the material being machined (steel, aluminum, etc.) in feet per minute.
D = Diameter of the work in inches. Convert all fractions to decimals.

Metric:

$$\text{rpm} = \frac{CS \times 1000}{D \times \pi}$$

Where:

rpm = Revolutions per minute.
CS = Cutting speed recommended for 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 calculations.)

Example: What spindle speed is required to finish-turn 4″ diameter aluminum alloy?

CS = Table recommends a cutting speed of 1000 fpm for finish-turning aluminum alloy.
D = 4″

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

Example: What spindle speed is required to finish-turn 100 mm diameter aluminum alloy?

CS = Table recommends a cutting speed of 300 mpm for finish-turning aluminum alloy.
D = 100 mm
π = 3

$$\text{rpm} = \frac{CS \times 1000}{D \times 3}$$

$$\text{rpm} = \frac{300 \times 1000}{100 \times 3} = 1000 \text{ rpm}$$

Refer to Chapter 14, *The Lathe*, for additional information on this topic.

Lathe Setover

The offset tailstock method, also called the tailstock setover method, is used for taper turning external tapers. This method of taper turning is imprecise and requires adjusting the offset for each job. The following terms are used when calculating tailstock setover:

D = Diameter at large end of taper
d = Diameter at small end of taper
l = Length of taper
L = Total length of piece
TPI = Taper per inch
TPF = Taper per foot

When taper per inch (TPI) and total length of the piece (L) are known, setover can be calculated using the following formula:

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

Example: What will be the tailstock setover for the following job?

Taper per inch = 0.125
Total length of piece = 8.000

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

$$= \frac{8.000 \times 0.125}{2}$$

$$= 0.500''$$

Note: The same procedure is followed when using metric units. However, all dimensions are in millimeters.

When taper per foot (TPF) is known, it must be converted to taper per inch (TPI). The following formula takes this into account:

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

Even if TPI or TPF is not specified, setover can be calculated using the dimensions of the tapered section. Calculations will be easier if all fractions are converted to decimals. All dimensions must either be in inches or in millimeters (not a mixture of both).

$$\text{Offset} = \frac{L \times (D - d)}{2 \times l}$$

Where:

D = Diameter at large end of taper
d = Diameter at small end of taper
l = Length of taper
L = Total length of piece

Example: Calculate the tailstock setover for the following job.

D = 1.250″
d = 0.875″
l = 3.000″
L = 9.000″

$$\text{Offset} = \frac{L \times (D - d)}{2 \times l}$$

$$= \frac{9.000 \times (1.250 - 0.875)}{2 \times 3.000}$$

$$= \frac{9.000 \times 0.375}{6}$$

$$= 0.562''$$

Refer to Chapter 16, *Cutting Tapers and Screw Threads on the Lathe*, for additional information on this topic.

Practice A-18

A. What will tailstock setover be if TPI is 0.250″ and the total length of the piece is 8.000″?

B. What will the tailstock setover be for the following job?

D = 3.50″
d = 2.75″
l = 8.00″
L = 12.00″

C. What will the tailstock setover be for the following job?

D = 65.0 mm
d = 35.0 mm
l = 250.0 mm
L = 400.0 mm

Three-Wire Thread Measuring

The three-wire method of measuring threads was developed to speed up the measuring process used to check accuracy when machining threaded fasteners.

The three-wire thread measuring formula is as follows:

$$M = D + 3G - \frac{1.5155}{N}$$

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.

Refer to Chapter 16, *Cutting Tapers and Screw Threads on the Lathe*, for additional information on this topic.

Practice A-19

A. Calculate M for a 1/2-20 UNF thread and wire size 0.045″.

B. Calculate M for a 3/8-16 UNC thread and wire size 0.050″.

Milling Speeds and Feeds

Milling cutting speed refers to the distance, measured in feet or meters, that a point (tooth) on the cutter's circumference

moves in one minute. It is expressed in feet per minute (fpm) or meters per minute (mpm). Milling cutting speed depends on the revolutions per minute (rpm) of the cutter. Milling feed is the rate at which work moves into the cutter. It is given in feed per tooth per revolution (ftr).

Example: Determine the approximate cutting speed and feed for a 6″ (152 mm) diameter side cutter (HSS) with 16 teeth, when milling free cutting steel.

Information available:

Recommended cutting speed for free cutting steel (midpoint in range)	= 200 fpm
Recommended feed per tooth (midpoint in range)	= 0.008″
Cutter diameter	= 6″
Number of teeth on cutter	= 16

Determine the speed setting (cutter rpm). Divide the feet per minute by the circumference of the cutter, expressed in feet.

Formula:

$$\text{rpm} = \frac{\text{fpm} \times 12}{\pi D} = \frac{200 \times 12}{3.14 \times 6} = \frac{2400}{18.84} = 127.39 \text{ rpm}$$

Determine feed setting (feed in inches per minute, F). Multiply feed per tooth per revolution by the number of teeth on the cutter and by speed (rpm).

Formula:

$$F = \text{ftr} \times T \times \text{rpm} = 0.008 \times 16 \times 127 = 16.25$$

Refer to Chapter 17, *The Milling Machine*, for additional information on this topic.

Practice A-20

A. Calculate machine speed (rpm) and feed (F) for a 2″ diameter tungsten carbide 8 tooth (T) end mill when machining cast iron. The recommended cutting speed is 190 fpm and feed per tooth (ftr) is 0.004″.

B. Determine machine speed (rpm) and feed (F) for a 5″ diameter HSS side milling cutter with 20 teeth (T) milling free cutting steel. Recommended cutting speed is 200 fpm. Feed per tooth (ftr) is 0.005″.

Cutting a Spur Gear

To cut any gear, you must determine the proper gear cutter to use, the depth of the teeth, and the dividing head setup. Various gear parts can be described mathematically using an array of formulas. Refer to Chapter 18, *Milling Machine Operations*, for additional information on this topic.

For this simple math practice, consider a spur gear. The spur gear has teeth that run straight across the face and are perpendicular to the sides. It is the simplest gear and is widely used.

Example: Calculate the data needed to cut a 40 tooth, 10 diametral pitch gear. First determine diameter (D_0) of blank needed.

Given:

Diametral pitch (P) = 10
Number of teeth (N) = 40

Formula:

$$D_0 = \frac{N + 2}{P} = \frac{40 + 2}{10} = \frac{42}{10} = 4.200 = 4.200'' \text{ diameter}$$

Next, determine the total depth of tooth (h_t) needed. This will be the depth of the cut.

Formula:

$$h_t = \frac{2.157}{P} = \frac{2.157}{10} = 0.216 = 0.216''$$

To check whether the gear is being cut to specifications, calculate the addendum (a):

$$a = \frac{1}{P} = \frac{1}{10} = 0.100 = 0.100''$$

Then find the tooth thickness (t):

$$t = \frac{1.5708}{P} = \frac{1.5708}{10} = 0.157 = 0.157''$$

Practice A-21

Perform the calculations for cutting a spur gear with diametral pitch of 12 and 46 teeth. Find D_0, h_t, a, and t.

Milling a Bevel Gear

Bevel gears are used to change the direction of power between shafts. The example given here deals with a straight tooth bevel gear. As with spur gears, there are many variables associated with bevel gears, which can be described by a collection of formulas. Refer to Chapter 18, *Milling Machine Operations*, for additional information on this topic.

Example: Calculate the tooth thickness and tooth space at the small end of a bevel gear with the following features:

Cone distance (C_r) = 3.535″

Face width (F) = 1.000″

First, determine the bevel gear ratio:

$$\frac{C_s}{C_r}$$

Where:

$C_s = C_r - F$

C_r = Cone distance

F = Face width

Formula:

$$C_s = C_r - F_s = 3.535'' - 1.000'' = 2.535''$$

$$\frac{C_s}{C_r} = \frac{2.535''}{3.535''} = 0.717$$

You obtain the resulting bevel gear ratio:

$$\frac{C_s}{C_r} = 0.717$$

Now, you can calculate tooth thickness and tooth space at the small end. Given tooth thickness and tooth space at the large end:

$$t_L = 0.2618''$$

Formula:

$$t_s = t_L \times \frac{C_s}{C_r} = 0.2618'' \times 0.717 = 0.1877$$

$$= 0.1877''$$

Practice A-22

Perform the calculations for milling a bevel gear with cone distance 2.468″, face width 1.750″, and tooth thickness at the large end 0.3892″.

Tapping Feeds and Speeds

Tapping is another milling operation with variables similar to those used for drilling and turning. Tapping has an added consideration: the tapping feed rate must be an even ratio of the thread lead and the spindle rpm. The tapping feed rate can be calculated using the following formula:

$$F = \text{Lead of tap} \times \text{rpm}$$

Where:

F = tapping feed rate

Lead of tap = 1/thread per inch

rpm = spindle rpm

Example: Calculate the tapping feed rate for 1/2-13 thread and a spindle speed of 260 rpm.

$$F = 1/13 \times 260$$

$$F = 20 \text{ inches per minute}$$

Refer to Chapter 23, *CNC Milling*, for additional information on this topic.

Practice A-23

Calculate the tapping feed rate given 1/4-20 thread and tapping speed 300 rpm.

Brinell Hardness

The Brinell Hardness test is used to measure a metal's resistance to deformation. The test is performed by applying a known load to the surface of the metal through a hardened steel ball of known diameter, D. The diameter of the resulting impression on the metal, d, is then measured and used to calculate the metal's Brinell hardness number, BHN:

$$BHN = \frac{P}{\frac{\pi D}{2}\left(D - \sqrt{D^2 - d^2}\right)}$$

Where:

BHN = Brinell hardness number in kilograms per mm^2.

P = Applied load in kilograms.

D = Diameter of steel ball in millimeters.

d = Diameter of impression in millimeters.

Refer to Chapter 29, *Heat Treatment of Metals*, for additional information on this topic.

Practice A-24

Calculate the Brinell hardness number for a metal that takes on a 4.88 mm diameter impression when a load of 1500 kg is applied through a 10 mm diameter steel ball.

APPENDIX B

Reference Section

The following pages contain a number of tables, charts, and other materials that will be useful as reference in a variety of machining-related areas. To make locating information easier, the material in this section is listed below, along with the page number.

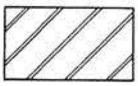
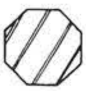
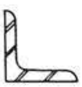
Shapes		Length	How Measured	How Purchased
	Sheet less than 1/4″ thick	Up to 144″	Thickness × width, widths to 72″	Weight, foot, or piece
	Plate more than 1/4″ thick	Up to 20′	Thickness × width	Weight, foot, or piece
	Band	Up to 20′	Thickness × width	Weight or piece
	Rod	12′ to 20′	Diameter	Weight, foot, or piece
	Square	12′ to 20′	Width	Weight, foot, or piece
	Flats	Hot rolled 20′–22′ Cold finished	Thickness × width	Weight, foot, or piece
	Hexagon	12′ to 20′	Distance across flats	Weight, foot, or piece
	Octagon	12′ to 20′	Distance across flats	Weight, foot, or piece
	Angle	Up to 40′	Leg length × leg length × thickness of legs	Weight, foot, or piece
	Channel	Up to 60′	Depth × web thickness × flange width	Weight, foot, or piece
	I-beam	Up to 60′	Height × web thickness × flange width	Weight, foot, or piece

Table title: **Common Shapes of Metals**

Goodheart-Willcox Publisher

Color Codes for Marking Steels

SAE Number	Color Code	SAE Number	Color Code	SAE Number	Color Code	SAE Number	Color Code
Carbon steels		2115	Red and bronze	T1340	Orange and green	3450	Black and bronze
1010	White	2315	Red and blue	T1345	Orange and red	4820	Green and purple
1015	White	2320	Red and blue	T1350	Orange and red	**Chromium steels**	
X1015	White	2330	Red and white	**Nickel-chromium steels**		5120	Black
1020	Brown	2335	Red and white	3115	Blue and black	5140	Black and white
X1020	Brown	2340	Red and green	3120	Blue and black	5150	Black and white
1025	Red	2345	Red and green	3125	Pink	52100	Black and brown
X1025	Red	2350	Red and aluminum	3130	Blue and green	**Chromium-vanadium steels**	
1030	Blue	2515	Red and black	3135	Blue and green	6115	White and brown
1035	Blue	**Molybdenum steels**		3140	Blue and white	6120	White and brown
1040	Green	4130	Green and white	X3140	Blue and white	6125	White and aluminum
X1040	Green	X4130	Green and bronze	3145	Blue and white	6130	White and yellow
1045	Orange	4135	Green and yellow	3150	Blue and brown	6135	White and yellow
X1045	Orange	4140	Green and brown	3215	Blue and purple	6140	White and bronze
1050	Bronze	4150	Green and brown	3220	Blue and purple	6145	White and orange
1095	Aluminum	4340	Green and aluminum	3230	Blue and purple	6150	White and orange
Free cutting steels		4345	Green and aluminum	3240	Blue and aluminum	6195	White and purple
1112	Yellow	4615	Green and black	3245	Blue and aluminum	**Tungsten steels**	
X1112	Yellow	4620	Green and black	3250	Blue and bronze	71360	Brown and orange
1120	Yellow and brown	4640	Green and pink	3312	Orange and black	71660	Brown and bronze
X1314	Yellow and blue	4815	Green and purple	3325	Orange and black	7260	Brown and aluminum
X1315	Yellow and red	X1340	Yellow and black	3335	Blue and orange	**Silicon-manganese steels**	
X1335	Yellow and black	**Manganese steels**		3340	Blue and orange	9255	Bronze and aluminum
Nickel steels		T1330	Orange and green	3415	Blue and pink	9260	Bronze and aluminum
2015	Red and brown	T1335	Orange and green	3435	Orange and aluminum		

Goodheart-Willcox Publisher

Metal Sheet Materials Chart			
Material (Sheet less than 1/4″ thick)	How Measured	How Purchased	Characteristics
Copper	Gage number (Brown & Sharpe and Amer. Std.)	24″ × 96″ sheet or 12″ or 18″ by lineal feet on roll	Pure metal
Brass	Gage number (Brown & Sharpe and Amer. Std.)	24″ × 76″ sheet or 12″ or 18″ by lineal feet on roll	Alloy of copper and zinc
Aluminum	Decimal	24″ × 72″ sheet or 12″ or 18″ by lineal feet on roll	Available as commercially pure metal or alloyed for strength, hardness, and ductility
Galvanized Steel	Gage number (Amer. Std.)	24″ × 96″ sheet	Mild steel sheet with zinc plating, also available with zinc coating that is part of sheet
Black Annealed Steel Sheet	Gage number (Amer. Std.)	24″ × 96″ sheet	Mild steel with oxide coating, hot-rolled
Cold-Rolled Steel Sheet	Gage number (Amer. Std.)	24″ × 96″ sheet	Oxide removed and cold-rolled to final thickness
Tin Plate	Gage number (Amer. Std.)	20″ × 28″ sheet 56 or 112 to pkg.	Mild steel with tin coating
Nickel Silver	Gage number (Brown & Sharpe)	6″ or 12″ wide by lineal feet on roll	Copper 50%, zinc 30%, nickel 20%
Expanded	Gage number (Amer. Std.)	36″ × 96″ sheet	Metal is pierced and expanded (stretched) to diamond shape; also available rolled to thickness after it has been expanded
Perforated	Gage number (Amer. Std.)	30″ × 36″ sheet 36″ × 48″ sheet	Design is cut in sheet; many designs available

Goodheart-Willcox Publisher

Order of Ductility of Metals	
1. Gold	6. Aluminum
2. Platinum	7. Nickel
3. Silver	8. Zinc
4. Iron	9. Tin
5. Copper	10. Lead

Goodheart-Willcox Publisher

Physical Properties of Metals

Metal	Symbol	Specific Gravity	Specific Heat	Melting Point*		Lb per Cubic Inch
				°C	°F	
Aluminum (cast)	Al	2.56	.2185	658	1217	.0924
Aluminum (rolled)	Al	2.71	–	658	1217	.0978
Antimony	Sb	6.71	.051	630	1166	.2424
Bismuth	Bi	9.80	.031	271	520	.3540
Boron	B	2.30	.3091	2300	4172	.0831
Brass	–	8.51	.094	–	–	.3075
Cadmium	Cd	8.60	.057	321	610	.3107
Calcium	Ca	1.57	.170	810	1490	.0567
Chromium	Cr	6.80	.120	1510	2750	.2457
Cobalt	Co	8.50	.110	1490	2714	.3071
Copper	Cu	8.89	.094	1083	1982	.3212
Columbium	Cb	8.57	–	1950	3542	.3096
Gold	Au	19.32	.032	1063	1945	.6979
Iridium	Ir	22.42	.033	2300	4170	.8099
Iron	Fe	7.86	.110	1520	2768	.2634
Iron (cast)	Fe	7.218	.1298	1375	2507	.2605
Iron (wrought)	Fe	7.70	.1138	1500–1600	2732–2912	.2779
Lead	Pb	11.37	.031	327	621	.4108
Lithium	Li	.057	.941	186	367	.0213
Magnesium	Mg	1.74	.250	651	1204	.0629
Manganese	Mn	8.00	.120	1225	2237	.2890
Mercury	Hg	13.59	.032	–39	–38	.4909
Molybdenum	Mo	10.2	.0647	2620	47.48	.368
Monel metal	–	8.87	.127	1360	2480	.320
Nickel	Ni	8.80	.130	1452	2646	.319
Phosphorus	P	1.82	.177	43	111.4	.0657
Platinum	Pt	21.50	.033	1755	3191	.7767
Potassium	K	0.87	.170	62	144	.0314
Selenium	Se	4.81	.084	220	428	.174
Silicon	Si	2.40	.1762	1427	2600	.087
Silver	Ag	10.53	.056	961	1761	.3805
Sodium	Na	0.97	.290	97	207	.0350
Steel	–	7.858	.1175	1330–1378	2372–2532	.2839
Strontium	Sr	2.54	.074	769	1416	.0918
Tantalum	Ta	10.80	–	2850	5160	.3902
Tin	Sn	7.29	.056	232	450	.2634
Titanium	Ti	5.3	.130	1900	3450	.1915
Tungsten	W	19.10	.033	3000	5432	.6900
Uranium	U	18.70	–	1132	2070	.6755
Vanadium	V	5.50	–	1730	3146	.1987
Zinc	Zn	7.19	.094	419	786	.2598

*Circular of the Bureau of Standards No.35, Department of Commerce and Labor

Goodheart-Willcox Publisher

Wire Gages in Decimal Inches

Number of Wire Gage	American or Brown & Sharpe	Washburn & Moen Mfg. Co., AS&W Roebling	Imperial Wire Gage	Stubs' Steel Wire	Birmingham or Stubs' Iron Wire
0000000	...	.4900	.5000	...	...
000000	.5800	.4615	.4640	...	...
00000	.5165	.4305	.4320	...	.500
0000	.460	.3938	.4000	...	.454
000	.40964	.3625	.3720	...	.425
00	.3648	.3310	.3480	...	.380
0	.32486	.3065	.3240	...	.340
1	.2893	.2830	.3000	.227	.300
2	.25763	.2625	.2760	.219	.284
3	.22942	.2437	.2520	.212	.259
4	.20431	.2253	.2320	.207	.238
5	.18194	.2070	.2120	.204	.220
6	.16202	.1920	.1920	.201	.203
7	.14428	.1770	.1760	.199	.180
8	.12849	.1620	.1600	.197	.165
9	.11443	.1483	.1440	.194	.148
10	.10189	.1350	.1280	.191	.134
11	.090742	.1205	.1160	.188	.120
12	.080808	.1055	.1040	.185	.109
13	.071961	.0915	.0920	.182	.095
14	.064084	.0800	.0800	.180	.083
15	.057068	.0720	.0720	.178	.072
16	.05082	.0625	.0640	.175	.065
17	.045257	.0540	.0560	.172	.058
18	.040303	.0475	.0480	.168	.049
19	.03589	.0410	.0400	.164	.042
20	.031961	.0348	.0360	.161	.035
21	.028462	.0317	.0320	.157	.032
22	.025347	.0286	.0280	.155	.028
23	.022571	.0258	.0240	.153	.025
24	.0201	.0230	.0220	.151	.022
25	.0179	.0204	.0200	.148	.020
26	.01594	.0181	.0180	.146	.018
27	.014195	.0173	.0164	.143	.016
28	.012641	.0162	.0148	.139	.014
29	.011257	.0150	.0136	.134	.013
30	.010025	.0140	.0124	.127	.012
31	.008928	.0132	.0116	.120	.010
32	.00795	.0128	.0108	.115	.009
33	.00708	.0118	.0100	.112	.008
34	.006304	.0104	.0092	.110	.007
35	.005614	.0095	.0084	.108	.005
36	.005	.0090	.0076	.106	.004
37	.004453	.0085	.0068	.103	...
38	.003965	.0080	.0060	.101	...
39	.003531	.0075	.0052	.099	...
40	.003144	.0070	.0048	.097	...

