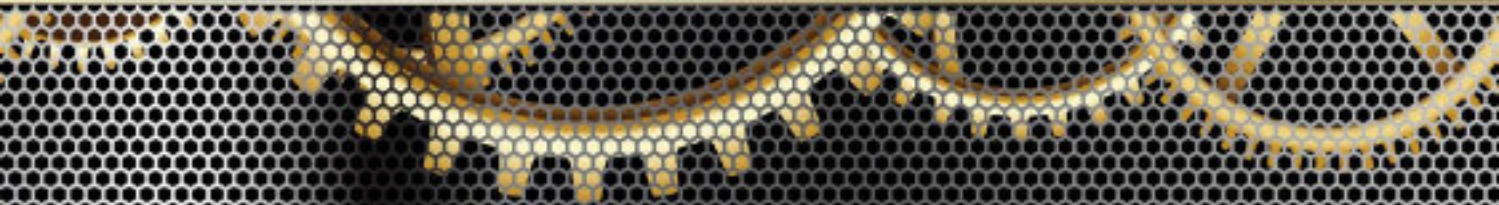


Volume 1



MACHINE SHOP PRACTICE



Machine Shop Practice, Volume 1



K.H. MOLTRECHT
Associate Professor of Industrial
and Systems Engineering
The Ohio State University
Columbus, Ohio

INDUSTRIAL PRESS INC.

200 Madison Avenue
New York, NY 10016

Library of Congress Cataloging in Publication Data

Moltrecht, Karl Hans.
Machine shop practice.

Includes index.

1. Machine-shop practice. I. Title.

TJ1160.M66 1979 670.42'3 79-91236

ISBN 0-8311-1126-7 (v. 1)

MACHINE SHOP PRACTICE

Volume 1

30 31 32 33 34 35 36

Copyright © 1981 by Industrial Press, Inc., New York, N.Y. Printed in the United States of America. All rights reserved. This book or parts thereof, may not be reproduced in any form without permission of the publishers.

Contents

	page
Preface	vii
Preface to First Edition	ix
List of Tables	xi
1. Basic Metal Cutting Operations	1
2. Basic Measuring Instruments	13
3. Layout Work	53
4. Drilling Machine, Twist Drills, and Auxiliary Tooling	75
5. Drilling Machine Operation	108
6. Engine Lathe Construction	139
7. Single-Point Cutting Tools and Their Performance	161
8. Turning on Lathe Centers	220
9. Chucking Work	252
10. Taper and Angle Turning	279
11. Faceplate Work	294
12. Screw Threads and Screw-Thread Measuring	311
13. Cutting Screw Threads on a Lathe	341
14. Turret Lathes, Production Lathes, and Vertical Lathes	382
15. Precision Hole Location—The Jig Borer	403
16. Metal Cutting Saws	459
Appendix	485
Index	491

Preface

The impact of new technology has made necessary a second edition of this book. It has been thoroughly revised while still providing complete and detailed descriptions of all the basic machine shop skills and the related techniques. It is written for the beginner, as well as for the more-advanced craftsman, technician, and manufacturing engineer. While great advancement in the numerical control of machine tools has been made (treated in Volume 2 of *Machine Shop Practice*), the actual machining operations of these machines are identical to those performed on the basic machine tools. Thus, an understanding of the basic machine tool operations and other shop procedures is essential at all levels of machine shop practice. In many shops metrication is now a fact of life; thus, SI metric units are introduced in this edition, without, however, sacrificing the application of the customary inch units which they usually follow in parentheses. Metric screw thread standards and the methods of cutting these threads have been added. New cutting tool materials and new cutting tool designs are discussed throughout both volumes. A unique feature found for the first time in this book is the method of estimating the cutting speed that will utilize the maximum power of a machine tool. Much additional and invaluable practical machining information has been added to the second edition, which will be found helpful to machinists, whether they are beginners or more-advanced practitioners.

I would like to take this opportunity to thank those readers who have responded with good criticism, suggestions, and occasionally with praise for the first edition. Their input has been carefully evaluated resulting in many of the changes in this edition. It is with much regret that space will not permit singling out by name all of those who have helped me in the preparation of this new edition. I would, however, like to pay a special tribute to Holbrook L. Horton of Industrial Press, Inc., for his valuable suggestions and his meticulous work in technically editing the two volumes of this edition. I must also express my appreciation for the encouragement and assistance provided by Paul B. Schubert. Clara Zwiebel and Maryanne Colas were most exacting copy editors with whom it was always a pleasure to work. Finally, I have a special word of gratitude for my wife, Wilma, whose untiring effort in typing the manuscript eased the burden of preparing this edition.