Goodheart-Willcox Publisher

Cutting Speeds for Round Stock

Diameter	Material	Roughing Cut (rpm)	Finishing Cut (rpm)	Threading (rpm)	Diameter	Material	Roughing Cut (rpm)	Finishing Cut (rpm)	Threading (rpm)
1/8″	Machine steel/bronze	2880	3200	1020	1 1/2″	Machine steel/bronze	240	270	67
	Cast iron	1920	2560	800		Tool steel	134	200	53
	Tool steel (annealed)	1600	2400	640		Brass	400	534	134
	Brass	4800	6400	1600		Aluminum	534	800	134
	Aluminum	6400	9600	1600					
3/16″	Machine steel/bronze	2880	3200	1120	1 3/4″	Machine steel/bronze	205	230	80
	Tool steel	1600	2400	640		Tool steel	115	170	50
	Brass	4800	6400	1600		Brass	340	450	115
	Aluminum	6400	9600	1600		Aluminum	456	680	115
1/4″	Machine steel/bronze	1440	1600	560	2″	Machine steel	180	200	50
	Tool steel	800	1200	320		Tool steel	100	150	40
	Brass	2400	3200	800		Brass	300	400	100
	Aluminum	3200	4800	800		Aluminum	400	600	100
3/8″	Machine steel/bronze	960	1066	270	2 1/2″	Machine steel	141	160	56
	Tool steel	540	800	220		Tool steel	80	120	32
	Brass	1700	2100	530		Brass	240	320	80
	Aluminum	2130	3200	540		Aluminum	320	480	80
1/2″	Machine steel/bronze	720	800	280	3″	Machine steel	120	140	40
	Tool steel	400	600	160		Tool steel	65	100	40
	Brass	1200	1600	400		Brass	200	270	65
	Aluminum	1600	2400	400		Aluminum	270	400	65
5/8″	Machine steel/bronze	576	640	160	3 1/2″	Machine steel	103	115	40
	Tool steel	320	480	200		Tool steel	60	85	23
	Brass	960	1280	320		Brass	171	228	57
	Aluminum	1280	192	320		Aluminum	228	342	57
3/4″	Machine steel/bronze	500	550	176	4″	Machine steel	90	100	35
	Tool steel	266	400	106		Tool steel	50	75	20
	Brass	800	1066	266		Brass	150	200	50
	Aluminum	1066	1600	266		Aluminum	200	300	50
1″	Machine steel/bronze	360	400	140	4 1/2″	Machine steel	80	90	31
	Tool steel	200	300	80		Tool steel	45	67	18
	Brass	600	800	200		Brass	133	178	45
	Aluminum	800	1200	200		Aluminum	178	267	45
1 1/4″	Machine steel	288	320	112	5″	Machine steel	72	80	28
	Tool steel	160	240	64		Tool steel	40	58	16
	Brass	480	640	160		Brass	120	160	40
	Aluminum	640	960	160		Aluminum	160	240	40

Goodheart-Willcox Publisher

Rules for Determining Speeds and Feeds

To Find	Having	Rule	Formula
Speed of cutter in feet per minute (FPM)	Diameter of cutter and revolutions per minute	Diameter of cutter (in inches), multiplied by 3.1416 (π), multiplied by revolutions per minute, divided by 12	$FPM = \frac{\pi D \times RPM}{12}$
Speed of cutter in meters per minute (MPM)	Diameter of cutter and revolutions per minute	Diameter of cutter, multiplied by 3.1416 (π), multiplied by revolutions per minute, divided by 1000	$MPM = \frac{D(mm) \times \pi \times RPM}{1000}$
Revolutions per minute (RPM)	Feet per minute and diameter of cutter	Feet per minute, multiplied by 12, divided by circumference of cutter (πD)	$RPM = \frac{FPM \times 12}{\pi D}$
Revolutions per minute (RPM)	Meters per minute and diameter of cutter in millimeters (mm)	Meters per minute, multiplied by 1000, divided by the circumference of cutter (πD)	$RPM = \frac{MPM \times 1000}{\pi D}$
Feed per revolution (FR)	Feed per minute and revolutions per minute	Feed per minute, divided by revolutions per minute	$FR = \frac{F}{RPM}$
Feed per tooth per revolution (FTR)	Feed per minute and number of teeth in cutter	Feed per minute (in inches or millimeters), divided by number of teeth in cutter × revolutions per minute	$RTR = \frac{F}{T \times RPM}$
Feed per minute (F)	Feed per tooth per revolution, number of teeth in cutter, and RPM	Feed per tooth per revolutions, multiplied by number of teeth in cutter, multiplied by revolutions per minute	$F = FTR \times T \times RPM$
Feed per minute (F)	Feed per revolution and revolutions per minute	Feed per revolution, multiplied by revolutions per minute	$F = FR \times RPM$
Number of teeth per minute (TM)	Number of teeth in cutter and revolutions per minute	Number of teeth in cutter, multiplied by revolutions per minute	$TM = T \times RPM$

RPM = Revolutions per minute
T = Teeth in cutter
D = Diameter of cutter
π = 3.1416 (pi)
FPM = Speed of cutter in feet per minute
TM = Teeth per minute
F = Feed per minute
FR = Feed per revolution
FTR = Feed per tooth per revolution
MPM = Speed of cutter in meters per minute

Goodheart-Willcox Publisher

Recommended Turning Rates for Stainless Steels Using High-Speed Tools

Nature of Stock	Type No.	Speed (sfpm)	Feed (inches per revolution)
Free machining grades	430 F	100–140	0.003–0.005 for finish cuts and up to 0.015 for roughing cuts
	416	90–135	
	303	80–120	
High-carbon grades that are slowed down due to their abrasive action on tools	410	75–115	0.003–0.008
	430	75–115	0.003–0.008
	420	45–85	0.003–0.008
	431	45–85	0.003–0.008
	440	30–60	0.003–0.008
	302		
	304		
	316	45–80	0.004–0.008

Goodheart-Willcox Publisher

Feeds and Speeds for HSS Drills, Reamers, and Taps

Material	Brinell	Drills (sfm)	Drills Point	Drills Feed	Reamers (sfm)	Reamers Feed	Taps (sfm) Threads per Inch 3–7 1/2	8–15	16–24	25–up
Aluminum	99–101	200–250	118°	M	150–160	M	50	100	150	200
Aluminum bronze	170–187	60	118°	M	40–45	M	12	25	45	60
Bakelite	...	80	60-90°	M	50–60	M	50	100	150	200
Brass	192–202	200–250	118°	H	150–160	H	50	100	150	200
Bronze, common	166–183	200–250	118°	H	150–160	H	40	80	100	150
Bronze, phosphor, 1/2 hard	187–202	175–180	118°	M	130–140	M	25	40	50	80
Bronze, phosphor, soft	149–163	200–250	118°	H	150–160	H	40	80	100	150
Cast iron, soft	126	140–150	90°	H	100–110	H	30	60	90	140
Cast iron, medium soft	196	80–110	118°	M	50–65	M	25	40	50	80
Cast iron, hard	293–302	45–50	118°	L	67–75	L	10	20	30	40
Cast iron, chilled*	402	15	150°	L	8–10	L	5	5	10	10
Cast steel	286–302	40–50*	118°	L	70–75	L	20	30	40	50
Celluloid	...	100	90°	M	75–80	M	50	100	150	200
Copper	80–85	70	100°	L	45–55	L	40	80	100	150
Drop forgings (steel)	170–196	60	118°	M	40–45	M	12	25	45	60
Duralumin	90–104	200	118°	M	150–160	M	50	100	150	200
Everdur	179–207	60	118°	L	40–45	L	20	30	40	50
Machinery steel	170–196	110	118°	H	67–75	H	35	50	60	85
Magnet steel, soft	241–302	35–40	118°	M	20–25	M	20	40	50	75
Magnet steel, hard*	321–512	15	150°	L	10	L	5	10	15	25
Manganese steel, 7% – 13%	187–217	15	150°	L	10	L	15	20	25	30
Manganese copper, 30% Mn.*	134	15	150°	L	10–12	L	...	...	...	...
Malleable iron	112–126	85–90	118°	H	...	H	20	30	40	50
Mild steel, .20 –.30 C	170–202	110–120	118°	H	75–85	H	40	55	70	90
Molybdenum steel	196–235	55	125°	M	35–45	M	20	30	35	45
Monel metal	149–170	50	118°	M	35–38	M	8	10	15	20
Nickel, pure*	187–202	75	118°	L	40	L	25	40	50	80
Nickel steel, 3 1/2%	196–241	60	118°	L	40–45	L	8	10	15	20
Rubber, hard	...	100	60-90°	L	70–80	L	50	100	150	200
Screw stock, C.R.	170–196	110	118°	H	75	H	20	30	40	50
Spring steel	402	20	150°	L	12–15	L	10	10	15	15
Stainless steel	146–149	50	118°	M	30	M	8	10	15	20
Stainless steel, C.R.*	460–477	20	118°	L	15	L	8	10	15	20
Steel, .40 to .50 C	170–196	80	118°	M	8–10	M	20	30	40	50
Tool, SAE, and forging steel	149	75	118°	H	35–40	H	25	35	45	55
Tool, SAE, and forging steel	241	50	125°	M	12	M	15	15	25	25
Tool, SAE, and forging steel*	402	15	150°	L	10	L	8	10	15	20
Zinc alloy	112–126	200–250	118°	M	150–175	M	50	100	150	200

*Use specially constructed heavy-duty drills.
Note: Carbon steel tools should be run at speeds 40% to 50% of those recommended for high speed steel.
Spiral point taps may be run at speeds 15% to 20% faster than regular taps.

Goodheart-Willcox Publisher

Feeds and Speeds for HSS Drills in Various Metals

Drill Diameter	Cast Iron		Bronze or Brass		Drop Forgings Alloy Steel Tool Steel Annealed		Drop Forgings Alloy Steel Heat-Treated		Steel Castings		Mild Steel	
	Feed	Speed	Feed	Speed	Feed	Speed	Feed	Speed	Feed	Speed	Feed	Speed
1/16″	.002	4550	.002	9150	.002	3650	.002	2750	.002	3650	.002	4250
	.004	6700	.004	12000	.003	4550	.003	3650	.003	4550	.003	5600
1/8″	.002	2550	.002	4550	.002	1800	.002	1225	.002	1800	.002	2100
	.004	3350	.004	5600	.003	2250	.003	1800	.003	2250	.003	2800
3/16″	.004	1500	.004	3100	.003	1200	.003	900	.003	1200	.003	1400
	.006	2200	.007	5600	.004	1500	.004	1200	.005	1500	.005	1900
1/4″	.004	1150	.004	2300	.003	925	.003	750	.003	925	.003	1050
	.006	1650	.007	2750	.004	1150	.004	925	.005	1150	.005	1500
5/16″	.006	925	.007	1825	.004	725	.004	500	.004	725	.005	850
	.009	1325	.010	2200	.006	925	.005	725	.006	925	.007	1200
3/8″	.006	750	.007	1525	.004	600	.004	400	.004	600	.005	700
	.009	1100	.010	1850	.006	750	.005	600	.006	750	.007	925
7/16″	.009	650	.010	1300	.006	525	.005	350	.006	525	.006	600
	.012	950	.014	1525	.009	650	.006	525	.010	650	.010	800
1/2″	.008	575	.010	1150	.006	375	.005	300	.006	375	.006	525
	.012	850	.014	1375	.009	575	.006	375	.010	575	.010	700
9/16″	.012	500	.014	1000	.008	350	.007	275	.010	350	.010	575
	.016	750	.018	1200	.012	500	.010	350	.014	500	.014	625
5/8″	.012	450	.014	900	.008	300	.007	250	.010	300	.010	425
	.016	675	.018	1100	.012	450	.010	300	.014	450	.014	565
11/16″	.012	410	.014	800	.008	275	.007	225	.010	275	.010	375
	.016	625	.018	1000	.012	410	.010	275	.014	410	.014	525
3/4″	.012	375	.014	750	.008	250	.007	200	.010	250	.010	350
	.016	550	.018	900	.012	375	.010	250	.014	375	.014	475
13/16″	.014	350	.016	700	.010	240	.009	190	.014	240	.014	325
	.020	525	.022	850	.014	350	.012	240	.016	350	.016	450
7/8″	.014	325	.016	650	.010	225	.009	175	.014	225	.014	300
	.020	475	.022	800	.014	325	.012	225	.016	325	.016	400
15/16″	.014	300	.016	625	.010	200	.009	160	.014	200	.014	275
	.020	450	.022	725	.014	300	.012	200	.016	300	.016	375
1″	.014	280	.016	575	.010	185	.009	150	.014	185	.014	265
	.020	425	.022	675	.014	280	.012	185	.016	280	.016	350

Speeds and feeds shown apply to average working conditions and materials. They are recommended with regard to conserving drills and avoiding excessive machine tool wear. Under many conditions, these speeds and feeds may be considerably increased; under others they must be decreased. This is dependent on judgment of operator, and performance obtained. Excessive speeds and feeds will show up by action of machine and drill. Same applies to lower speeds and feeds. Operator will notice whether he/she is getting proper performance by experience, and will advance or retard as case may justify. Feeds and speeds should be changed in proper proportions and a liberal use of cooling compound will increase life of tools.

Never dip a drill into water to cool it while grinding. This will cause tiny checks, or cracks at the cutting edge, which will cause the drill to dull quickly.

Do not leave a drill in after it shows signs of dulling or laboring; then is the time to regrind. Proper grinding is essential.

To determine feed and speed according to the above chart, proceed as follows:

You are going to drill heat-treated drop forgings. We suppose you will use a 1/2″ drill. Follow column down to where the 1/2″ drill meets it; there you will find that a feed from .005 to .006 and a speed of from 300 to 375 rpm are recommended. Start by using .005 feed and 300 rpm. If drill and machine seem to turn smoothly without strain, then both feed and speed can be advanced. Operator will soon find which is best.

Chicago-Latrobe

Formulas for Machining Bar Stock

Surface Speed (feet/minute)	
Round bars	$\frac{\text{Diameter} \times 3.1416 \times \text{rpm}}{12}$
Hexagon bars (distance across corners)	$\frac{\text{Size} \times 3.1416 \times \text{rpm} \times 1.155}{12}$ $\frac{\text{Distance} \times 3.1416 \times \text{rpm}}{12}$
Square bars (distance across corners)	$\frac{\text{Size} \times 3.1416 \times \text{rpm} \times 1.414}{12}$ $\frac{\text{Distance} \times 3.1416 \times \text{rpm}}{12}$
Revolutions (number/minute)	
Round bars	$\frac{\text{sfm} \times 12}{\text{Distance} \times 3.1416}$
Hexagon bars	$\frac{\text{sfm} \times 12}{\text{Size} \times 3.1416 \times 1.155}$ $\frac{\text{sfm} \times 12}{\text{Distance} \times 3.1416}$
Square bars	$\frac{\text{sfm} \times 12}{\text{Size} \times 3.1416 \times 1.414}$ $\frac{\text{sfm} \times 12}{\text{Distance} \times 3.1416}$
Feed (inches/revolution)	$\frac{\text{Feed inches per minute}}{\text{rpm}}$ $\frac{\text{Diameter} \times 3.1416 \times \text{Feed}}{\text{sfm} \times 12}$
Feed (inches/tooth)	$\frac{\text{Feed}}{\text{Number of teeth}}$
Time for actual machining (seconds)	$\frac{\text{Revolutions required} \times 60 \text{ seconds}}{\text{rpm}}$
Machine time	Time for machining + idle time
Tapping or threading time (seconds)	$\frac{\text{Number of threads} \times 60 \text{ seconds}}{\text{Actual threading speed in rpm}}$

Goodheart-Willcox Publisher

Standard Dimensional Tolerances for Bar Stock

Hot-Rolled Bars: Rounds and Squares

Specified Size (Inches)	Variations from Size (Inches)		Out-of-Round (1) or -Square (2) (Inches)
	Over	Under	
1/4 to 5/16 inclusive (3)	(4)	(4)	(4)
Over 5/16 to 7/16 inclusive (3)	0.006	0.006	0.009
Over 7/16 to 5/8 inclusive (3)	0.007	0.007	0.010
Over 5/8 to 7/8 inclusive	0.008	0.008	0.012
Over 7/8 to 1 inclusive	0.009	0.009	0.013
Over 1 to 1 1/8 inclusive	0.010	0.010	0.015
Over 1 1/8 to 1 1/4 inclusive	0.011	0.011	0.016
Over 1 1/4 to 1 3/8 inclusive	0.012	0.012	0.018
Over 1 3/8 to 1 1/2 inclusive	0.014	0.014	0.021
Over 1 1/2 to 2 inclusive	1/64	1/64	0.023
Over 2 to 2 1/2 inclusive	1/32	0	0.023
Over 2 1/2 to 3 1/2 inclusive	3/64	0	0.035
Over 3 1/2 to 4 1/2 inclusive	1/16	0	0.046
Over 4 1/2 to 5 1/2 inclusive	5/64	0	0.058
Over 5 1/2 to 6 1/2 inclusive	1/8	0	0.070
Over 6 1/2 to 8 inclusive	5/32	0	0.085

(1) Out-of-round is difference between maximum and minimum diameters of bar, measured at same cross section.

(2) Out-of-square is difference in two dimensions at same cross section of a square bar, each dimension being distance between opposite faces.

(3) Round sections in size range of 1/4" to approximately 5/8" diameter are commonly produced on rod mills in coils. Tolerances on product made this way have not been established; for such tolerances, producer should be consulted. Variations in size of coiled product made on rod mills are greater than size tolerances for product made on bar mills.

(4) Squares in this size are not commonly produced as hot-rolled product.

Hot-Rolled Bars: Hexagons and Octagons

Specified Sizes Between Opposite Sides (Inches)	Variations from Size (Inches)		Maximum Difference 3 Measurements for Hexagons Only (Inches)
	Over	Under	
1/4 to 1/2 inclusive	0.007	0.007	0.011
Over 1/2 to 1 inclusive	0.010	0.010	0.015
Over 1 to 1 1/2 inclusive	0.021	0.021	0.025
Over 1 1/2 to 2 inclusive	1/32	1/32	1/32
Over 2 to 2 1/2 inclusive	3/64	3/64	3/64
Over 2 1/2 to 3 1/2 inclusive	1/16	1/16	1/16

(Continued)

Goodheart-Willcox Publisher

Standard Dimensional Tolerances for Bar Stock (Continued)

Cold-Finished Bars: Flats

Specified Width Size or Thickness (Inches)	Variations from Width Over or Under (Inches)*		Variations from Thicknesses Over or Under (Inch)
	For Thicknesses 1/4″ and Under	For Thicknesses Over 1/4″	
Over 1/8 to 1 inclusive	—	—	0.002
Over 3/8 to 1 inclusive	0.004	0.002	0.002
Over 1 to 2 inclusive	0.006	0.003	0.003
Over 2 to 3 inclusive	0.008	0.004	0.004
Over 3 to 4 1/2 inclusive	0.010	0.005	0.005

*When it is necessary to heat treat, or heat treat and pickle after cold finishing, because of special hardness or mechanical property requirements, tolerances are double those shown in the table.