Preface to First Edition

A very large variety of machine tools is used in modern machine shop practice. Many of these machine tools contain sophisticated devices which are derived from the latest discoveries in science and technology. With the ever-increasing knowledge of science and technology, future machine tools promise to be even more complex to satisfy the demand for more productivity, lower-cost productivity, and greater precision. However, for all of their advantages, the more sophisticated machine tools of today have not replaced the need for the basic machine tools. The basic machine tools are still required to make precision parts for tools, dies, machines, instruments, and for all classes of machine tools.

The best starting point in learning the principles of the more complex machine tools is still to learn the fundamentals of operating the basic machine tools. This can be a stepping stone to a career on the management level in the metalworking industry or as a designer of tools, dies, and machines. It can also lead to a very rewarding career as a skilled craftsman, such as a general machinist, or as a tool and die maker.

This book is intended to treat the operation of the basic machine tools and the technology which supports these operations. The skilled craftsman must be able to operate machine tools intelligently. His ability to advance will, to a large measure, depend upon his ability to apply the supporting technology. The beginning reader may wish to bypass some of the more complicated topics and concentrate at first on the principles of machine-tool operation; however, he will have available for future reference much very valuable technical information on shop theory.

The teacher may likewise wish to bypass the more complicated topics in his course of study when using this book. The inclusion of more advanced topics in this book is an added dimension which can be used in the advanced vocational and technical courses. For the beginning courses, the procedures for operating the basic machine tools are treated in depth. The more advanced lessons can be used to show the extent of the technology of machine shop practice. They can also be used selectively as added assignments for the more ambitious student.

The treatment of the operation of the machine tools in this book makes it possible to use it in the shop. Since it is rarely possible to have all of the students operating the same kind of machine tool at one time, the reading assignments can be made on the basis of the machine tool

that the student is learning to operate. This book can also be used in the related course in which machine shop theory is taught. For this course the subject matter is developed in an orderly and progressive manner. The questions at the end of each chapter can be used for review, for homework assignments, or for examinations.

I would like to take this opportunity to acknowledge the debt of gratitude that I owe to the many persons who have made this work possible. First are the teachers who, in the classroom and in the industrial workshop, taught me the basic skills and principles treated in these volumes and who inspired me to continue my career in this field. It gives me a genuine sense of pleasure to include their names in this book:

Orlan W. Boston	Clarence J. Metzgar*
Lester V. Colwell	John H. Moltrecht*
William W. Gilbert	Karl Schatz
Herman Hermann	G. Harold Silvius
Frank Kiefer*	Robert E. Smith*
Richard O. Knight	William C. Truckenmiller
Robert E. McKee	Carl Wolff*

* *deceased*

The acknowledgments in the captions of the illustrations give some idea of the help offered by the machine tool, and allied industries in the preparation of this book. The generous help and cooperation of the persons and their employers who furnished this material is most sincerely appreciated. It is with deep regret that it is not possible to list here all of these individuals, and their firms, by name.

I must also express my appreciation to Holbrook Horton of the Industrial Press Inc. His suggestions and corrections contributed very materially to the clarity and accuracy of these volumes.

Lastly, I am much indebted to my wife, Wilma, for her constant help and encouragement. To her I tender my very best thanks for deciphering the original handwritten manuscript and for toiling endless hours at the typewriter in the preparation of this book.

List of Tables

2-1. Accuracy Standards for Precision Gage Blocks	36
2-2. Sizes for an 83-piece Gage Block Set	38
2-3. Inch-Millimeter and Inch-Centimeter Conversion Table	44
4-1. Oversize Amount Normally Cut by a Drill Under Normal Shop Conditions, in Inches	84
4-2. Suggested Lip Relief Angles at the Periphery	85
4-3. Work Material Factor, K_d , for Drilling with a Sharp Drill	103
4-4. Feed Factors, F_f , for Drilling	103
4-5. Drill Diameter Factors: F_T for Thrust; F_M for Torque	104
4-6. Chisel Edge Factors for Torque and Thrust	104
4-7. Tool Wear Factors, W	106
4-8. Machine Tool Efficiency Factor, E	106
5-1. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Plain Carbon and Alloy Steels	111-112
5-2. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Tool Steels	113
5-3. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Stainless Steels	114
5-4. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Ferrous Cast Metals	115
5-5. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Light Metals and Copper Alloys	116
5-6. Recommended Feeds in Inches per Revolution for High-Speed Steel Twist Drills	116
7-1. Recommended Cutting Speeds in Feet per Minute for Turning Plain Carbon and Alloy Steels	163
7-2. Recommended Cutting Speeds in Feet per Minute for Turning Tool Steels	165
7-3. Recommended Cutting Speeds in Feet per Minute for Turning Stainless Steels	166
7-4. Recommended Cutting Speeds in Feet per Minute for Turning Ferrous Cast Metals	167
7-5. Recommended Cutting Speed, in Feet per Minute, for Turning Nonferrous Metals and Alloys	168
7-6. Cutting Speed Feed and Depth of Cut Factors for Turning	170

7-7. Chemical Composition of Frequently Used High-Speed Steels	170
7-8. Recommended Rake Angles	181
7-9. Power Constants, K_p , for Wrought Steels, Using Sharp Cutting Tools	215
7-10. Power Constants, K_p , for Ferrous Cast Metals, Using Sharp Cutting Tools	216
7-11. Power Constant, K_p , for High Temperature Alloys, Tool Steel, Stainless Steel, and Nonferrous Metals, Using Sharp Cutting Tools	216
7-12. Feed Factor, C , for Power Constants	217
8-1. Guide for Selecting Feed Rates	228
12-1. Coarse-Thread Series, UNC and NC—Basic Dimensions	315
12-2. Fine-Thread Series, UNF and NF—Basic Dimensions	316
12-3. ISO Metric Thread Sizes, to 45 mm	324
12-4. Thread Sizes Specified by ANSI, for Screws, Bolts, and Nuts, with Comparison to Closest Unified Inch Size	325
16-1. Recommended Saw-blade Pitch, Cutting Speed, and Feed for Power Hacksaw Operation	461
16-2. Recommended Cutting Speeds for Band Sawing	466
16-3. Basic Dimensions for Making Punch and Die, by Method Shown in Fig. 16-16	476
A-1. ANSI Standard Combined Drills and Countersinks, Plain and Bell Types (ANSI B94.11-1967, R1972)	485
A-2. ANSI Drill Sizes, Their Inch and Millimeter Equivalents, and Millimeter Size Drills for Which They Can Be Used	486-487
A-3. Some Frequently Used Conversion Factors	488
A-4. Decimal Equivalents	489
A-5. Cutting Speeds for Machine Tapping	490

Basic Metal Cutting Operations

This treatise deals with the principles, tools, and processes which are utilized in all types of machine shops or machine building plants. The information is largely fundamental in character, and the various tools and processes described are applicable in making many different classes of machines and other mechanical products. While machine building, in a broad sense, embraces every craft and process from the making of patterns, castings, forgings, stampings, etc., to the exact procedure in the finishing and assembling of parts, this course of instruction applies principally to the work of the machine shop—because of the general demand for such information—and its usefulness in technical schools and in the industrial training departments of manufacturing plants. The application of different types of machine tools and machining operations is explained. It will be noted that many of the operations described can be performed in the same, or very similar manner, on different machine tools. For example, boring is done on a lathe, a horizontal boring machine, a drilling machine, a milling machine, and a jig borer. An awareness of these similarities is very helpful in mastering the skills and the related knowledge required to operate machine tools.

The Work of the Machine Shop

The work of the machine shop in building machines is to make each part to the required shape and size. The unique feature of all machine shop operations is that the parts are precisely formed to the desired shape and size by removing metal from the workpiece in the form of chips. These metal chips may be very large, or, as in the case of grinding operations, very small. Assembling of the finished parts is, of course, the final job. The material which is to be machined is generally in the form of hot-rolled bars, cold-drawn bars, centerless ground or machined bars, castings, or forgings. Tools, dies, machine parts, and engine parts frequently must have very precise dimensions and smooth surface finishes. These characteristics can sometimes be obtained by other manufacturing methods, but they can usually be achieved more economically, and to a higher degree, by machining. Value is added to the raw materials that are processed to become machine parts in the machine shop, by virtue of the work done to change their shape, provide precise dimensions, and create their final, smooth surface finishes.

Machine Tools

The general term "machine tool," is applied to various classes of power-driven metal cutting machines such as are used in machine shops. Most machine tools change the shape of a material by producing chips; however, the National Machine Tool Builders' Association includes power-driven presses in the category of machine tools even though they do not produce chips. In these two volumes, however, power-driven presses, such as are used in the stamping and forging industry, will not be considered since, generally, they are not used in the average machine shop.

Machine tools can be considered to serve four main purposes:

1. They hold the work or the part to be cut.
2. They hold the cutting tool or tools.
3. They impart to the cutting tool or work—or both—whatever motion is required for cutting or forming the part.
4. They are arranged, ordinarily, for regulating the cutting speed and also the feeding movement between the tool and the work.