Hot-Rolled Bars: Flats

Specified Widths (Inches)	Variations from Thickness for Thicknesses Given Over or Under (Inches)			Variations from Width (Inches)	
	1/8 to 1/2 Inclusive	Over 1/2 to 1 Inclusive	Over 1 to 2 Inclusive	Over	Under
To 1 inclusive	0.008	0.010	—	1/64	1/64
Over 1 to 2 inclusive	0.012	0.015	1/32	1/32	1/32
Over 2 to 4 inclusive	0.015	0.020	1/32	1/16	1/32
Over 4 to 6 inclusive	0.015	0.020	1/32	3/32	1/16
Over 6 to 8 inclusive	0.016	0.025	1/32	1/8	5/32
Over 8 to 10 inclusive	0.021	0.031	1/32	5/32	3/16

Cold-Finished Bars: Hexagons, Octagons, and Squares

Specified Size (Inches)	Variations from Size (Inches)*	
	Over	Under
Over 1/2 to 1 inclusive	0	0.004
Over 1 to 2 inclusive	0	0.006
Over 2 to 3 inclusive	0	0.008
Over 3	0	0.010

*When it is necessary to heat treat, or heat treat and pickle after cold finishing, because of special hardness or mechanical property requirements, tolerances are double those shown in the table.

Cold-Finished Bars: Rounds

Specified Size (Inches)	Variations from Size (Inches)*	
	Over	Under
Over 1/2 to 1 exclusive	0.002	0.002
1 to 1 1/2 exclusive	0.0025	0.0025
1 1/2 to 4 inclusive	0.003	0.003

*When it is necessary to heat treat, or heat treat and pickle after cold finishing, because of special hardness or mechanical property requirements, tolerances are double those shown in table.

(Continued)

Goodheart-Willcox Publisher

Standard Dimensional Tolerances for Bar Stock (Continued)				
Machine-Cut Bars (Cut after Machine Straightening)				
Specified Sizes as They Apply to Rounds, Squares, Hexagons, Octagons, and Width of Flats (Inches)	Variations from Specified Lengths (Inches)			
	To 12′ Inclusive		Over 12′ to 25′ Inclusive	
	Over	Under	Over	Under
To 3 inclusive	1/8	0	3/16	0
Over 3 to 6 inclusive	3/16	0	1/4	0
Over 6 to 9 inclusive	1/4	0	5/16	0
Over 9 to 12 inclusive	1/2	0	1/2	0

Allowances for Turning Machine-Straightened Bars

When ordering bars that are to be turned, it is recommended that allowances be made for finishing from hot-rolled diameters not less than amounts shown in following table, and specify hot-rolled sizes accordingly:

Nominal Diameter of Hot-Rolled Bar (Inches)	Minimum Allowance on Diameter for Turning (Inches)
1 1/2 to 3 inclusive	1/8
Over 3 to 6 inclusive	1/4
Over 6 to 8 inclusive	3/8

Hot or Cold Cutting Length Tolerances				
Specified Sizes as They Apply to Rounds, Squares, Hexagons, Octagons, and Width of Flats (Inches)	Variations from Specified Lengths (Inches)			
	To 12′ Inclusive		Over 12′ to 25′ Inclusive	
	Over	Under	Over	Under
To 2 inclusive	1/2	0	3/4	0
Over 2 to 4 inclusive	3/4	0	1	0
Over 4 to 6 inclusive	1	0	1 1/4	0
Over 6 to 9 inclusive	1 1/4	0	1 1/2	0
Over 9 to 12 inclusive	1 1/2	0	2	0

Hot-Finished and Cold-Finished Bars for Machining

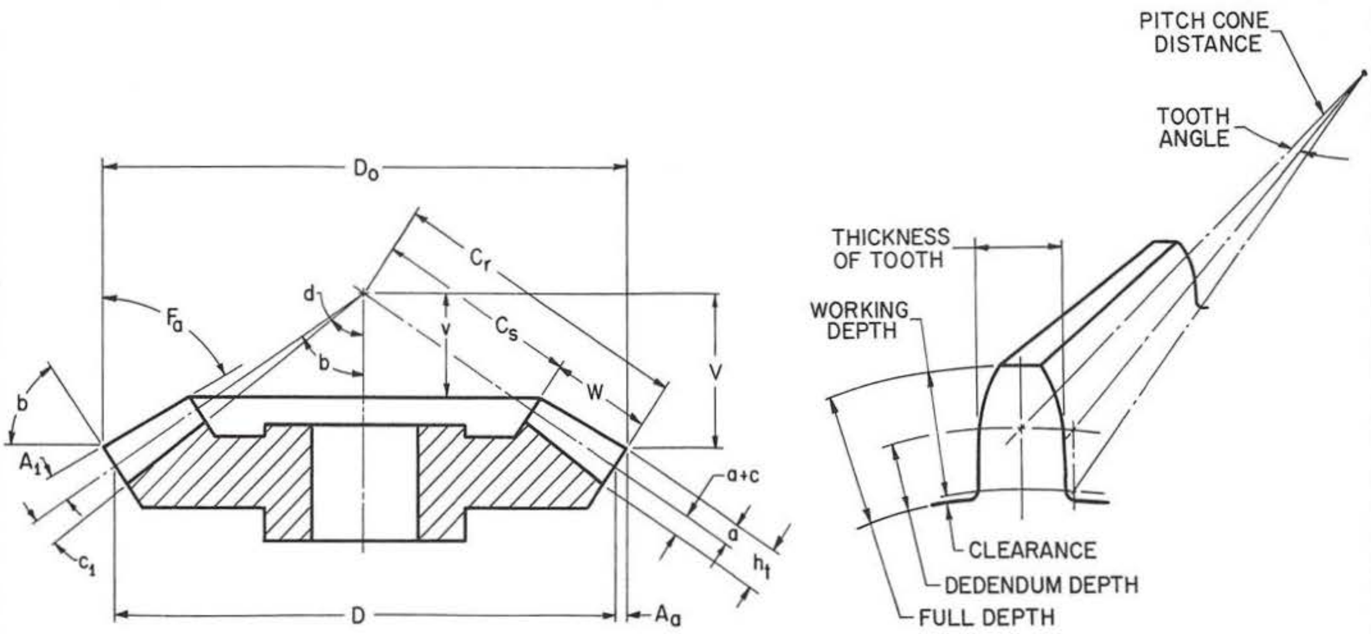
Camber is greatest deviation of a side from a straight line. Measurement is taken on concave side of bar with a straightedge. Unless otherwise specified, hot-finished and cold-finished bars for machining purposes are furnished machine-straightened to following tolerances:

Hot-finished: 1/8″ in any 5 feet; but may not exceed $1/8'' \times \frac{\text{No. of feet in length}}{5}$

Cold-finished: 1/16″ in any 5 feet; but may not exceed $1/16'' \times \frac{\text{No. of feet in length}}{5}$

Goodheart-Willcox Publisher

Rules and Formulas for Bevel Gear Calculations



To Find	Rule	Formula
Diametral pitch (*P*)	Divide the number of teeth by the pitch diameter.	$P = \frac{N}{D}$
Circular pitch (*p*)	Divide 3.1416 by the diametral pitch.	$p = \frac{3.1416}{P}$
Pitch diameter (*D*)	Divide the number of teeth by the diametral pitch.	$D = \frac{N}{P}$
Pitch angle of pinion tan (b_p)	Divide the number of teeth in the pinion by the number of teeth in the gear to obtain the tangent.	$\tan b_p = \frac{N_p}{N_g}$
Pitch angle of gear tan (b_g)	Divide the number of teeth in the gear by the number of teeth in the pinion to obtain the tangent.	$\tan b_g = \frac{N_g}{N_p}$
Pitch cone distance (C_r)	Divide the pitch diameter by twice the sine of the pitch angle.	$C_r = \frac{D}{2 (\sin b)}$
Addendum (*a*)	Divide 1.0 by the diametral pitch.	$a = \frac{1.0}{P}$
Addendum angle tan (A_1)	Divide the addendum by the pitch cone distance to obtain the tangent.	$\tan A_1 = \frac{a}{C_r}$
Angular addendum (A_a)	Multiply the addendum by the cosine of the pitch angle.	$A_a = a \cos b$
Outside diameter (D_o)	Add twice the angular addendum to the pitch diameter.	$D_o = D + 2A_a$
Dedendum angle tan (c_1)	Divide the dedendum by the pitch cone distance to obtain the tangent.	$\tan c_1 = \frac{a + c}{C_r}$
Addendum of small end of tooth (a_s)	Subtract the width of face from the pitch cone distance, divide the remainder by the pitch cone distance and multiply by the addendum.	$a_s = a \left(\frac{C_r - W}{C_r} \right)$
Thickness of tooth at pitch line (t_L)	Divide the circular pitch by 2.	$t_L = \frac{P_c}{2}$

(Continued)

Goodheart-Willcox Publisher

Rules and Formulas for Bevel Gear Calculations (Continued)		
To Find	**Rule**	**Formula**
Thickness of tooth at pitch line at small end of gear (t_s)	Subtract the width of face from the pitch cone distance, divide the remainder by the pitch cone distance and multiply by the thickness of the tooth at the pitch line.	$t_s = t_L \left(\frac{C_r - W}{C_r}\right)$
Face angle (F_a)	Face cone of blank turned parallel to root cone of mating gear.	$F_a = b + c_1$
Whole depth of tooth space (h_t)	Divide 2.157 by the diametral pitch.	$h_t = \frac{2.157}{P}$
Apex distance at large end of tooth (V)	Multiply one-half the outside diameter by the tangent of the face angle.	$V = \left(\frac{D_o}{2}\right) \tan F_a$
Apex distance at small end of tooth (v)	Subtract the width of face from the pitch cone distance, divide the remainder by the pitch cone distance and multiply by the apex distance.	$v = V\left(\frac{C_r - W}{C_r}\right)$
Gear ratio (m_g)	Divide the number of teeth in the gear by the number of teeth in the pinion.	$m_g = \frac{N_g}{N_p}$
Number of teeth in gear and/or pinion (N_g, N_p)	Multiply the pitch diameter by the diametral pitch.	$N_g = DP$ $N_p = DP$
Cutting angle (d)	Subtract the addendum plus clearance angle from the pitch angle.	$d = b - c_1$
Number of teeth of imaginary spur gear for which cutter is selected (N_o)	Divide the number of teeth in actual gear by the cosine of the pitch angle.	$N_o = \frac{N}{\cos b}$

Goodheart-Willcox Publisher

Decimal Equivalents: Number-Size Drills							
Drill	**Size of Drill in Inches**	**Drill**	**Size of Drill in Inches**	**Drill**	**Size of Drill in Inches**	**Drill**	**Size of Drill in Inches**
1	.2280	21	.1590	41	.0960	61	.0390
2	.2210	22	.1570	42	.0935	62	.0380
3	.2130	23	.1540	43	.0890	63	.0370
4	.2090	24	.1520	44	.0860	64	.0360
5	.2055	25	.1495	45	.0820	65	.0350
6	.2040	26	.1470	46	.0810	66	.0330
7	.2010	27	.1440	47	.0785	67	.0320
8	.1990	28	.1405	48	.0760	68	.0310
9	.1960	29	.1360	49	.0730	69	.0292
10	.1935	30	.1285	50	.0700	70	.0280
11	.1910	31	.1200	51	.0670	71	.0260
12	.1890	32	.1160	52	.0635	72	.0250
13	.1850	33	.1130	53	.0595	73	.0240
14	.1820	34	.1110	54	.0550	74	.0225
15	.1800	35	.1100	55	.0520	75	.0210
16	.1770	36	.1065	56	.0465	76	.0200
17	.1730	37	.1040	57	.0430	77	.0180
18	.1695	38	.1015	58	.0420	78	.0160
19	.1660	39	.0995	59	.0410	79	.0145
20	.1610	40	.0980	60	.0400	80	.0135

Goodheart-Willcox Publisher

Decimal Equivalents: Letter-Size Drills

Drill	Size of Drill in Inches	Drill	Size of Drill in Inches	Drill	Size of Drill in Inches	Drill	Size of Drill in Inches
A	0.234	H	0.266	O	0.316	V	0.377
B	0.238	I	0.272	P	0.323	W	0.386
C	0.242	J	0.277	Q	0.332	X	0.397
D	0.246	K	0.281	R	0.339	Y	0.404
E	0.250	L	0.290	S	0.348	Z	0.413
F	0.257	M	0.295	T	0.358		
G	0.261	N	0.302	U	0.368		

Goodheart-Willcox Publisher

60° V-Type Thread Dimensions: Fractional Sizes

National Special Thread Series

Nominal Size	Threads per Inch	Major Diameter (Inches)	Minor Diameter (Inches)	Pitch Diameter (Inches)	Tap Drill for 75% Thread †	Clearance Drill Size*
1/16″	64	.0625	.0422	.0524	3/64″	51
5/64″	60	.0781	.0563	.0673	1/16″	45
3/32″	48	.0938	.0667	.0803	49	40
7/64″	48	.1094	.0823	.0959	43	32
1/8″	32	.1250	.0844	.1047	3/32″	29
9/64″	40	.1406	.1081	.1244	32	24
5/32″	32	.1563	.1157	.1360	1/8″	19
5/32″	36	.1563	.1202	.1382	30	19
11/64″	32	.1719	.1313	.1516	9/64″	14
3/16″	24	.1875	.1334	.1604	26	8
3/16″	32	.1875	.1469	.1672	22	8
13/64″	24	.2031	.1490	.1760	20	3
7/32″	24	.2188	.1646	.1917	16	1
7/32″	32	.2188	.1782	.1985	12	1
15/64″	24	.2344	.1806	.2073	10	1/4″
1/4″	24	.2500	.1959	.2229	4	17/64″
1/4″	27	.2500	.2019	.2260	3	17/64″
1/4″	32	.2500	.2094	.2297	7/32″	17/64″

† *Refer to tables "Decimal Equivalents: Number-Size Drills" and "Decimal Equivalents: Letter-Size Drills."*

* *Clearance drill makes hole with standard clearance for diameter of nominal size.*

** *Standard spark plug size.*

(Continued)

Goodheart-Willcox Publisher

60° V-Type Thread Dimensions: Fractional Sizes (Continued)

National Special Thread Series

Nominal Size	Threads per Inch	Major Diameter (Inches)	Minor Diameter (Inches)	Pitch Diameter (Inches)	Tap Drill for 75% Thread †	Clearance Drill Size*
5/16″	20	.3125	.2476	.2800	17/64″	21/64″
5/16″	27	.3125	.2644	.2884	J	21/64″
5/16″	32	.3125	.2719	.2922	9/32″	21/64″
3/8″	20	.3750	.3100	.3425	21/64″	25/64″
3/8″	27	.3750	.3269	.3509	R	25/64″
7/16″	24	.4375	.3834	.4104	X	29/64″
7/16″	27	.4375	.3894	.4134	Y	29/64″
1/2″	12	.5000	.3918	.4459	27/64″	33/64″
1/2″	24	.5000	.4459	.4729	29/64″	33/64″
1/2″	27	.5000	.4519	.4759	15/32″	33/64″
9/16″	27	.5625	.5144	.5384	17/32″	37/64″
5/8″	12	.6250	.5168	.5709	35/64″	41/64″
5/8″	27	.6250	.5769	.6009	19/32″	41/64″
11/16″	11	.6875	.5694	.6285	19/32″	45/64″
11/16″	16	.6875	.6063	.6469	5/8″	45/64″
3/4″	12	.7500	.6418	.6959	43/64″	49/64″
3/4″	27	.7500	.7019	.7259	23/32″	49/64″
13/16″	10	.8125	.6826	.7476	23/32″	53/64″
7/8″	12	.8750	.7668	.8209	51/64″	57/64″
7/8″	18**	.8750	.8028	.8389	53/64″	57/64″
7/8″	27	.8750	.8269	.8509	27/32″	57/64″
15/16″	9	.9375	.7932	.8654	53/64″	61/64″
1″	12	1.0000	.8918	.9459	59/64″	1 1/64″
1″	27	1.0000	.9519	.9759	31/32″	1 1/64″
1 5/8″	5 1/2	1.6250	1.3888	1.5069	1 29/64″	1 41/64″
1 7/8″	5	1.8750	1.6152	1.7451	1 11/16″	1 57/64″
2 1/8″	4 1/2	2.1250	1.8363	1.9807	1 29/32″	2 5/32″
2 3/8″	4	2.3750	2.0502	2.2126	2 1/8″	2 13/32″

† Refer to tables "Decimal Equivalents: Number-Size Drills" and "Decimal Equivalents: Letter-Size Drills."

** Clearance drill makes hole with standard clearance for diameter of nominal size.*

*** Standard spark plug size.*

Goodheart-Willcox Publisher

60° V-Type Thread Dimensions: Metric Sizes

International Standard

Major Diameter (mm)	Pitch (mm)	Minor Diameter (mm)	Pitch Diameter (mm)	Tap Drill for 75% Thread (mm)	Tap Drill for 75% Thread † (No. or Inches)	Clearance Drill Size*
2.0	.40	1.48	1.740	1.6	1/16″	41
2.3	.40	1.78	2.040	1.9	48	36
2.6	.45	2.02	2.308	2.1	45	31
3.0	.50	2.35	2.675	2.5	40	29
3.5	.60	2.72	3.110	2.9	33	23
4.0	.70	3.09	3.545	3.3	30	16
4.5	.75	3.53	4.013	3.75	26	10
5.0	.80	3.96	4.480	4.2	19	3
5.5	.90	4.33	4.915	4.6	14	15/64″
6.0	1.00	4.70	5.350	5.0	9	1/4″
7.0	1.00	5.70	6.350	6.0	15/64″	19/64″
8.0	1.25	6.38	7.188	6.8	H	11/32″
9.0	1.25	7.38	8.188	7.8	5/16″	3/8″
10.0	1.50	8.05	9.026	8.6	R	27/64″
11.0	1.50	9.05	10.026	9.6	V	29/64″
12.0	1.75	9.73	10.863	10.5	Z	1/2″
14.0**	1.25	12.38	13.188	13.0	33/64″	9/16″
14.0	2.00	11.40	12.701	12.0	15/32″	9/16″
16.0	2.00	13.40	14.701	14.0	35/64″	21/32″
18.0	1.50	16.05	17.026	16.5	41/64″	47/64″
18.0	2.50	14.75	16.376	15.5	39/64″	47/64″
20.0	2.50	16.75	18.376	17.5	11/16″	13/16″
22.0	2.50	18.75	20.376	19.5	49/64″	57/64″
24.0	3.00	20.10	22.051	21.0	53/64″	31/32″
27.0	3.00	23.10	25.051	24.0	15/16″	13/32″
30.0	3.50	25.45	27.727	26.5	1 3/64″	1 13/64″
33.0	3.50	28.45	30.727	29.5	1 11/64″	1 21/64″
36.0	4.00	30.80	33.402	32.0	1 17/64″	1 7/16″
39.0	4.00	33.80	36.402	35.0	1 3/8″	1 9/16″
42.0	4.50	36.15	39.077	37.0	1 29/64″	1 43/64″
45.0	4.50	39.15	42.077	40.0	1 37/64″	1 13/16″
48.0	5.00	41.50	44.752	43.0	1 11/16″	1 29/64″

† Refer to tables "Decimal Equivalents: Number-Size Drills" and "Decimal Equivalents: Letter-Size Drills."

** Clearance drill makes hole with standard clearance for diameter of nominal size.*

*** Standard spark plug size.*

Goodheart-Willcox Publisher

Tap Drill Sizes

Probable Percentage of Full Thread Produced in Tapped Hole Using Stock Sizes of Drill

Tap	Tap Drill	Decimal Equivalent of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread	Tap	Tap Drill	Decimal Equivalent of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
0–80	56	.0465	83	.0015	.0480	74	8–32	29	.1360	69	.0029	.1389	62
	3/64	.0469	81	.0015	.0484	71		28	.1405	58	.0029	.1434	51
1–64	54	.0550	89	.0015	.0565	81	8–36	29	.1360	78	.0029	.1389	70
	53	.0595	67	.0015	.0610	59		28	.1405	68	.0029	.1434	57
1–72	53	.0595	75	.0015	.0610	67		9/64	.1406	68	.0029	.1435	57
	1/16	.0625	58	.0015	.0640	50	10–24	27	.1440	85	.0032	.1472	79
2–56	51	.0670	82	.0017	.0687	74		26	.1470	79	.0032	.1502	74
	50	.0700	69	.0017	.0717	62		25	.1495	75	.0032	.1527	69
	49	.0730	56	.0017	.0747	49		24	.1520	70	.0032	.1552	64
2–64	50	.0700	79	.0017	.0717	70		23	.1540	67	.0032	.1572	61
	49	.0730	64	.0017	.0747	56		5/32	.1563	62	.0032	.1595	56
3–48	48	.0760	85	.0019	.0779	78		22	.1570	61	.0032	.1602	55
	5/64	.0781	77	.0019	.0800	70	10–32	5/32	.1563	83	.0032	.1595	75
	47	.0785	76	.0019	.0804	69		22	.1570	81	.0032	.1602	73
	46	.0810	67	.0019	.0829	60		21	.1590	76	.0032	.1622	68
	45	.0820	63	.0019	.0839	56		20	.1610	71	.0032	.1642	64
3–56	46	.0810	78	.0019	.0829	69		19	.1660	59	.0032	.1692	51
	45	.0820	73	.0019	.0839	65	12–24	11/64	.1719	82	.0035	.1754	75
	44	.0860	56	.0019	.0879	48		17	.1730	79	.0035	.1765	73
4–40	44	.0860	80	.0020	.0880	74		16	.1770	72	.0035	.1805	66
	43	.0890	71	.0020	.0910	65		15	.1800	67	.0035	.1835	60
	42	.0935	57	.0020	.0955	51		14	.1820	63	.0035	.1855	56
	3/32	.0938	56	.0020	.0958	50	12–28	16	.1770	84	.0035	.1805	77
4–48	42	.0935	68	.0020	.0955	61		15	.1800	78	.0035	.1835	70
	3/32	.0938	68	.0020	.0958	60		14	.1820	73	.0035	.1855	66
	41	.0960	59	.0020	.0980	52		13	.1850	67	.0035	.1885	59
5–40	40	.0980	83	.0023	.1003	76		3/16	.1875	61	.0035	.1910	54
	39	.0995	79	.0023	.1018	71	1/4–20	9	.1960	83	.0038	.1998	77
	38	.1015	72	.0023	.1038	65		8	.1990	79	.0038	.2028	73
	37	.1040	65	.0023	.1063	58		7	.2010	75	.0038	.2048	70
5–44	38	.1015	79	.0023	.1038	72		13/64	.2031	72	.0038	.2069	66
	37	.1040	71	.0023	.1063	63		6	.2040	71	.0038	.2078	65
	36	.1065	63	.0023	.1088	55		5	.2055	69	.0038	.2093	63
6–32	37	.1040	84	.0023	.1063	78		4	.2090	63	.0038	.2128	57
	36	.1065	78	.0026	.1091	71	1/4–28	3	.2130	80	.0038	.2168	72
	7/64	.1094	70	.0026	.1120	64		7/32	.2188	67	.0038	.2226	59
	35	.1100	69	.0026	.1126	63		2	.2210	63	.0038	.2248	55
	34	.1110	67	.0026	.1136	60	5/16–18	F	.2570	77	.0038	.2608	72
	33	.1130	62	.0026	.1156	55		G	.2610	71	.0041	.2651	66
6–40	34	.1110	83	.0026	.1136	75		17/64	.2656	65	.0041	.2697	59
	33	.1130	77	.0026	.1156	69		H	.2660	64	.0041	.2701	59
	32	.1160	68	.0026	.1186	60							

(Continued)

Standard Tool Co.

Tap Drill Sizes (Continued)

Probable Percentage of Full Thread Produced in Tapped Hole Using Stock Sizes of Drill

Tap	Tap Drill	Decimal Equivalent of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
5/16–24	H	.2660	86	.0041	.2701	78
	I	.2720	75	.0041	.2761	67
	J	.2770	66	.0041	.2811	58
3/8–16	5/16	.3125	77	.0044	.3169	72
	O	.3160	73	.0044	.3204	68
	P	.3230	64	.0044	.3274	59
3/8–24	21/64	.3281	87	.0044	.3325	79
	Q	.3320	79	.0044	.3364	71
	R	.3390	67	.0044	.3434	58
7/16–14	T	.3580	86	.0046	.3626	81
	23/64	.3594	84	.0046	.3640	79
	U	.3680	75	.0046	.3726	70
	3/8	.3750	67	.0046	.3796	62
	V	.3770	65	.0046	.3816	60
7/16–20	W	.3860	79	.0046	.3906	72
	25/64	.3906	72	.0046	.3952	65
	X	.3970	62	.0046	.4016	55
1/2–13	27/64	.4219	78	.0047	.4266	73
	7/16	.4375	63	.0047	.4422	58
1/2–20	29/64	.4531	72	.0047	.4578	65
9/16–12	15/32	.4688	87	.0048	.4736	82
	31/64	.4844	72	.0048	.4892	68
9/16–18	1/2	.5000	87	.0048	.5048	80
	33/64	.5156	65	.0048	.5204	58
5/8–11	17/32	.5313	79	.0049	.5362	75
	35/64	.5469	66	.0049	.5518	62
5/8–18	9/16	.5625	87	.0049	.5674	80
	37/64	.5781	65	.0049	.5831	58
3/4–10	41/64	.6406	84	.0050	.6456	80
	21/32	.6563	72	.0050	.6613	68
3/4–16	11/16	.6875	77	.0050	.6925	71
7/8–9	49/64	.7656	76	.0052	.7708	72
	25/32	.7812	65	.0052	.7864	61
7/8–14	51/64	.7969	84	.0052	.8021	79
	13/16	.8125	67	.0052	.8177	62
1″–8	55/64	.8594	87	.0059	.8653	83
	7/8	.8750	77	.0059	.8809	73
	57/64	.8906	67	.0059	.8965	64
	29/32	.9063	58	.0059	.9122	54
1″–12	29/32	.9063	87	.0060	.9123	81
	59/64	.9219	72	.0060	.9279	67
	15/16	.9375	58	.0060	.9435	52

Tap	Tap Drill	Decimal Equivalent of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
1″–14	59/64	.9219	84	.0060	.9279	78
	15/16	.9375	67	.0060	.9435	61
1 1/8–7	31/32	.9688	84	.0062	.9750	81
	63/64	.9844	76	.0067	.9911	72
	1″	1.0000	67	.0070	1.0070	64
	1 1/64	1.0156	59	.0070	1.0226	55
1 1/8–12	1 1/32	1.0313	87	.0071	1.0384	80
	1 3/64	1.0469	72	.0072	1.0541	66
1 1/4–7	1 3/32	1.0938	84			
	1 7/64	1.1094	76			
	1 1/8	1.1250	67			
1 1/4–12	1 5/32	1.1563	87			
	1 11/64	1.1719	72			
1 3/8–6	1 3/16	1.1875	87			
	1 13/64	1.2031	79			
	1 7/32	1.2188	72			
	1 15/64	1.2344	65			
1 3/8–12	1 9/32	1.2813	87			
	1 19/64	1.2969	72			
1 1/2–6	1 5/16	1.3125	87			
	1 21/64	1.3281	79			
	1 11/32	1.3438	72			
	1 23/64	1.3594	65			
1 1/2–12	1 13/32	1.4063	87			
	1 27/64	1.4219	72			

No test results available

Reaming recommended

Taper Pipe	
Thread	**Drill**
1/8–27	R
1/4–18	7/16
3/8–18	37/64
1/2–14	23/32
3/4–14	59/64
1–11 1/2	1 5/32
1 1/4–11 1/2	1 1/2
1 1/2–11 1/2	1 47/64
2–11 1/2	2 7/32
2 1/2–8	2 5/8
3–8	3 1/4
3 1/2–8	3 3/4
4–8	4 1/4

Straight Pipe	
Thread	**Drill**
1/8–27	S
1/4–18	29/64
3/8–18	19/32
1/2–14	47/64
3/4–14	15/16
1–11 1/2	1 3/16
1 1/4–11 1/2	1 33/64
1 1/2–11 1/2	1 3/4
2–11 1/2	2 7/32
2 1/2–8	2 21/32
3–8	3 9/32
3 1/2–8	3 25/32
4–8	4 9/32

Standard Tool Co.

Taper Pin and Reamer Sizes

Length of Pin	Reamer															
	6/0	5/0	4/0	3/0	2/0	0	1	2	3	4	5	6	7	8	9	10
3/8	50	44	38	32	29											
1/2	51	45	39	33	30	27										
5/8	52	46	41	34	30	27	21									
3/4	1/16	47	42	7/64	1/8	9/64	5/32	16	13/64	15/64	I	P				
1		49	44	37	31	29	25	11/64	9	1	H	O	W			
1 1/4					32	30	26	19	10	2	G	5/16	V	29/64		
1 1/2						*1/8	*9/64	20	3/16	7/32	F	N	V	29/64	35/64	43/64
1 3/4						*31	*29	*5/32	14	3	E	N	U	29/64	35/64	21/32
2						*33	*30	*25	*16	4	D	M	U	7/16	35/64	21/32
2 1/4						*7/64	*1/8	*26	*11/64	*13/64	C	M	23/64	7/16	17/32	21/32
2 1/2						*37	*31	*28	*19	*8	*B	L	T	7/16	17/32	41/64
2 3/4						*40	*33	*29	*21	*11	*1	9/32	S	27/64	17/32	41/64
3						*42	*35	*30	*5/32	*3/16	*2	J	11/32	27/64	33/64	41/64
3 1/4									*24	*14	*2	*I	R	27/64	33/64	5/8
3 1/2									*26	*16	*3	*H	Q	Z	33/64	5/8
3 3/4										*17	*4	*G	*Q	Z	1/2	5/8
4										*19	*5	*F	*P	13/32	1/2	39/64
4 1/4												*1/4	*O	Y	1/2	39/64
4 1/2												*D	*O	X	31/32	39/64
4 3/4												*C	*5/16	*25/64	31/64	19/32
5												*B	*N	*W	31/64	19/32
5 1/4															15/32	19/32
5 1/2															*15/32	37/64
5 3/4															*15/32	37/64
6															*29/64	37/64

** Hole sizes too small to admit taper pin reamers of standard length. Special, extra-length reamers are required for these cases.*

Goodheart-Willcox Publisher

Hardness Conversions

Brinell Indentation Diameter (mm)	Brinell Hardness Number		Rockwell Hardness Number		Rockwell Superficial Hardness Number (Superficial Diamond Penetrator)			Tensile Strength (Approximate) × 1000 psi
	Standard Ball	Tungsten-Carbide Ball	B Scale	C Scale	15 N Scale	30 N Scale	45 N Scale	
2.45	—	627	—	58.7	89.6	76.3	65.1	347
2.50	—	601	—	57.3	89.0	75.1	63.5	328
2.55	—	578	—	56.0	88.4	73.9	62.1	313
2.60	—	555	—	54.7	87.8	72.7	60.6	298
2.65	—	534	—	53.5	87.2	71.6	59.2	288
2.70	—	514	—	52.1	86.5	70.3	57.6	274
2.75	—	495	—	51.0	85.9	69.4	56.1	264
2.80	—	477	—	49.6	85.3	68.2	54.5	252
2.85	—	461	—	48.5	84.7	67.2	53.2	242
2.90	—	444	—	47.1	84.0	65.8	51.5	230
2.95	429	429	—	45.7	83.4	64.6	49.9	219
3.00	415	415	—	44.5	82.8	63.5	48.4	212
3.05	401	401	—	43.1	82.0	62.3	46.9	202
3.10	388	388	—	41.8	81.4	61.1	45.3	193
3.15	375	375	—	40.4	80.6	59.9	43.6	184
3.20	363	363	—	39.1	80.0	58.7	42.0	177
3.25	352	352	—	37.9	79.3	57.6	40.5	170
3.30	341	341	—	36.6	78.6	56.4	39.1	163
3.35	331	331	—	35.5	78.0	55.4	37.8	158
3.40	321	321	—	34.3	77.3	54.3	36.4	152
3.45	311	311	—	33.1	76.7	53.3	34.4	147
3.50	302	302	—	32.1	76.1	52.2	33.8	143
3.55	293	293	—	30.9	75.5	51.2	32.4	139
3.60	285	285	—	29.9	75.0	50.3	31.2	136
3.65	277	277	—	28.8	74.4	49.3	29.9	131
3.70	269	269	—	27.6	73.7	48.3	28.5	128
3.75	262	262	—	26.6	73.1	47.3	27.3	125
3.80	255	255	—	25.4	72.5	46.2	26.0	121
3.85	248	248	—	24.2	71.7	45.1	24.5	118
3.90	241	241	100.0	22.8	70.9	43.9	22.8	114
3.95	235	235	99.0	21.7	70.3	42.9	21.5	111
4.00	229	229	98.2	20.5	69.7	41.9	20.1	109
4.05	223	223	97.3	—	—	—	—	104
4.10	217	217	96.4	—	—	—	—	103
4.15	212	212	95.5	—	—	—	—	100
4.20	207	207	94.6	—	—	—	—	99
4.25	201	201	93.8	—	—	—	—	97
4.30	197	197	92.8	—	—	—	—	94
4.35	192	192	91.9	—	—	—	—	92
4.40	187	187	90.7	—	—	—	—	90
4.45	183	183	90.0	—	—	—	—	89
4.50	179	179	89.0	—	—	—	—	88
4.55	174	174	87.8	—	—	—	—	86
4.60	170	170	86.8	—	—	—	—	84
4.65	167	167	86.0	—	—	—	—	83
4.70	163	163	85.0	—	—	—	—	82
4.80	156	156	82.9	—	—	—	—	80
4.90	149	149	80.8	—	—	—	—	73
5.00	143	143	78.7	—	—	—	—	71
5.10	137	137	76.4	—	—	—	—	67
5.20	131	131	74.0	—	—	—	—	65
5.30	126	131	72.0	—	—	—	—	63
5.40	121	121	69.0	—	—	—	—	60
5.50	116	116	67.6	—	—	—	—	58
5.60	111	111	65.7	—	—	—	—	56

Carpenter Steel Co.

Conversion Table: US Customary to SI Metric

When You Know:	Multiply By: Very Accurate	Multiply By: Approximate	To Find:
Length			
inches	* 25.4		millimeters
inches	* 2.54		centimeters
feet	* 0.3048		meters
feet	* 30.48		centimeters
yards	* 0.9144	0.9	meters
miles	* 1.609344	1.6	kilometers
Weight			
grains	15.43236	15.4	grams
ounces	* 28.349523125	28.0	grams
ounces	* 0.028349523125	0.028	kilograms
pounds	* 0.45359237	0.45	kilograms
short ton	* 0.90718474	0.9	tonnes
Volume			
teaspoons		5.0	milliliters
tablespoons		15.0	milliliters
fluid ounces	29.57353	30.0	milliliters
cups		0.24	liters
pints	* 0.473176473	0.47	liters
quarts	* 0.946352946	0.95	liters
gallons	* 3.785411784	3.8	liters
cubic inches	* 0.016387064	0.02	liters
cubic feet	* 0.028316846592	0.03	cubic meters
cubic yards	* 0.764554857984	0.76	cubic meters
Area			
square inches	* 6.4516	6.5	square centimeters
square feet	* 0.09290304	0.09	square meters
square yards	* 0.83612736	0.8	square meters
square miles		2.6	square kilometers
acres	* 0.40468564224	0.4	hectares
Temperature			
Fahrenheit	*5/9 (after subtracting 32)		Celsius
Density			
pounds per cubic feet	1.602×10	16	kilograms per cubic meter
Force			
ounces (F)	2.780×10^{-1}		newtons
pounds (F)	4.448×10^{-3}		kilonewtons
kips	4.448		meganewtons
Stress			
pounds/square inch (psi)	6.895×10^{-3}		megapascals
kips/square inch (ksi)	6.895		megapascals
Torque			
ounce-inches	7.062×10^{-3}		newton-meters
pound-inches	1.130×10^{-1}		newton-meters
pound-feet	1.356		newton-meters

* = Exact

Goodheart-Willcox Publisher

Conversion Table: SI Metric to US Customary

When You Know:	Multiply By:		To Find:
	Very Accurate	Approximate	
Length			
millimeters	0.0393701	0.04	inches
centimeters	0.3937008	0.4	inches
meters	3.280840	3.3	feet
meters	1.093613	1.1	yards
kilometers	0.621371	0.6	miles
Weight			
grains	0.00228571	0.0023	ounces
grams	0.03527396	0.035	ounces
kilograms	2.204623	2.2	pounds
tonnes	1.1023113	1.1	short tons
Volume			
milliliters		0.2	teaspoons
milliliters	0.06667	0.067	tablespoons
milliliters	0.03381402	0.03	fluid ounces
liters	61.02374	61.024	cubic inches
liters	2.113376	2.1	pints
liters	1.056688	1.06	quarts
liters	0.26417205	0.26	gallons
liters	0.03531467	0.035	cubic feet
cubic meters	61023.74	61023.7	cubic inches
cubic meters	35.31467	35.0	cubic feet
cubic meters	1.3079506	1.3	cubic yards
cubic meters	264.17205	264.0	gallons
Area			
square centimeters	0.1550003	0.16	square inches
square centimeters	0.00107639	0.001	square feet
square meters	10.76391	10.8	square feet
square meters	1.195990	1.2	square yards
square kilometers		0.4	square miles
hectares	2.471054	2.5	acres
Temperature			
Celsius	*9/5 (then add 32)		Fahrenheit

* = Exact

Goodheart-Willcox Publisher

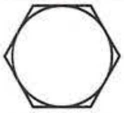
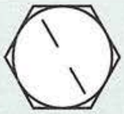
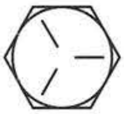
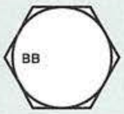
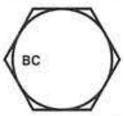
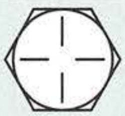
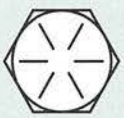
Decimal Conversion Chart

Fraction	Inches	mm
1/64	.01563	.397
1/32	.03125	.794
3/64	.04688	1.191
1/16	.0625	1.588
5/64	.07813	1.984
3/32	.09375	2.381
7/64	.10938	2.778
1/8	.12500	3.175
9/64	.14063	3.572
5/32	.15625	3.969
11/64	.17188	4.366
3/16	.18750	4.763
13/64	.20313	5.159
7/32	.21875	5.556
15/64	.23438	5.953
1/4	.25000	6.350
17/64	.26563	6.747
9/32	.28125	7.144
19/64	.29688	7.541
5/16	.31250	7.938
21/64	.32813	8.334
11/32	.34375	8.731
23/64	.35938	9.128
3/8	.37500	9.525
25/64	.39063	9.922
13/32	.40625	10.319
27/64	.42188	10.716
7/16	.43750	11.113
29/64	.45313	11.509
15/32	.46875	11.906
31/64	.48438	12.303
1/2	.50000	12.700

Fraction	Inches	mm
33/64	.51563	13.097
17/32	.53125	13.494
35/64	.54688	13.891
9/16	.56250	14.288
37/64	.57813	14.684
19/32	.59375	15.081
39/64	.60938	15.478
5/8	.62500	15.875
41/64	.64063	16.272
21/32	.65625	16.669
43/64	.67188	17.066
11/16	.68750	17.463
45/64	.70313	17.859
23/32	.71875	18.256
47/64	.73438	18.653
3/4	.75000	19.050
49/64	.76563	19.447
25/32	.78125	19.844
51/64	.79688	20.241
13/16	.81250	20.638
53/64	.82813	21.034
27/32	.84375	21.431
55/64	.85938	21.828
7/8	.87500	22.225
57/64	.89063	22.622
29/32	.90625	23.019
59/64	.92188	23.416
15/16	.93750	23.813
61/64	.95313	24.209
31/32	.96875	24.606
63/64	.98438	25.003
1	1.00000	25.400

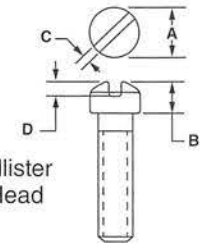
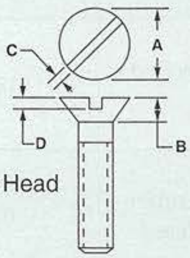
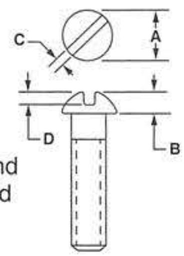
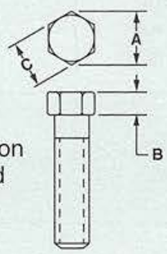
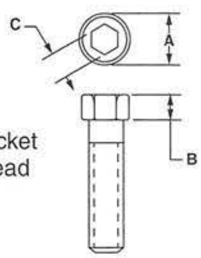
Goodheart-Willcox Publisher

Grade Marking for Bolts

Bolt Head Marking		SAE = Society of Automotive Engineers ASTM = American Society for Testing and Materials	Bolt Material	Minimum Tensile Strength in Pounds per Square Inch (psi)
No Marks		SAE Grade 1 SAE Grade 2 Indeterminate quality	Low-carbon steel Low-carbon steel	65,000 psi
2 Marks		SAE Grade 3	Medium-carbon steel, cold worked	110,000 psi
3 Marks		SAE Grade 5 ASTM – A 325 Common commercial quality	Medium-carbon steel, quenched and tempered	120,000 psi
Letters BB		ASTM – A 354	Low-alloy steel or medium-carbon steel, quenched and tempered	105,000 psi
Letters BC		ASTM – A 354	Low-alloy steel or medium-carbon steel, quenched and tempered	125,000 psi
4 Marks		SAE Grade 6 Better commercial quality	Medium-carbon steel, quenched and tempered	140,000 psi
5 Marks		SAE Grade 7	Medium-carbon alloy steel, quenched and tempered, roll-threaded after heat treatment	133,000 psi
6 Marks		SAE Grade 8 ASTM – A 345 Best commercial quality	Medium-carbon alloy steel, quenched and tempered	150,000 psi

Goodheart-Willcox Publisher

Machine Screw and Cap Screw Heads

	Size	A	B	C	D
 Fillister Head	#8	.260	.141	.042	.060
	#10	.302	.164	.048	.072
	1/4	3/8	.205	.064	.087
	5/16	7/16	.242	.077	.102
	3/8	9/16	.300	.086	.125
	1/2	3/4	.394	.102	.168
	5/8	7/8	.500	.128	.215
	3/4	1	.590	.144	.258
	1	1 5/16	.774	.182	.352
 Flat Head	#8	.320	.092	.043	.037
	#10	.372	.107	.048	.044
	1/4	1/2	.146	.064	.063
	5/16	5/8	.183	.072	.078
	3/8	3/4	.220	.081	.095
	1/2	7/8	.220	.102	.090
	5/8	1 1/8	.293	.128	.125
	3/4	1 3/8	.366	.144	.153
 Round Head	#8	.297	.113	.044	.067
	#10	.346	.130	.048	.073
	1/4	7/16	.183	.064	.107
	5/16	9/16	.236	.072	.150
	3/8	5/8	.262	.081	.160
	1/2	13/16	.340	.102	.200
	5/8	1	.422	.128	.255
	3/4	1 1/4	.526	.144	.320
 Hexagon Head	1/4	.494	.170	7/16	
	5/16	.564	.215	1/2	
	3/8	.635	.246	9/16	
	1/2	.846	.333	3/4	
	5/8	1.058	.411	15/16	
	3/4	1.270	.490	1 1/8	
	7/8	1.482	.566	1 5/16	
	1	1.693	.640	1 1/2	
 Socket Head	#8	.265	.164	1/8	
	#10	5/16	.190	5/32	
	1/4	3/8	1/4	3/16	
	5/16	7/16	5/16	7/32	
	3/8	9/16	3/8	5/16	
	7/16	5/8	7/16	5/16	
	1/2	3/4	1/2	3/8	
	5/8	7/8	5/8	1/2	
	3/4	1	3/4	9/16	
	7/8	1 1/8	7/8	9/16	
	1	1 5/16	1	5/8	

Goodheart-Willcox Publisher

Screw Thread Elements for Unified and National Thread Form

Threads per Inch (n)	Pitch (p) $p = \frac{1}{n}$	Single Height: Subtract from basic major diameter to get basic pitch diameter	Double Height: Subtract from basic major diameter to get basic minor diameter	83 1/3% Double Height: Subtract from basic major diameter to get minor diameter of ring gage	Basic Width of Crest and Root Flat $\frac{P}{8}$	Constant for Best Size Wire Also Single Height of 60° V-Thread	Diameter of Best Size Wire
3	.333333	.216506	.43301	.36084	.0417	.28868	.19245
3 1/4	.307692	.199852	.39970	.33309	.0385	.26647	.17765
3 1/2	.285714	.185577	.37115	.30929	.0357	.24744	.16496
4	.250000	.162379	.32476	.27063	.0312	.21651	.14434
4 1/2	.222222	.144337	.28867	.24056	.0278	.19245	.12830
5	.200000	.129903	.25981	.21650	.0250	.17321	.11547
5 1/2	.181818	.118093	.23619	.19682	.0227	.15746	.10497
6	.166666	.108253	.21651	.18042	.0208	.14434	.09623
7	.142857	.092788	.18558	.15465	.0179	.12372	.08248
8	.125000	.081189	.16238	.13531	.0156	.10825	.07217
9	.111111	.072168	.14434	.12028	.0139	.09623	.06415
10	.100000	.064952	.12990	.10825	.0125	.08660	.05774
11	.090909	.059046	.11809	.09841	.0114	.07873	.05249
11 1/2	.086956	.056480	.11296	.09413	.0109	.07531	.05020
12	.083333	.054127	.10826	.09021	.0104	.07217	.04811
13	.076923	.049963	.09993	.08327	.0096	.06662	.04441
14	.071428	.046394	.09279	.07732	.0089	.06186	.04124
16	.062500	.040595	.08119	.06766	.0078	.05413	.03608
18	.055555	.036086	.07217	.06014	.0069	.04811	.03208
20	.050000	.032475	.06495	.05412	.0062	.04330	.02887
22	.045454	.029523	.05905	.04920	.0057	.03936	.02624
24	.041666	.027063	.05413	.04510	.0052	.03608	.02406
27	.037037	.024056	.04811	.04009	.0046	.03208	.02138
28	.035714	.023197	.04639	.03866	.0045	.03093	.02062
30	.033333	.021651	.04330	.03608	.0042	.02887	.01925
32	.031250	.020297	.04059	.03383	.0039	.02706	.01804
36	.027777	.018042	.03608	.03007	.0035	.02406	.01604
40	.025000	.016237	.03247	.02706	.0031	.02165	.01443
44	.022727	.014761	.02952	.02460	.0028	.01968	.01312
48	.020833	.013531	.02706	.02255	.0026	.01804	.01203
50	.020000	.012990	.02598	.02165	.0025	.01732	.01155
56	.017857	.011598	.02320	.01933	.0022	.01546	.01031
60	.016666	.010825	.02165	.01804	.0021	.01443	.00962
64	.015625	.010148	.02030	.01691	.0020	.01353	.00902
72	.013888	.009021	.01804	.01503	.0017	.01203	.00802
80	.012500	.008118	.01624	.01353	.0016	.01083	.00722
90	.011111	.007217	.01443	.01202	.0014	.00962	.00642
96	.010417	.006766	.01353	.01127	.0013	.00902	.00601
100	.010000	.006495	.01299	.01082	.0012	.00866	.00577
120	.008333	.005413	.01083	.00902	.0010	.00722	.00481

Note: Using the Best Size Wires, measurement over three wires minus Constant for Best Size Wire equals Pitch Diameter.

Goodheart-Willcox Publisher

Bolt Torquing Chart

Metric Standard							SAE Standard/Foot Pounds						
Grade of Bolt		5D	.8G	10K	12K		Grade of Bolt	SAE 1 & 2	SAE 5	SAE 6	SAE 8		
Min. Ten. Strength		71,160 psi	113,800 psi	142,200 psi	170,679 psi		Min. Ten. Strength	64,000 psi	105,000 psi	133,000 psi	150,000 psi		
Markings on Head		5D	.8G	10K	12K	Size of Socket or Wrench Opening	Markings on Head					Size of Socket or Wrench Opening	
Metric		Foot Pounds				Metric	U.S. Standard	Foot Pounds				U.S. Standard	
Bolt Dia.	U.S. Dec. Equiv.					Bolt Head	Bolt Dia.					Bolt Head	Nut
6 mm	.2362	5	6	8	10	10 mm	1/4	5	7	10	10.5	3/8	7/16
8 mm	.3150	10	16	22	27	14 mm	5/16	9	14	19	22	1/2	9/16
10 mm	.3937	19	31	40	49	17 mm	3/8	15	25	34	37	9/16	5/8
12 mm	.4720	34	54	70	86	19 mm	7/16	24	40	55	60	5/8	3/4
14 mm	.5512	55	89	117	137	22 mm	1/2	37	60	85	92	3/4	13/16
16 mm	.6299	83	132	175	208	24 mm	9/16	53	88	120	132	7/8	7/8
18 mm	.7090	111	182	236	283	27 mm	5/8	74	120	167	180	15/16	1
22 mm	.8661	182	284	394	464	32 mm	3/4	120	200	280	296	1 1/8	1 1/8

Goodheart-Willcox Publisher

Cutting Fluids for Various Metals

Aluminum and its Alloys	Kerosene, kerosene and lard oil, soluble oil	**Monel Metal**	Lard oil, soluble oil
Plastics	Dry	**Slate**	Dry
Brass, Soft	Dry, soluble oil, kerosene and lard oil	**Steel, Forging**	Soluble oil, sulfurized oil, mineral lard oil
Bronze, High Tensile	Soluble oil, lard oil, mineral oil, dry	**Steel, Manganese**	Soluble oil, sulfurized oil, mineral lard oil
Cast Iron	Dry, air jet, soluble oil	**Steel, Soft**	Soluble oil, mineral lard oil, sulfurized oil, lard oil
Copper	Soluble oil, dry, mineral lard oil, kerosene	**Steel, Stainless**	Sulfurized mineral oil, soluble oil
Magnesium	Low viscosity neutral oils	**Steel, Tool**	Soluble oil, mineral lard oil, sulfurized oil
Malleable Iron	Dry, soda water	**Wrought Iron**	Soluble oil, mineral lard oil, sulfurized oil

Goodheart-Willcox Publisher

EIA and AIA National Codes for CNC Programming

Preparatory (G) Functions	
G word	**Explanation**
G00	Denotes a rapid traverse rate for point-to-point positioning.
G01	Describes linear interpolation blocks; reserved for contouring.
G02, G03	Used with circular interpolation.
G04	Sets a calculated time delay during which there is no machine motion (dwell).
G05, G07	Unassigned by the EIA. May be used at the discretion of the machine tool or system builder. Could also be standardized at a future date.
G06	Used with parabolic interpolation.
G08	Acceleration code. Causes the machine, assuming it is capable, to accelerate at a smooth exponential rate.
G09	Deceleration code. Causes the machine, assuming it is capable, to decelerate at a smooth exponential rate.
G10-G12	Normally unassigned for CNC systems. Used with some hard-wired systems to express blocks of abnormal dimensions.
G13-G16	Direct the control system to operate on a particular set of axes.
G17-G19	Identify or select a coordinate plane for such functions as circular interpolation or cutter compensation.
G20-G32	Unassigned according to EIA standards. May be assigned by the control system or machine tool builder.
G33-G35	Selected for machines equipped with thread-cutting capabilities (generally referring to lathes). G33 is used when a constant lead is sought, G34 is used when a constantly increasing lead is required, and G35 is used to designate a constantly decreasing lead.
G36-G39	Unassigned.
G40	Terminates any cutter compensation.
G41	Activates cutter compensation in which the cutter is on the left side of the work surface (relative to the direction of the cutter motion).
G42	Activates cutter compensation in which the cutter is on the right side of the work surface.
G43, G44	Used with cutter offset to adjust for the difference between the actual and programmed cutter radii or diameters.
G43	refers to an inside corner, and G44 refers to an outside corner.
G45-G49	Unassigned.
G50-G59	Reserved for adaptive control.
G60-G69	Unassigned.
G70	Selects inch programming.
G71	Selects metric programming.
G72	Selects three-dimensional CW circular interpolation.
G73	Selects three-dimensional CCW circular interpolation.
G74	Cancels multiquadrant circular interpolation.
G75	Activates multiquadrant circular interpolation.
G76-G79	Unassigned.
G80	Cancel cycle.
G81	Activates drill, or spotdrill, cycle.
G82	Activates drill with a dwell.
G83	Activates intermittent, or deep-hole, drilling.
G84	Activates tapping cycle.
G85-G89	Activates boring cycles.

(Continued)

Goodheart-Willcox Publisher

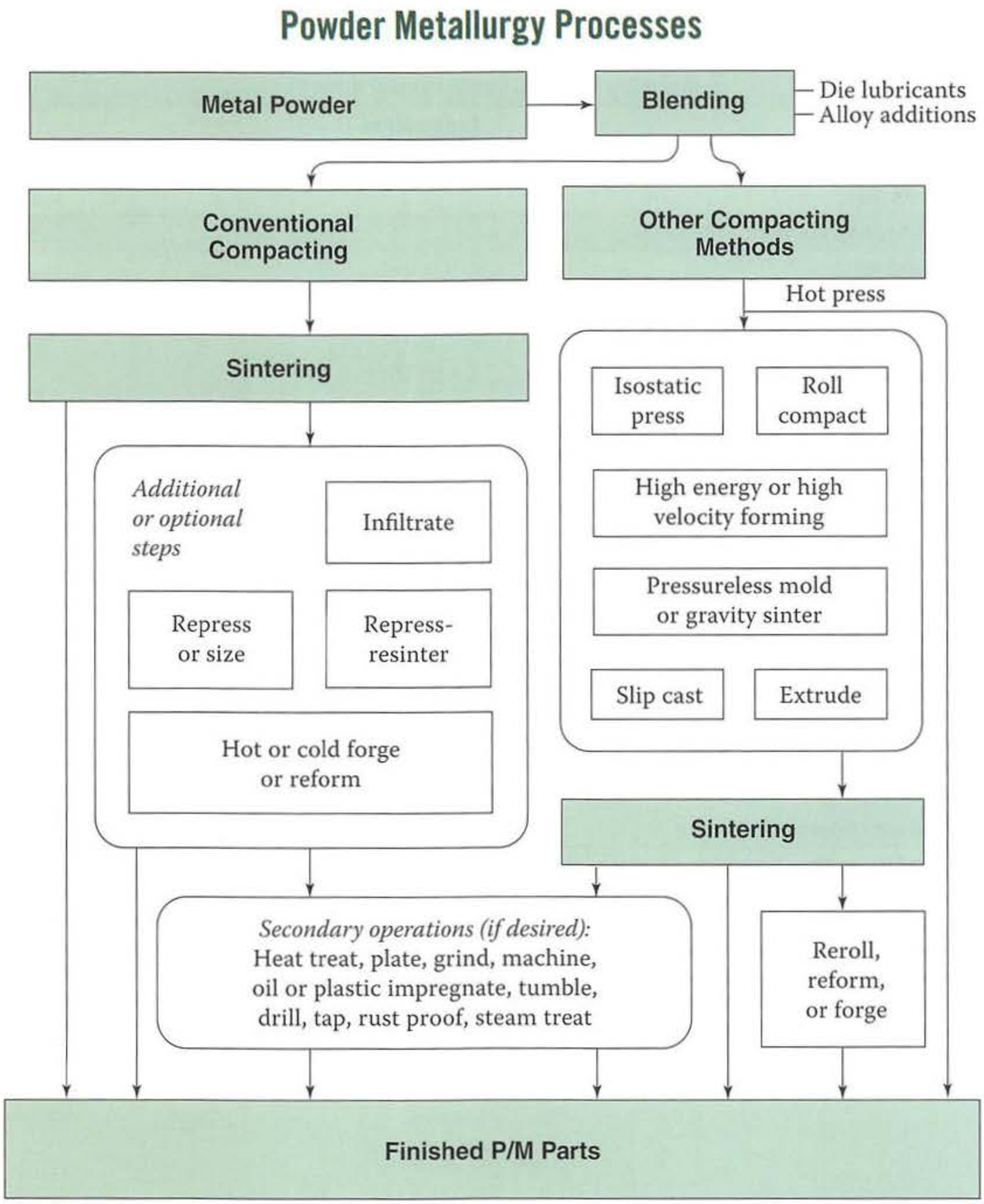
EIA and AIA National Codes for CNC Programming (Continued)	
Preparatory (G) Functions (Continued)	
G word	**Explanation**
G90	Selects absolute input. Input data is to be in absolute dimensional form.
G91	Selects incremental input. Input data is to be in incremental form.
G92	Preloads registers to desired values (for example, preloads axis position registers).
G93	Sets inverse time feed rate.
G94	Sets inches (or millimeters) per minute feed rate.
G95	Sets inches (or millimeters) per revolution feed rate.
G97	Sets spindle speed in revolutions per minute.
G98, G99	Unassigned.
Miscellaneous (M) Functions	
M word	**Explanation**
M00	Program stop. Operator must cycle start in order to continue with the remainder of the program.
M01	Optional stop. Acted upon only when the operator has previously signaled for this command by pushing a button. When the control system senses the M01 code, machine will automatically stop.
M02	End of program. Stops the machine after completion of all commands in the block. May include rewinding of tape.
M03	Starts spindle rotation in a clockwise direction.
M04	Starts spindle rotation in a counterclockwise direction.
M05	Spindle stop.
M06	Executes the change of a tool (or tools) manually or automatically.
M07	Turns coolant on (flood).
M08	Turns coolant on (mist).
M09	Turns coolant off.
M10	Activates automatic clamping of the machine slides, workpiece, fixture, spindle, etc.
M11	Deactivates automatic clamping.
M12	Inhibiting code used to synchronize multiple set of axes, such as a four-axis lathe that has two independently operated heads or slides.
M13	Combines simultaneous clockwise spindle motion and coolant on.
M14	Combines simultaneous counterclockwise spindle motion and coolant on.
M15	Sets rapid traverse or feed motion in the + direction.
M16	Sets rapid traverse or feed motion in the – direction.
M17, M18	Unassigned.
M19	Oriented spindle stop. Stops spindle at a predetermined angular position.
M20-M29	Unassigned.
M30	End of data. Used to reset control and/or machine.
M31	Interlock bypass. Temporarily circumvents a normally provided interlock.
M32-M39	Unassigned.
M40-M46	Signals gear changes if required at the machine; otherwise, unassigned.
M47	Continues program execution from the start of the program, unless inhibited by an interlock signal.
M48	Cancels M49.
M49	Deactivates a manual spindle or feed-override and returns to the programmed value.

(Continued)

Goodheart-Willcox Publisher

EIA and AIA National Codes for CNC Programming (Continued)	
Miscellaneous (M) Functions (Continued)	
M word	**Explanation**
M50-M57	Unassigned.
M58	Cancels M59.
M59	Holds the rpm constant at its value.
M60-M99	Unassigned.
Other Address Characters	
Address character	**Explanation**
A	Angular dimension about the X axis.
B	Angular dimension about the Y axis.
C	Angular dimension about the Z axis.
D	Can be used for an angular dimension around a special axis, for a third feed function, or for tool offset.
E	Used for angular dimension around a special axis or for a second feed function.
H	Fixture offset.
I, J, K	Centerpoint coordinates for circular interpolation.
L	Not used.
O	Used on some N/C controls in place of the customary sequence number word address N.
P	Third rapid traverse code—tertiary motion dimension parallel to the X axis.
Q	Second rapid traverse code—tertiary motion dimension parallel to the Y axis.
R	First rapid traverse code—tertiary motion dimension parallel to the Z axis (or to the radius) for constant surface speed calculation.
U	Secondary motion dimension parallel to the X axis.
V	Secondary motion dimension parallel to the Y axis.
W	Secondary motion dimension parallel to the Z axis.

Goodheart-Willcox Publisher



Sintering Sintering is a solid-state phenomenon in which powdered metal particles become metallurgically bonded below the melting point of the metal. No adhesives or cements are used.

Infiltration Pores of P/M parts are filled with a lower-melting-point metal such as copper-based alloy. When the part is sintered, the infiltrant material melts and penetrates into the P/M part by capillary action.

Coining and sizing Basically, this operation involves repressing sintered parts in a die similar to the original compacting die.

Impregnation The pores of the P/M part are filled with a lubricant or other nonmetallic such as plastic resin. This may be done by means of a vacuum or by soaking. The part then becomes self-lubricating (or pressure-tight if resin is used).

Goodheart-Willcox Publisher

Grinding Wheel Markings

32A46-H8VBE

Abrasive

Alundum =	A
19 Alundum =	19A
23 Alundum =	23A
32 Alundum =	32A
38 Alundum =	38A
44 Alundum =	44A
57 Alundum =	57A
75 Alundum =	75A
37 Crystolon=	37C
39 Crystolon=	39C

Grit Size

Coarse	Medium	Fine	Very Fine
10	30	70	220
12	36	80	240
14		90	280
16	46	100	320
20	54	120	400
24	60	150	500
		180	600

Grade

Soft	Medium	Hard
AE	IM	QV
BF	JN	RW
CG	KO	SX
DH	LP	TY
		UZ

Structure

(Sometimes omitted)

The structure number of a wheel refers to the relative spacing of the grains of abrasive; the larger the number, the wider the grain spacing.

Bond Type

V = Vitrified
S = Silicate
B = Resinoid
R = Rubber
E = Shellac

Norton Symbol

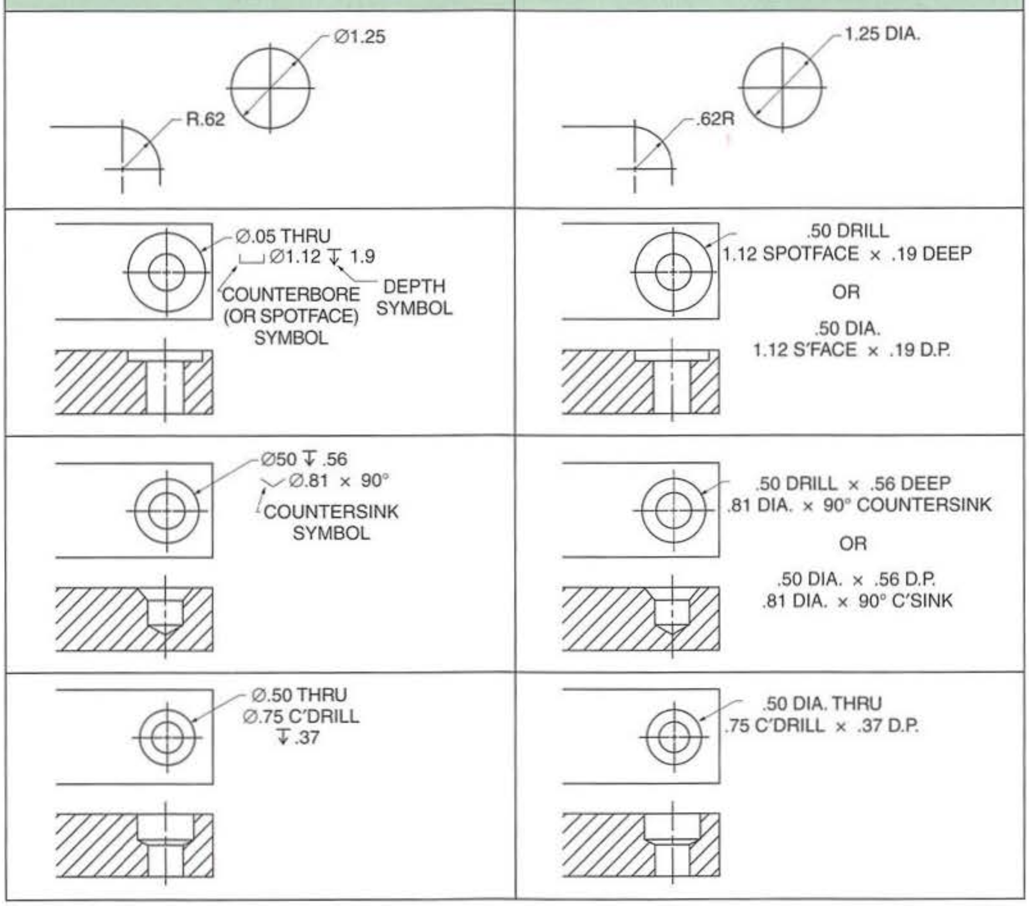
(Sometimes omitted)

Letter or numeral or both to designate a variation or modification of bond or other characteristic of the wheel. Typical symbols are "P," "G," "BE."

Norton Co.

Standard Symbols Used in Dimensioning

New	Old



Goodheart-Willcox Publisher

Glossary

A

abrasive. A material that penetrates and cuts a material that is softer than itself. It may be natural (emery, corundum, diamond) or artificial (silicon carbide, aluminum oxide). (7)

abrasive cutoff saw. A sawing machine that uses a rotary abrasive wheel with or without a liquid coolant. Wet abrasive cutting uses a liquid coolant, can cut to close tolerances, and produces a fine surface finish. Dry abrasive cutting uses no coolant and is used for rapid, less critical cutting. (11)

absolute positioning. A programming mode, indicated by the G90 code, in which the coordinate values for any point are interpreted in relation to the X0,Y0 position. (22)

accuracy. An indication of how close the value of a measurement is to the true value. (26)

actual size. The measured size of a part after it is manufactured. (27)

addendum. The distance a gear tooth extends above or outside the pitch circle. (18)

additive manufacturing. The process of using fused deposition technologies to build parts in layers. Similar to rapid prototyping processes, but additive manufacturing results in the creation of an actual part, rather than a model of the part. (25)

adhesive. A material that joins metals and keeps 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, and riveting. (8)

all-hard blade. One of two types of power hacksaw blades (the other is the flexible back blade). The all-hard blade is hardened throughout and is best for straight, accurate cutting under a variety of conditions. (11)

alloy. A mixture of two or more metals. (28)

Aluminum Association Designation System. A method of identifying aluminum using a four-digit code plus a temper designation. The identification code and temper designation are separated by a dash. (28)

American National Standards Institute (ANSI). An association that serves as a clearinghouse for nationally coordinated voluntary standards for fields ranging from information technology to building construction. Standards are established for areas such as definitions, terminology, symbols, materials, performance characteristics, procedures, and testing methods. (4)

American National Thread System. The common thread form used in the United States, characterized by the 60° angle formed by the sides of the thread. (7)

angularity. The condition of a surface, center plane, or axis at any specified angle from a datum plane or axis other than 90°. (27)

annealing. A heat-treatment process in which the metal is heated above its critical temperature and cooled slowly to reduce its hardness and make it easier to machine or work. (29)

anodizing. An electrolytic process that forms a protective layer of aluminum oxide on aluminum parts. (30)

apprentice. A student who receives on-the-job training by working with a skilled machinist while also studying work-related subjects. (2)

approved respirator. A type of mask worn over the mouth and nose that provides clean air to breathe and prevents the inhalation of dangerous toxins. (3)

arbor. A device used to hold and drive cutters on metalworking and other industrial machines. (17)

arbor press. A manually operated press used on smaller work that can have a leverage of several tons. (7)

assigned code. A code that has the same application or function on all machine controllers, regardless of application. (22)

associate's degree. A two-year educational degree focused on preparing students for technical positions. (2)

automatic accumulator. An automated device that collects and organizes finished parts after the CNC turning process. (24)

automatic screw machine. A variation of the lathe that was developed for high-speed production of large numbers of small parts. The machine performs a large number of operations either simultaneously or in a very rapid sequence. (15)

automatic tool changer (ATC). A device included on some CNC machine tools that automatically changes and stores the tools. (21)

B

bachelor's degree. A four-year educational degree focused on advanced theoretical education and skill training in specific disciplines such as engineering. (2)

backlash. Clearance designed into mating threaded rods and nuts to allow movement. (21)

bacteriostatic additive. An additive that inhibits the growth of bacteria. (10)

band machining. A widely used 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. (20)

bar puller. An attachment mounted in the turret of a turning center used to clamp and pull bar stock through the spindle for machining. (24)

base metal. A pure, nonferrous, nonprecious metal. (28)

bevel gear. A toothed wheel used to change the angular direction of power between shafts. The teeth may be either straight or curved. (18)

bilateral tolerance. A tolerance that allows the dimension to vary by a specific amount both above and below the stated dimension or size. (4)

Note: The number in parentheses following each definition indicates the chapter in which the term can be found.

bill of materials. A listing of the numbers, names, materials, and quantities of the parts specified on a set of working drawings. Compare *parts list*. (4)

blind hole. A hole that does not go completely through the workpiece. (7, 12)

block. In a word address format programming language, groups of words that are written on a single line and are meant to be executed together. (22)

bonus tolerancing. A method of applying the unused tolerance of a part that is not at LMC or MMC to its mating part to provide a larger acceptable tolerance for the mating part. (27)

boring. An internal machining operation in which a single-point cutting tool is used to enlarge a hole. (15)

boring mill. A huge machine capable of turning and boring work with diameters as large as 40′ (12 m). Work that is too large or too heavy to be turned in a horizontal position is machined on a vertical boring mill. (15)

box annealing. A method of annealing in which the work is placed into a metal box, which is then placed into the furnace, heated to the proper temperature, and allowed to cool inside the furnace. Keeping the work inside the box during heating and cooling prevents the work from scaling or decarbonizing. (29)

Brinell hardness test. A method of measuring a metal's resistance to deformation. In this test, a known load is applied to a metal through a steel ball of known size. The hardness reading is based on the diameter of the impression made by the steel ball. (29)

briquetting. In the powder metallurgy process, the operation that compresses and forms metal powder into the desired shape. (33)

broach. A long, multitoothed cutting tool with three kinds of teeth shaped to give a desired surface: rough, semifinished, and finished. (7)

broaching. A manufacturing process for machining flat, round, and contoured surfaces, both internal and external. A broach is pushed or pulled across the work, with each tooth removing only a small portion of the material. Cutting a keyway is typically a broaching process. (7)

broaching machine. A machine designed to push or pull a multitoothed cutter across the work; each tooth of the cutting tool (broach) removes only a small amount of material. (1)

burnishing. The process of finishing a metal surface by compressing the surface. It is often done by tumbling the work with steel balls. (20)

bushing. A bearing for a revolving shaft. Also, a hardened steel tube used on jigs to guide drills and reamers. (9)

C

calibration. Inspection and adjustment of measuring devices using gage blocks or other equipment that has been proved to be compliant with the NIST standards. (26)

canned cycle. A set of commands that follows a prescribed sequence again and again until the cycle is canceled. Also called *subroutine*. (23)

carbon content. The amount of carbon that a material contains. In reference to steel, it is measured in percent or in points. (28)

career. An occupation that requires specialized training and a commitment to the profession. (2)

Cartesian coordinate system. A coordinate system that specifies positive (+) and negative (−) movement along X, Y, and Z axes positioned at right angles to each other; used in CNC programming to specify movements of the workpiece or tool. (21)

case hardening. A heat-treatment process that creates a hard shell on the surface of low-carbon steel while leaving the inner core of the metal unaffected. (29)

center finder. A device used in a drilling machine to position the point to be drilled directly under the chuck or spindle. Also called *wiggler*. (12)

centerless grinding. A technique in which a workpiece is not supported between centers; instead, it is positioned on a work support blade and fed automatically between a regulating or feed wheel and a grinding wheel. (19)

chemical blackening. A process that chemically bonds a black oxide coating to the surface of ferrous metals. The coating improves corrosion and wear resistance, reduces glare from the surface, and improves the adhesion of subsequent layers of paint or other coatings. (30)

chemical blanking. A variation of the chemical milling operation that results in the removal of metal from certain areas by chemical action. Commonly used to cut parts out of ultrathin materials. (32)

chemical cutting fluid. Cooling and lubricating liquid that contains no oil. Because chemical cutting fluids are not actually fluids (graphite, mica, and white lead are examples), a wetting agent is often added to provide lubricating qualities. Compare *cutting fluid*. (10)

chemical machining. A category of processes that use chemicals, usually in an aqueous (with water) solution, to etch away selected portions of material. (32)

chemical milling. A process that uses a masking technique and a chemical etchant to remove metal selectively from relatively large surface areas. (32)

chipbreaker. A small groove cut into the face of a lathe tool near the cutting edge to break chips into small pieces. (14)

chipless machining. A manufacturing process that forms metal wire or rod into desired shapes using a series of dies. Also called *cold forming* or *cold heading*. (33)

chuck. A movable jaw mechanism that holds straight-shank drills in a drill press. (12)

circular interpolation. Movement in a circular or radial pattern (in the context of CNC programming). (22)

circularity. A form tolerance that is characterized by any given cross section taken perpendicular to the axis of a cylinder or cone, or through the common center of a sphere. (27)

circular pitch. The distance measured on the pitch circle between similar points on adjacent teeth. (18)

circular runout. A type of runout that provides control of single circular elements of a surface. (27)

class of fit. Standard working tolerances for thread accuracy, indicated by the last number on a thread description. Fits for inch-based threads are as follows: Class 1, loose fit; Class 2, free fit; Class 3, medium fit; and Class 4, close fit. (7)

climb milling. A milling technique in which the teeth of a cutting tool advance into the work in the same direction as the feed. Also called *down-milling*. (17)

closed-loop system. A system in which feedback is provided to the controlling mechanism; used with servomotors in CNC systems to provide feedback to the controllers regarding the position of the axis they are driving. (21)

cold circular saw. A tool that uses a circular, toothed blade capable of producing very accurate cuts. Large cold circular saws can be used to sever round metal stock up to 27″ (675 mm) in diameter. (11)

cold-finished steel. Steel that has been "pickled" or treated with a dilute acid solution to remove its oxide coating. After pickling, the steel is drawn or rolled to finished size and shape while cold. Cold-finished steel is characterized by a smooth, bright finish. (28)

column-and-knee milling machine. Milling machine that consists of a column that supports and guides the knee in vertical (up and down/Z-axis) movement and a knee that supports the mechanism for obtaining table movements. These movements are transverse (in and out/Y-axis) and longitudinal (back and forth/X-axis). (17)

combustible material. A solid, liquid, or gas that is capable of burning. Fires are classified according to the type of combustible materials that are burning. Class A fires involve ordinary combustible materials (paper, wood, textiles); Class B fires involve flammable liquids and grease; Class C fires involve electrical components; and Class D fires involve flammable metals, such as magnesium and lithium. (3)

comment code. Text in a CNC program that is enclosed in parentheses and is not read by the controller. Instead, the text appears on the controller screen as a reminder or prompt for the machine operator. (23)

composite. A relatively new class of material composed of fibers that are bonded together in a special plastic matrix under heat and pressure. Composite materials are generally lighter, stronger, and more rigid than many conventional metals. (28)

compound rest. A slide in the lathe located above a base cross-slide. The upper slide can be revolved to any required angular position. (14)

computer integrated manufacturing (CIM). A manufacturing system with computer-controlled machinery and adaptive tooling, which allows the system to be quickly adapted to changes in the product or the manufacturing process. Also called *flexible manufacturing system*. (25)

computer numerical control (CNC). A system in which a computer program is used to precisely position tools or the workpiece and to carry out the sequence of operations needed to produce a part. (1)

concentricity. The condition in which the axes of all cross-sectional elements of a cylindrical surface are common with the axis of a datum feature. (27)

conventional milling. A milling operation in which the work is fed into the rotation of the cutter. Also called *up-milling*. (17)

conversational language. An interface on CNC machines that allows the operator to select operations from menus, without having to understand G-code. (21)

coordinate measuring machine (CMM). An instrument that makes precise measurements electronically. A CMM can be used manually or programmed to check any number of individual reference points on an object against specifications. (26)

coordinate system. A method of locating specific points or positions in three-dimensional space. (21)

counterboring. The process of cutting a cylindrical enlargement of a hole to a given depth and diameter to allow bolt heads to be flush with the surface of the workpiece. (12)

countersinking. The process of chamfering a hole to receive a flathead screw. (12)

creep grinding. A machining technique that makes a deep cut into the work; usually performed in a single pass. (19)

critical temperature. The temperature at which a metal's crystal structure changes. Most heat-treatment processes require that the metal be heated to its critical temperature. (29)

cross-slide. A part of a machine tool that permits the carriage to make transverse tool movements. (14)

cryogenic. Having or associated with extremely low temperatures. The cryogenic temperature range is generally considered to be –300°F to –460°F (–184°C to –273°C). (33)

cutting fluid. A substance formulated to help cool and lubricate various metalworking tasks in order to improve the quality of the surface finish. There are four basic types of cutting fluids: mineral oils, emulsifiable (water-based) oils, chemical and semichemical cutting fluids, and gaseous fluids. (10)

cutting speed. In reference to milling machines, the distance, measured in feet or meters, that a point (tooth) on the cutter's circumference will move in one minute. In reference to lathes, 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. In reference to drilling machines, it is the speed at which the cutting tool rotates. (12, 14, 17)

cyanoacrylate quick-setting adhesive. A bonding agent known by brand names such as Eastman 910©, Super Glue©, and Crazy Glue©. It is used to hold matching metal sections together while they are being machined. (8)

cylindricity. A form tolerance identified by a radius tolerance zone establishing two perfectly concentric cylinders within which the actual surface must lie. (27)

D

datum. A theoretically exact point, axis, line, or plane. (27)

dedendum. The distance a gear tooth extends below the pitch circle. (18)

depth of cut. The distance the cutter is fed into the work surface. The depth of cut 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. (14)

destructive testing. A quality control process in which the part being tested is destroyed by the testing process. (26)

dial indicator. An instrument used to center and align work on machine tools, check for eccentricity, and inspect manufactured parts. There are two types of indicators: balanced indicators take measurements on either side of a zero line, and continuous indicators read from zero in a clockwise direction. (5)

diametral pitch. The number of gear teeth per inch of pitch diameter. (18)

diamond-edge band. A band machining tool designed for cutting material that is difficult or impossible to cut with a conventional toothed blade. The diamonds are only on the front edge of the band, where the cutting is accomplished. (20)

dielectric fluid. In electrical discharge machining, a fluid that forms a nonconductive barrier between the electrode and the work at the arc gap. (31)

dimensions. Exact sizes or measurements needed to produce a part. (4)

direct shell production casting (DSPC). A variation of the fused deposition modeling process that prints a three-dimensional model using sand or ceramics instead of plastic. The finished model can be used as a mold or mold core for casting. (25)

divider. A layout tool with pointed legs of equal length used to draw circles and arcs. (6)

drift. A tool used to separate sleeves, sockets, and taper shank drills from the drill spindle. (12)

drill gage. An instrument used to measure the diameter of a drill; drill gages are available for various drill series. (12)

drill point gage. An instrument used to ensure that a drill is correctly sharpened. (12)

drill press. A machine that rotates a drill against stationary material with sufficient pressure to cause the drill to penetrate the material. It is primarily used for cutting round holes. (1)

drilling machine. A power-driven machine that holds the material and cutting tool and brings them together to cut or enlarge a hole. (12)

dry cycle. Cycling through all programs of a CNC machine once during setup without a part in place to check for potential problems. (21)

dual dimensioning. A system in which both the US Conventional system units and SI Metric units are displayed on the same drawing. (4)

ductility. A property of metal that permits permanent deformation by hammering, rolling, and drawing without breaking or fracturing. (28)

dwell. A pause in the movement of the cutting tool after it has reached the final cutting depth. This helps to relieve pressure on the cutting tool and improves the finish. (19)

dye penetrant inspection. A nondestructive testing method in which a dye is applied to a part's surface and drawn into flaws by capillary action. The excess dye is removed and a developer is applied. The developer draws the dye back to the surface, marking the location of any flaw. (26)

E

eddy-current inspection. A nondestructive testing method that detects flaws by measuring changes in impedance in a test coil placed near the test sample. A material without flaws will generate a different impedance than a material with flaws. (26)

electrical discharge machining (EDM). A process in which material is eroded from a workpiece by an electric arc between the electrode and workpiece. (31)

electrical discharge wire cutting (EDWC). A machining process that removes material from the workpiece by creating an arc between a wire electrode and the workpiece. It is similar in function to band machining. (31)

electrochemical machining (ECM). A process that removes material from the workpiece through electrolysis. In this process, an electrolyte surrounds the electrode and workpiece, which are electrically charged. (31)

electrode. In electrical discharge machining, a negatively charged tool. When voltage builds to sufficient strength, an arc jumps from the electrode to the workpiece, eroding a small amount of metal from the workpiece. (31)

electrohydraulic forming. A type of high-energy-rate forming in which a high-voltage discharge from a capacitor bank vaporizes an initiation wire and surrounding wire. The gases formed during the vaporization expand rapidly, applying pressure to the outer surface of the workpiece and forcing it into a die. Also called *capacitor discharge forming* or *spark forming*. (33)

electrolytic grinding. A form of electrochemical machining in which electric current is passed between a metal-bonded grinding wheel (the cathode) and the work (the anode) through an electrolyte to dissolve the work. The dissolved material is removed by the grinding wheel. (19)

electromachining. A category of machining processes in which electric current is used to remove material through erosion rather than by direct physical contact. (31)

electron beam machining (EBM). A process that uses a high-energy, highly focused beam of electrons to weld or cut materials. (32)

electroplating. A process in which an electric current is used to deposit a layer of one metal on the surface of another metal. (30)

emulsifiable oil. Cutting fluid composed of oil droplets that are suspended in water by blending the oil with emulsifying agents and other materials. Provides increased cooling capacity over straight cutting oils with reduced misting and fogging. (10)

encoder. A transducer in the motor that drives a CNC axis that measures the position of the moving axis and provides electronic feedback to the controller. (21)

engineer. A trained professional who uses applied mathematics, science, and knowledge of manufacturing to design industrial products and processes. (2)

etchant. The fluid used to erode material in a chemical machining process; usually a strong, aqueous alkaline solution. (32)

explosive forming. A type of high-energy-rate forming in which the material is shaped by pressure resulting from the detonation of an explosive charge. The workpiece, die, and explosive charge are submerged in water, which distributes the force of the blast evenly over the outer surface of the workpiece. (33)

external thread. A screw thread cut on an outside surface. (16)

F

face milling. Machining large, flat surfaces parallel to the cutter face. (17)

facing. In lathe work, cutting across the end of a workpiece to create a flat surface. (14)

fastener. Any device used to hold two objects or parts together, such as bolts, nuts, screws, pins, keys, rivets, and chemical bonding agents or adhesives. (8)

feature. A general term applied to a physical portion of a part. (27)

feature control frame. A symbol used to define the geometric tolerancing characteristics of a feature. It contains the geometric characteristic symbol, allowable tolerance, and datum reference letter(s). (27)

feed. The rate at which work moves into the cutter or the cutter moves into the work. (12, 14, 17)

ferrous. Containing iron. (28)

file band. A band machine accessory used to obtain a smooth, uniformly finished surface. A series of small file segments make up the file band. The segments interlock to form a continuous file. (20)

fixed-bed milling machine. A type of milling machine that has a very rigid worktable construction and support. The worktable moves only in a longitudinal (back and forth/X-axis) direction and can vary in length. Vertical (up and down/Z-axis) and cross (in and out/Y-axis) movements are obtained by moving the cutter head. (17)

fixture. A device for holding work rigidly while machining operations are performed. It does not guide the cutting tool. (9)

flatness. A measure of the variation of a surface perpendicular to its plane. (27)

flexible back blade. A blade on which only the teeth are hardened. It is used on metal-cutting saws when safety requirements demand a shatterproof tool. These blades are also used for cutting odd-shaped work if there is a possibility of the work coming loose in the vise. (11)

flexible manufacturing system (FMS). A manufacturing system with computer-controlled machinery and adaptive tooling, which allows the system to be quickly adapted to changes in the product or the manufacturing process. Also called *computer integrated manufacturing.* (25)

flutes. Two or more spiral grooves machined into a cutting tool to form the cutting edges of the drill point, facilitate easy chip removal, and permit cutting fluid to reach the cutting point. (12)

follower rest. Similar to a steady rest except it provides support directly in back of the cutting tool and follows along during the cut. Compare *steady rest.* (15)

foot-pound (ft·lb). The US Conventional unit of measurement for torque. (7)

form grinding. A cutting operation in which the grinding wheel is shaped to produce the required contour on the work. (19)

form tolerances. Tolerances that control the straightness, flatness, circularity, or cylindricity of a geometric shape. (27)

friction saw. A metal-cutting tool with a blade that may or may not have teeth. The saw operates at very high speeds (20,000 surface feet or 6000 meters per minute) and actually melts its way through the metal. (11)

fused deposition modeling (FDM). A rapid prototyping technique that produces three-dimensional objects based on CAD-generated solid or surface models. A temperature-controlled head extrudes thermoplastic material layer by layer. (25)

G

gage blocks. Precisely made steel blocks used by industry as a standard of measurement. They are made in a range of sizes and with a dimensional accuracy of ± 0.000002 (two millionths) inch, with a flatness and parallelism of ±0.000003 (three millionths) inch. Also called *Jo-blocks* or *Johansson blocks.* (5)

gaging. Checking parts using various gages to determine whether the pieces are made within specified tolerances. (5)

gang milling. Using two or more milling cutters to machine several surfaces at one time. (18)

gaseous fluid. A type of cutting fluid composed of gas, rather than liquid. Compressed air is the most commonly used. (10)

G-codes. Preparatory codes in the ANSI/EIA 274D code format that position the machining cutter. (22)

geometric dimensioning and tolerancing (GD&T). The control of the size of the features of a part and the allowances (either oversize or undersize) to achieve interchangeable manufacturing. (27)

geometric tolerance. A general term that refers to tolerances that control form, profile, orientation, or location of a feature. (27)

glazing. A process in which the grains of a grinding wheel become dull before they can wear away, leaving a shiny surface and causing inefficient cutting action. (13)

graduations. Lines that indicate measurement points on tools and machine dials. (5)

grinding. An operation that removes material by rotating an abrasive wheel or belt against the work. (13)

grinding machine. A machine that removes material from work by means of a rotating grinding wheel made of abrasive particles or an abrasive belt. Also called *grinder.* (1)

grooving. The turning operation of cutting a groove or recess into a workpiece to terminate a thread or provide adequate clearance for mating parts. (14)

H

hardening. A heat treatment that reduces the ductility of steel. It is performed by heating metal to the proper temperature and then quenching it. (29)

headstock. On a lathe, the structure that contains the spindle to which various work-holding attachments are fitted. (14)

heat treatment. The controlled heating and cooling of a metal or alloy to obtain certain desirable changes in its physical characteristics. These changes include improving resistance to shock, developing toughness, and increasing wear resistance and hardness. (29)

hermaphrodite caliper. A layout tool with one leg that is shaped like a caliper and the other pointed like a divider; used to scribe lines parallel to the edge of material and to locate the center of irregularly shaped stock. (6)

high-carbon steels. Steels that contain 0.6%–1.5% carbon. They are used in products that must be heat-treated. (28)

high-energy-rate forming (HERF). A manufacturing process in which a material is formed around a die by the rapid application of extreme pressures. (33)

high-speed steels (HSS). Tool steel that has red hardness and a high resistance to abrasion. Used in some cutting tools. (28)

honeycomb sandwich panels. A material made by bonding a honeycomb panel between two flat panels. Honeycomb sandwich panels have a very high strength-to-weight ratio and rigidity-to-weight ratio. (28)

horizontal band saw. A tool with a long, continuous blade that moves in only one direction. Cutting is continuous, and the blade can run at very high speeds because it rapidly dissipates the cutting heat. Also called *cutoff machine.* (11)

horizontal machining center (HMC). A machining center with a spindle oriented horizontally. (21)

horizontal milling machine. A type of milling machine in which the cutter is fitted onto an arbor mounted in the machine on an axis parallel with the worktable. Multiple cutters may be mounted on the arbor for some operations. (17)

hot-rolled steel. Steel that has been rolled to finished size while hot. Hot-rolled steel is identifiable by its black oxide surface scale. (28)

hydrodynamic machining (HDM). A machining process that uses a high-velocity, high-pressure stream of water to cut through materials. Also called *water jet cutting.* (32)

I

impact machining. A machining process that removes metal by using ultrasonic frequencies and a tool that forces abrasives against the work. Also called *slurry machining.* (32)

incremental positioning. A programming mode, indicated by the G91 code, in which the coordinate values for any point are interpreted relative to the location of the previous point. (22)

indexable insert cutting tool. Insertable piece that contains multiple cutting edges for a cutting tool. Widely used for turning and milling operations, the inserts are manufactured in a number of shapes and sizes for different turning geometries. As an edge dulls, the next edge is rotated into position until all of its edges are dulled; the insert is then discarded. (14)

internal grinding. A cutting operation done to produce a fine surface finish with high accuracy on inside diameters. The work is mounted in a chuck and rotated. During the grinding operation, the revolving grinding wheel moves in and out of the hole. (19)

internal thread. A screw thread cut on the inside surface of a piece. Internal threads are made on the lathe with a conventional boring bar and a cutting tool sharpened to the proper shape. (16)

International System of Units (SI). The metric system of weights and measures. Abbreviated SI (*Système International*). Also called *SI Metric system.* (4)

J

jig. A device that guides a cutting tool and aligns it with the workpiece so that all parts produced are uniform and within specifications. (9)

job shop. A machine shop where specialized or experimental work is machined or where the production runs are very small. (2)

just-in-time (JIT) inventory system. An inventory management system in which parts and materials are scheduled for arrival at the time they are needed, and not before. (25)

K

key. A small piece of metal embedded partially in the shaft and partially in the hub to prevent rotation of the gear or pulley on the shaft. (8)

keyway. The slot or recess in a shaft that holds the key. (7)

knife-edge blade. A vertical band machine tool used to cut material that would tear or fray if machined by a conventional blade. Such materials include sponge rubber, cork, cloth, corrugated cardboard, and rubber. (20)

knurling. The process of impressing diamond or straight-line patterns onto a metal surface by rolling with pressure to improve the appearance and provide better grip. The rolls are called *knurls.* (15)

L

laminated object manufacturing (LOM). A rapid prototyping technique that uses progressive layers of inexpensive solid sheet material, each layer bonded to the previous layer, to form the model. (25)

laser. A device that produces a narrow, intense beam of light; *laser* stands for "Light Amplification by Stimulated Emission of Radiation." (32)

lathe. A machine in which a workpiece in a work-holding device is rotated while a stationary cutting tool is forced against it. Some operations performed on a lathe include turning, boring, facing, thread cutting, drilling, and reaming. (1)

lathe center. A pointed work-holding device used to accurately align the workpiece along an axis. More frequently referred to as a *center.* (14)

lathe dog. A device that clamps to one end of the workpiece and drives it so that it can be machined between centers. (14)

lay. The direction of the predominant tool marks, grain, or pattern of surface roughness. (30)

layout. The process of locating and scribing points for machining and forming operations. (6)

layout dye. A coating applied to metal to make layout lines more visible. (6)

lead. The distance a nut advances on a screw in one revolution. (16)

lead screw. A rotating screw that translates the rotating motion of an electric motor into straight-line (linear) motion. (21)

lean manufacturing work cell. A manufacturing layout developed by Toyota that includes a U-shaped "work cell" in order to minimize inventory buildup and the number of workers required while maximizing output. (25)

least material condition (LMC). The condition in which a feature contains the least amount of material within the stated size limits. (27)

limit dimensions. Dimensions in which only the highest and lowest possible values for the dimension are provided. The basic dimension is not stated. (4)

linear interpolation. Straight-line movement (in the context of CNC programming). (22)

loading. A process in which small pieces of soft metal clog the surface of the grinding wheel, causing inefficient cutting action. (13)

location tolerances. Tolerances to locate features from datums or to establish coaxiality or symmetry. (27)

lockout/tagout procedures. A series of steps to stop equipment and remove its energy source to prevent accidental injuries while the equipment is being serviced or maintained. (21)

low-carbon steels. Steels that contain less than 0.3%, or 30 points, of carbon. Low-carbon steel does not contain enough carbon to be hardened. However, case hardening is possible if the surface is exposed to an external source of carbon during the process. Also called *mild steel.* (28)

M

machine backlash. A hesitation or failure to begin moving immediately when an axis is reversed. It is caused by the clearance between the drive screw and the mating nuts that move the axes and increases with wear over time. (18)

machine control unit (MCU). Also simply called a *control,* the part of a CNC machine that reads the CNC program from memory and translates it into the electronic signals needed for machine operation. (22)

machine home. For purposes of CNC programming, the location of the table and spindle on the CNC machine, given in Cartesian coordinates. (23)

machine shield. A type of barrier on a machine that blocks flying chips or splashing cutting fluids or coolants from escaping and harming the machinist. (3)

machine tools. That class of machines which, taken as a group, can reproduce themselves. (1)

machining center. A CNC milling machine equipped with an automatic tool changer (ATC). (21)

machinist. A person who is skilled in the use of machine tools and is capable of making complex machine setups. (1)

magnetic forming. A type of high-energy-rate forming in which the magnetic force from a specially shaped induction coil provides the pressure needed to shape the material. Also called *electromagnetic forming* or *magnetic pulse forming.* (33)

magnetic particle inspection. A nondestructive testing technique that uses iron particles and induced magnetic fields to detect flaws on or near the surface of ferromagnetic (iron-based) materials. (26)

major diameter. The largest diameter of a thread measured perpendicular to the axis. (16)

mandrel. A slightly tapered, hardened steel shaft that supports work machined between centers. (15)

manual data input (MDI). A method of programming CNC programs by entering the program codes at the machine control unit. (22)

maximum material condition (MMC). The condition in which a feature contains the maximum amount of material within the stated size limits. (27)

M-codes. Miscellaneous codes in the ANSI/EIA 274D code format that control machine functions in CNC programming. (22)

medium-carbon steels. Steels that contain 0.30% to 0.60% (30 to 60 points) carbon. The carbon content is sufficient to allow partial hardening with proper heat treatment. (28)

megahertz (MHz). A unit of measure equal to a million cycles per second. Used to measure ultrasound frequencies. (26)

metal finish. The degree of smoothness or roughness remaining on the surface of a part after it has been machined. (30)

metal spraying. A process in which a metal wire or powder is heated to its melting point and sprayed by air pressure to produce the desired coating on the work surface. (30)

metrology. The science that deals with systems of measurement. (5)

microinch. One millionth of an inch (0.000001″). (4, 30)

micrometer. One millionth of a meter (0.000001 m). Also called *micron.* (4, 30)

micrometer caliper. A precision tool capable of measuring to 0.001″ (0.01 mm). When fitted with a vernier scale, it will read to 0.0001″ (0.002 mm). (5)

mild steel. Steel that contains less than 0.3%, or 30 points, of carbon. Mild steel does not contain enough carbon to be hardened. However, case hardening is possible if the surface is exposed to an external source of carbon during the process. Also called *low-carbon steel.* (28)

milling machine. A machine that removes material from a workpiece by means of a rotary cutter. (1)

mineral oil. Cutting fluid best suited for light-duty (low speed, light feed) operations where high levels of cooling and lubrication are not required. (10)

minor diameter. The smallest diameter of a screw thread, measured across the roots and perpendicular to the axis. Also called *root diameter.* (16)

mist coolant. Cutting fluid applied on a band machine by flooding in mist form. It is used for high-speed sawing of free-machining nonferrous metals. It is also used for tough, hard-to-machine materials. (20)

modal command. A G-code command that, once activated, remains activated until the machine encounters another modal command in the CNC program. (22)

Mohs hardness scale. A system of determining the relative hardness of materials by comparing the ability of harder materials to scratch softer materials. (33)

N

newton-meter (N·m). The SI Metric unit of measurement for torque. (7)

nitriding. A process in which nitrogen is diffused into a base metal to create a case-hardened surface. (30)

nondestructive testing. A quality control process in which the usefulness of the product is *not* impaired. Also called *nondestructive examination* or *nondestructive inspection.* (26)

nonferrous. Containing no iron. (28)

normalizing. A process that refines the grain structure of some metals and thereby improves their machinability. It is closely related to the annealing process. (29)

numerical control (NC). A system (composed of a control program, a control unit, and a machine tool) that controls the actions of the machine through coded command instructions. (1)

O

Occupational Safety and Health Administration (OSHA). A government agency responsible for setting and enforcing regulations regarding safety and health in the workplace. (3)

offhand grinding. A grinding operation done on a bench, pedestal, or belt grinder on work that does not require great accuracy. The work is handheld and manipulated until ground to the desired shape. (13)

offline programming. Entering a program using a computer that is not currently being used to control the operation of a CNC machine. (22)

offset tailstock method. A method of machining external tapers on a lathe. Jobs that can be turned between centers may be taper-turned using this technique. (16)

open-loop system. A system in which no feedback is provided to the controlling mechanism; used with stepper motors in relatively inexpensive CNC machines. (21)

optical comparator. A gaging system in which an enlarged image of the part being inspected is projected onto a screen. The projected image can be superimposed on a grid or an accurate drawing overlay of the part to allow precise measurement of the part. (26)

orientation tolerances. Tolerances that control the degree of parallelism, perpendicularity, or angularity of a feature with respect to one or more datums. When controlling orientation tolerances, the feature is related to one or more datum features. (27)

P

parallelism. The condition of a surface or center plane equidistant at all its points from a datum plane or axis. (27)

part catcher. A programmable device mounted to turning centers to catch finished parts as they are cut off the bar stock. (24)

parting. The turning operation of cutting off material after it has been machined. This is one of the more difficult operations performed on a lathe. (14)

part programmer. A skilled worker who specializes in writing programs for computer-controlled machine tools. (2)

parts list. A list of all the parts needed to make an assembly, including the number of each type of part needed. Compare with "bill of materials." (4)

peck drilling. A method of drilling in which the cutter is retracted at intervals to allow chips to be cleared or coolant to be flooded through the hole. (23)

peripheral milling. A milling operation in which the surface being machined is parallel with the periphery of the cutter. Also called *edge milling*. (17)

perpendicularity. The condition of a surface, center plane, or axis at a right angle (90°) to a datum plane or axis. (27)

pilot hole. A smaller hole drilled prior to drilling a larger hole that greatly reduces feed pressure, improves accuracy, and allows faster drilling. Also called *lead hole*. (12)

pitch. The distance from a point on one thread or gear tooth to the corresponding point on the next thread or tooth. (16)

pitch circle. An imaginary circle located approximately half the distance from the roots and tops of gear teeth. It is tangent to the pitch circle of the mating gear. (18)

pitch diameter. For threads, the diameter of an imaginary cylinder that would pass through the threads at such points as to make the width of the thread and the width of the thread groove equal. For gears, the diameter of the pitch circle. (16)

plain turning. Turning in which the entire length of the piece is machined to a specified diameter. (14)

platens. Flat metal plates used to provide support behind the belt of an abrasive belt grinder. (19)

plunge grinding. Grinding method in which work is mounted between centers and rotated while in contact with the grinding wheel. The area being ground is no wider than the wheel face. (19)

polar coordinate system. A coordinate system in which straight-line distance and travel angle are used to specify locations or movement. (21)

position tolerance. A tolerance used to define a zone in which the center, axis, or center plane of a feature of size is permitted to vary from true position. (27)

post-processing. The process of translating a CAM program into a format the CNC controller can understand. (22)

pot broaching. A broaching method in which the tool remains stationary and the work is pushed or pulled through the broach. (20)

powder metallurgy (P/M). A manufacturing process used to make parts by compressing and heating metal powders. Also called *sintering*. (33)

precision. An indication how close two or more results are to each other, or the repeatability of a measurement. (26)

precision grinding. A finishing operation in which a minute amount of material is removed with each pass of the grinding wheel to generate a smooth, accurate surface. (19)

process annealing. Removing internal stresses that have developed in parts that have been cold-worked, machined, or welded. Also called *stress relieving*. (29)

profile of a line tolerance. A two-dimensional or cross-sectional geometric tolerance that extends along the length of the feature. (27)

profile of a surface tolerance. A geometric tolerance that controls the entire surface of a feature or object as a single entity. (27)

profile tolerance. A tolerance that specifies a uniform boundary along the true profile within which the elements of the surface must lie. (27)

profilometer. An electronic instrument for measuring surface roughness. (4)

program zero position. For purposes of CNC programming, a reference point specified on the part or on the work-holding device as the zero position. (23)

programmable logic controller (PLC). A small, digital computer that acts as a communications hub for flexible manufacturing cells. Using various sensors and machine-specific codes from CNC equipment, the PLC manages the communications process between different machines (CNC machines, robots, conveyance systems, etc.) within a flexible manufacturing cell. (25)

protective clothing. Clothing that is worn in a machine shop to protect the body. Safety glasses and hearing protectors are two of the most important articles because shop areas produce both noise and flying chips. Other protective clothing includes steel-toed shoes, lead aprons, caps or hairnets, and respirators. (3)

protractor. An angle-measuring tool used in layout work when angles do not need to be laid out or checked to extreme accuracy. The head is graduated from 0° to 180° in both directions for easy reading. (6)

Q

quality control (QC). A process that identifies and prevents potential product defects in the manufacturing process before they can cause injuries or damage or result in substandard products. (26)

quench. To cool a metal rapidly by immersing it in a fluid or spraying a fluid on its surface. (29)

R

radiographic inspection. A nondestructive testing method that uses X rays and gamma radiation to detect flaws such as cracks or pores in an object. Also called *X-ray inspection.* (26)

raker set. A three-tooth saw set in which one tooth is angled toward the left, the next one straight, and the next one angled toward the right, alternating continuously along the length of the blade. Raker set is recommended for cutting large solids or thick plate and bar stock. (20)

reamer. A cutting tool used to enlarge, smooth, and size a drilled hole by removing a small amount of metal. (7)

reaming. Finishing a drilled hole with a reamer. (12)

red hardness. The ability of a metal to remain hard even when red-hot. (28)

reference line. A layout line from which all measurements are made. Also called *baseline.* (6)

regardless of feature size (RFS). Formerly indicated by an S within a circle; specifies that the size of a feature tolerance must not be exceeded. Today, RFS is assumed for all geometric tolerances unless otherwise specified. (27)

repetitive cycle. Predefined operations specified by G-codes. These operations are found on most turning centers and are similar to the canned cycles used on CNC machining centers. (24)

résumé. A summary of an applicant's educational and employment backgrounds. (2)

ring test. A test that involves suspending a grinding wheel and tapping it to produce a sound. If the sound is a clear ring, the grinding wheel is safe. A grinding wheel that produces a dull thud is not safe and should not be used. (13)

robot. A reprogrammable, multifunctional manipulator designed to move material, parts, tools, or specialized devices through variable programmed motions to perform a variety of tasks. (25)

Rockwell hardness test. The most widely used hardness testing method. A penetrator is pressed into the metal being tested under a known load. The hardness reading is based on the depth of penetration. (29)

roller burnishing. A cold-working operation that uses rollers to flatten the peaks of a metal's surface into the valleys. With this process, no honing or grinding is necessary. (30)

roughing cut. Deep cut made to remove considerable material from a workpiece in a single pass. (14)

runout tolerance. A combination of geometric tolerances used to control the relationship of one or more features of a part to a datum axis. (27)

S

safe edge. A file edge without teeth. (7)

safety data sheet (SDS). Document that provides information about the various chemicals, chemical compounds, and mixtures used in the industrial setting. Every shop is required to keep an SDS for every chemical used in the shop readily available to employees and emergency response personnel in event of an emergency. (3)

safety equipment. Tools or equipment that help prevent or mitigate accidents in the potentially dangerous environment of a machine shop. Machine guards, fire extinguishers, eye wash stations, vacuum dust collectors, power switches (for locking off equipment), and brushes (for removing machine chips) are a few examples. (3)

sample term. A term used as a sample. Also called *model term.* (1)

sawing machine. A machine that cuts away material from the work by means of a multitoothed saw blade. (1)

scale drawing. A drawing made at a size other than the actual size of the part (1:1). A drawing made one-half size has a scale of 1:2. A scale of 2:1 means that the drawing is twice the size of the actual part. (4)

scleroscope. A testing device that drops a hammer onto the test piece. The resulting rebound of the hammer determines the hardness of the material being tested. (29)

scriber. A layout tool with a point of hardened steel used to scribe (scratch) fine, accurate lines into metal. (6)

semichemical cutting fluid. Cooling and lubricating fluid that may have a small amount of mineral oil added to improve its lubricating qualities. (10)

semiskilled worker. A worker who performs basic, routine operations that do not require a high degree of skill or training. (2)

servomechanism. In electrical discharge machining, the drive unit used to accurately control electrode movement and to maintain the correct distance between the work and the electrode as machining progresses. (31)

servomotor. A motor that can provide feedback to a controlling mechanism; used in higher-end CNC machines to drive axis positions. (21)

set. On a saw blade, the orientation of the teeth; the set provides the necessary clearance and prevents the blade from binding in the cut. (7)

setover. The distance a lathe tailstock is offset from the normal centerline of the machine. It is a method of turning a taper. (16)

side milling cutter. A type of milling cutter with cutting edges located on the circumference and on one or both sides of the cutter. They are made in solid form or with inserted teeth. (17)

SI Metric system. The metric system of weights and measures. (*SI* stands for the French words *Système International*). Also called *International System of Units (SI)*. Compare *US Conventional system*. (4)

sine bar. A tool used with a surface plate and gage blocks to create an accurate right triangle with a specific size and angles in order to lay out angles very precisely. (6)

single-point cutting tool. A cutting tool with one face and one cutting edge. (14)

sintering. In the powder metallurgy process, the operation in which a newly formed part is heated in a controlled-atmosphere furnace to give it strength. *Sintering* may also be used to refer to the powder metallurgy process itself. (33)

skilled worker. A worker who has been trained, often as an apprentice, to do more complex tasks. (2)

skill standards. Industry requirements for skilled workers and the basis for industry-recognized certification obtained through performance testing. (1)

slitting. An operation in which a thin cutter or rotary knife is used to cut sheet metal into narrow strips. (18)

slotting. Similar to slitting, except that the cut is made only part way through the work. The slot in a screw head is an example of slotting. (18)

small-hole EDM drilling. A process that bores small-diameter holes in material by creating an arc between a hollow, spinning electrode and the workpiece. (31)

smart tooling. Cutting tools and work-holding devices that can be easily reconfigured to produce a variety of shapes and sizes within a given part family. (25)

spindle. On a lathe, the structure that receives tools, attachments, or the workpiece, revolves in heavy-duty bearings, and is rotated by belts, gears, or a combination of the two. (14)

spontaneous combustion. Ignition by rapid oxidation or burning of oil without an external source of heat. (3)

spotfacing. Machining a circular spot on the surface of a part to furnish a flat bearing surface for mounting a bolt head or nut head. (12)

spur gear. A wheel with teeth that run straight across the gear face, perpendicular to the sides. (18)

square. A tool used to check 90° (square) angles and to lay out lines that must be at right angles to a given edge or parallel to another edge. (6)

statistical process control (SPC). A quality control technique in which a percentage of the products made during a production run are tested. Statistical analysis of variations detected in the products allows the manufacturer to predict and correct problems before they result in unacceptable products. (26)

steady rest. A support for long, thin workpieces that keeps the work from springing or bending away from the cutting tool. The rest also reduces chatter when long shafts are machined. Compare *follower rest*. (15)

steel rule. A measuring tool, available in several basic types of graduations, including fractional inch, decimal inch, and metric. (5)

stepper motor. A motor that moves in small steps, or increments, and does not provide feedback to the controlling mechanism. (21)

stereolithography. A rapid prototyping technique that uses a computer-guided low-power laser beam to harden a liquid photocurable polymer plastic into the programmed shape. (25)

straddle milling. Using milling cutters in pairs to machine opposite sides of a piece simultaneously. (18)

straightedge. A precision tool for checking the accuracy of flat surfaces. (6)

straightness. The measure of how close an element of a surface or axis is to a perfectly straight line. (27)

straight set. A two-tooth saw set in which one tooth is angled to the side and the next one straight, alternating continuously along the length of the blade. Straight set is recommended for materials such as aluminum and magnesium. (20)

stress-relieving treatment. Removing internal stresses that have developed in parts that have been cold-worked, machined, or welded. Also called *process annealing*. (29)

subroutine. A set of commands that follows a prescribed sequence again and again until canceled. Also called *canned cycle*. (23)

surface gage. A scribing tool used to make lines at a given height and parallel to a surface, and to check whether a part is parallel to a given surface. (6)

surface hardening. A heat-treatment process used to create a medium-hard surface on high-carbon or alloy steel while leaving the inner core of the metal unaffected. (29)

surface plate. A cast iron or granite plate, ground or lapped to a smooth, flat surface and used for precision layout and inspection. (6)

surface roughness standards. Documents that specify how surfaces produced by machining, grinding, casting, molding, forging, or similar processes are to be measured and communicated. (30)

swing. The largest diameter that can be turned over the ways of a lathe. Along with the length of the bed, the swing determines the size of the lathe. (14)

Swiss-type turning center. A type of turning center or lathe that can produce elaborate detail in machined parts with a high degree of precision and accuracy. (21)

symmetry. A relationship that indicates equal or balanced proportions on either side of a central plane or datum. (27)

T

tailstock. A movable lathe fixture that mounts on ways to support the workpiece between centers. It can be fitted with tools for drilling, reaming, and threading. (14)

taper. A uniform increase or decrease in slope that causes a piece to assume a wedge or conical shape. (16)

taper attachment. A guide attached to a lathe and used to accurately cut internal and external tapers. (16)

tapping. Forming an internal screw thread in a hole or other part by means of a tap. Also, opening the pouring hole of a melting furnace to remove molten metal. (12)

technician. A specialist in the technical details of an occupation who assists the engineer by testing various experimental devices and machines, compiling statistics, making cost estimates, and preparing technical reports. (2)

temper. The hardness and strength of a rolled metal. (13)

tempering. A heat-treatment process that reduces a metal's brittleness or hardness. In this process, metal is heated to a temperature below its critical temperature and then quenched. (29)

thermal expansion. The rate at which a material expands and contracts in response to temperature changes. (9)

threadcutting stop. A device used to stop the threadcutting operation on a lathe so the tool can be more easily removed from the work after each cut and repositioned before the next cut is started. (16)

thread dial. A dial that meshes with the lead screw of a lathe and indicates when to engage the half-nuts, which permits the tool to follow exactly in the original cut. (16)

threaded fastener. A bolt or similar device that uses the wedging action of a screw thread to clamp parts together. 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. (8)

thread rolling. The process of creating screw threads on external diameters by deforming the material to create the screw thread. (16)

three-tooth rule. A guideline for power metal-cutting saws that says at least three teeth must be in contact with the work at all times. (11)

three-wire method. A—To accurately measure thread size, three wires of a specific diameter are fitted into screw threads, and a micrometer measurement is made over the wires. B—A mathematical formula is used to calculate the correct measurement. (16)

tolerance. A permissible deviation from a basic dimension. (4)

tombstone. A fixture-holding device commonly used with machining centers and other CNC machine tools. Tombstones are made from heavy castings and are precisely machined. (9)

toolmaker. A highly skilled machinist who specializes in producing the tools and tooling needed for machining operations. (2)

tool post. Mounts the cutting tool on the carriage of a lathe. (14)

tool rest. The part of a grinder that supports the workpiece during grinding operations. (13)

tooth form. The shape of the tooth on a band machine saw. There are three basic forms: standard, skip, and hook. (20)

tooth rest. Support mounted on the worktable or on the grinding wheel housing that places each tooth of a cutter quickly and accurately into position for sharpening. (19)

torque. The amount of turning or twisting force applied to a threaded fastener or part. It is measured in force units of foot-pounds (ft·lb) or the SI Metric equivalent, newton-meters (N·m). Torque is the product of the force applied times the length of the lever arm. (7)

total runout. A type of runout that provides a combined control of the circularity, straightness, angularity, and cylindricity of a surface rotated around a datum axis. (27)

traceability. The ability to validate a calibration by tracing the equipment and procedures used in the calibration back to the NIST standards. (26)

trammel. A scribing tool that consists of a long beam with two sliding scriber points. Trammels are used to scribe circles and arcs that are too large to be made with a divider. (6)

traverse grinding. A cylindrical grinding operation in which a fixed amount of material is removed from the rotating workpiece as it moves past the revolving grinding wheel. Work wider than the grinding wheel can be ground using this method. (19)

turning. A machining process that operates by rotating work against the edge of a cutting tool. (1)

turning center. A CNC turning machine equipped with an automatic tool changer. (21)

turret lathe. A lathe equipped with a rotating tower, or turret, that holds multiple tools and can be revolved to present the appropriate tool for a particular operation. Turret lathes can also be controlled by a CNC program. (15)

twist drill. A common drill 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. Finally, they are heat-treated and ground to exact size. (12)

U

ultrasonic-assist machining. A machining process in which ultrasound waves are used to enhance the function of a conventional machine tool. (32)

ultrasonic testing. A nondestructive testing method that uses high-frequency sound waves to detect flaws and cracks in materials. (26)

unassigned code. A code that can be assigned to different functions on different types of machine controllers for specialty purposes. (22)

Unified System. A thread form system adopted by NATO after World War II. (7)

unilateral tolerance. A tolerance that allows the dimension to vary by a specific amount in one direction only, either above or below the stated dimension or size. (4)

universal bevel protractor. An angle-measuring tool used in layout work. This tool consists of a dial, a base or stock, and a sliding blade. It is graduated into 360°. (5)

universal tool and cutter grinder. A grinding machine designed to support cutters (primarily milling cutters) while they are sharpened to specified tolerances. Common attachments permit straight, spiral, and helical cutters to be sharpened accurately. Other attachments are available to adapt the machine for all types of internal and external cylindrical grinding. (19)

US Conventional system. The system of weights and measures used in the United States. Compare *SI Metric system*. Also called *US Customary system*. (4)

V

V-block. A square or rectangular steel block with a 90° V-groove through the center, with a clamp to hold round stock in place for drilling, milling, and layout operations. (6)

ventilation. Circulation of air that brings in fresh air and lowers the concentration of toxic fumes within the working area. (3)

vernier caliper. A precision measuring instrument, used for both inside and outside measurements, that is accurate to 1/1000″ (0.001″) and 1/50 mm (0.02 mm). (5)

vernier protractor. An angle-measuring tool used in layout work when angles must be extremely accurate. With this tool, angles of 1/12 of a degree (5 minutes of arc) can be precisely measured. (6)

vernier scale. A sliding scale designed to move along a larger scale to specify measurements between the divisions or graduations on the larger scale. (5)

vertical machining center (VMC). A machining center with a vertically oriented spindle. (21)

vertical milling machine. A type of milling machine in which the cutter is normally perpendicular (at a right angle) to the worktable. On many vertical milling machines, the spindle can be tilted to perform angular cutting operations. (17)

vitreous enamel. A glass coating that can be fused to most metals, including steel sheet and cast iron surfaces. It forms an extremely hard coating that is smooth and easy to clean. (30)

W

waterjet cutting. A machining process that uses a high-velocity, high-pressure stream of water to cut through materials. Also called *hydrodynamic machining (HDM)*. (32)

waviness. The presence of smoothly rounded peaks and valleys caused by tool and machine vibration and chatter. Waviness is of greater magnitude than roughness. (30)

wavy set. A saw tooth set in which several teeth are angled to the right, followed by several to the left, alternating continuously along the blade. Wavy set is recommended for work with varying thicknesses, such as pipe, tubing, and structural materials. (20)

ways. The flat or V-shaped bearing surface that aligns and guides the movable part of a lathe. (14)

Webster hardness tester. A portable testing device for checking the hardness of materials such as aluminum, brass, copper, and mild steel. (29)

wheel dresser. A tool for cleaning, resharpening, and restoring the mechanical accuracy of the cutting faces of grinding wheels. (13)

word. In a word address format programming language, a combination of letters and numerical data. (22)

word address format. A style of programming that uses groups of letters and numbers, referred to as *words*, to program instructions for a CNC machine tool. (22)

work envelope. The volume of space defined by the reach of the robot arm in three-dimensional space. (25)

working drawing. A drawing or set of drawings that gives a machinist the necessary information to make and assemble a mechanism. (4)

X

X-Y coordinate system. A grid system that allows the locations of features on a drawing to be identified exactly using their horizontal (X) and vertical (Y) position relative to a point designated as 0,0. At point 0,0, both the X value and the Y value are equal to 0. (4)

Index

D

E

F

N

O

P

Q

R

S

T

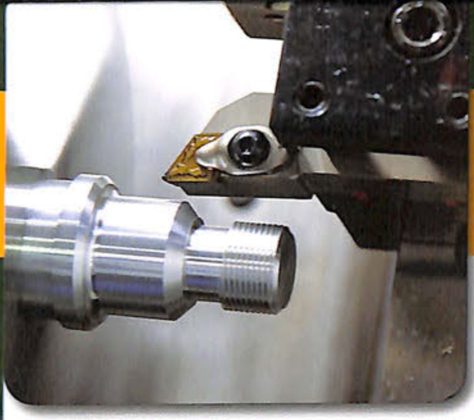
U

V

W

X

Z



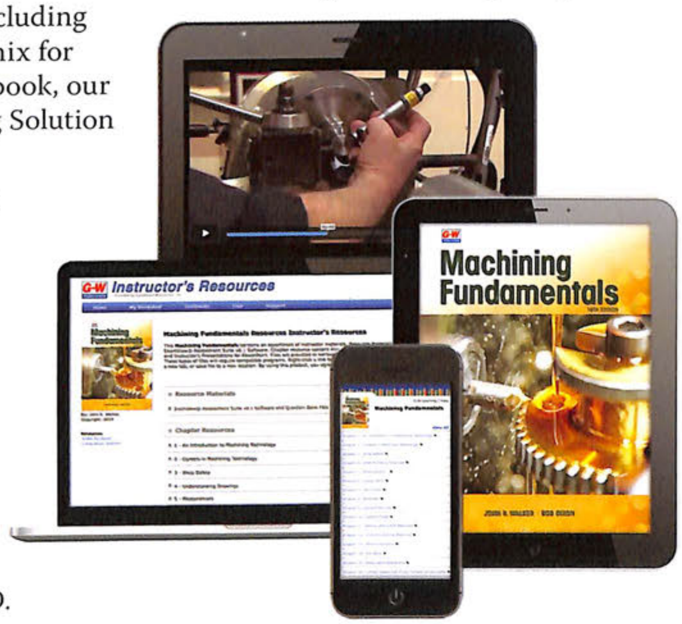
Machining Fundamentals

Machining Fundamentals provides a comprehensive introduction to the various machining operations, setups, and procedures. This colorful textbook covers all traditional machining methods, as well as newer and nontraditional methods. This edition includes expanded coverage of geometric dimensioning and tolerancing; features on careers, workplace skills, safety, and green machining practices; and updated illustrations.

- Strong emphasis on safety throughout the textbook.
- Career Connections, Workplace Skills, and Green Machining features engage, motivate, and prepare students for career success.
- Correlated to the duties and standards for NIMS Machining Skills Level I to help students achieve certification.
- The Online Learning Suite includes an online textbook, student workbook with digital form fields, vocabulary activities, drag-and-drop activities, and 30 video clips shot in an actual manufacturing facility and loaded with practical hands-on demonstrations.

G-W Integrated Learning Solution

At Goodheart-Willcox, we combine the technical knowledge of our authors, the guidance of our reviewers, and our publishing experience to create an Integrated Learning Solution: ***content and tools to help students and instructors achieve success***. Our products start with theory and applied content based on a strong foundation of accepted standards and curriculum. To that base, we add features and tools designed to help promote effective and efficient teaching and learning. We provide a variety of print, digital, and online formats, including bundles, allowing educators to select the right mix for their classroom. In addition to this printed textbook, our *Machining Fundamentals* Integrated Learning Solution offers the following products:



- **Workbook.** Provides minds-on practice with questions and activities.
- **Online Learning Suite.** Brings digital learning to the classroom by providing an online student text and online workbook, along with rich supplemental content including video clips, vocabulary activities, and drag-and-drop activities.
- **Instructor Resources.** Instructor Presentations for PowerPoint®, ExamView® Assessment Suite, Answer Keys, Lesson Plans, and more are available online or on CD.

ISBN 978-1-63563-208-8
90000
9 781635 632088

10TH EDITION

Machining Fundamentals

JOHN R. WALKER | BOB DIXON

G-W PUBLISHER