In the production of machine parts of various shapes and sizes, the type of machine and cutting tool used will depend upon the nature of the metal-cutting operation, the character of the work, and, possibly, other factors such as the number of parts required and the degree of accuracy to which the part must be made. The development of machine tools has been largely an evolutionary process, as they have been designed to produce parts meeting increasingly stringent mechanical standards. Developments in power transmission, accuracy, and control of the movements and functions of the machine are constantly being incorporated into the design of new machine tools.

Machine tools are built to produce certain shapes such as cylindrical and conical surfaces, holes, plane surfaces, irregular contours, gear teeth, etc. Many machines, however, can produce a variety of surfaces. Thus, machine tools are built as general purpose machines, high production machines, and as special purpose machines. As the name implies, general purpose machine tools are designed to be quickly and easily adapted to a large variety of operations on many different kinds of parts. They are used extensively in jobbing shops, repair shops, and in tool and die shops. Sometimes these machine tools are called job shop machine tools. Production machine tools are designed to perform an operation, or a sequence of operations, in a repetitive manner in order to achieve a rapid output of machined parts at minimum cost. They can be set up to machine a variety of different parts; however, their operation is economical only when the quantity of parts to be machined is relatively large. Special purpose machine tools are designed to perform one operation, or a sequence of operations, repetitively, on a specific part. These machines are usually automatic and are unattended except when it is necessary to change and to adjust the cutting tools. They are used in mass-production

shops such as are found in the automotive industry. Numerically controlled machine tools have their functions programmed on a punched tape. The tape may be made on a tape writer in accordance with a program written to produce the required part or it may be made from a program determined by a computer. Numerically controlled machine tools achieve a rapid and economical output of machine parts when the quantities to be produced are low or medium. They cannot cut metal any faster than can other machine tools; however, they reduce the amount of "cut and try" by the machine operator thereby increasing the output of parts. They are particularly useful where repeat orders for the same part are encountered. The tape containing the information required to produce the part can be stored conveniently and reused when required.

Metal Cutting Tools

The metal cutting tool separates chips from the workpiece in order to cut the part to the desired shape and size. There is a great variety of metal cutting tools each of which is designed to perform a particular job or a group of metal cutting operations in an efficient manner. For example, a twist drill is designed to drill a hole having a particular size, while a turning tool might be used to turn a variety of cylindrical shapes. In order to sever a chip from the workpiece the following conditions must be present:

1. The tool is harder than the metal to be cut.
2. The tool is so shaped that its cutting edge can penetrate the work.
3. The tool is strong and rigid enough to resist the cutting forces.
4. There must be movement (provided by the machine tool) of the tool relative to the material in order to make the cutting action possible.

Modern metal cutting tools are made from tool steels, powdered metals, ceramics, and industrial diamonds. These materials can be made to be very hard and they can retain their hardness at the high temperatures resulting from the metal cutting action. All metal cutting tools wear as the result of stresses and temperatures encountered in separating the chips. The rate-of-wear must be controlled by the application of the correct cutting speed and feed. After wear has progressed to certain limits the cutting edge may be resharpened by grinding. Ultimately, further sharpening is not practical and the tool must be discarded. For this reason metal cutting tools are classified in industry as perishable tools.

There are three basic types of metal cutting tools: single-point tools, multiple-point tools, and abrasives. These names are quite descriptive. A single-point metal cutting tool has a single cutting edge and is used for turning, boring, shaping, and planing. Multiple-point tools have two or more cutting edges such as drills, reamers, and milling cutters. Grinding wheels are an example of abrasive cutting tools. Each grinding wheel has thousands of embedded abrasive particles which are capable of pen-

etrating the workpiece and removing a tiny chip. The combined total of the tiny grinding chips can result in a substantial amount of metal being removed from the workpiece.

Principles of Turning

The diagram at the left in Fig. 1-1 illustrates how metal is cut by turning, as in a lathe. The turning tool is held rigidly in some form of holder. The part to be turned rotates in direction R , and the tool feeds along continuously in direction F , parallel to the axis of the part being turned. Consequently, the part is reduced from some diameter D , to some other diameter d , depending upon the requirements in each case. The upper surface of the tool, against which the chip bears as it is being severed, usually is ground to some rake angle α , to make the edge keener and to lessen cutting resistance. (There are exceptions to this rule as explained later in Chapter 7 which covers Single-Point Tool Forms and Tool Grinding.) This top or chip-bearing surface generally slopes away from that part or section of the cutting edge which normally does the cutting; hence, usually there is both backward and side slope or, in tool parlance, a back rake angle and a side rake angle. (The angle, α , of the diagram is

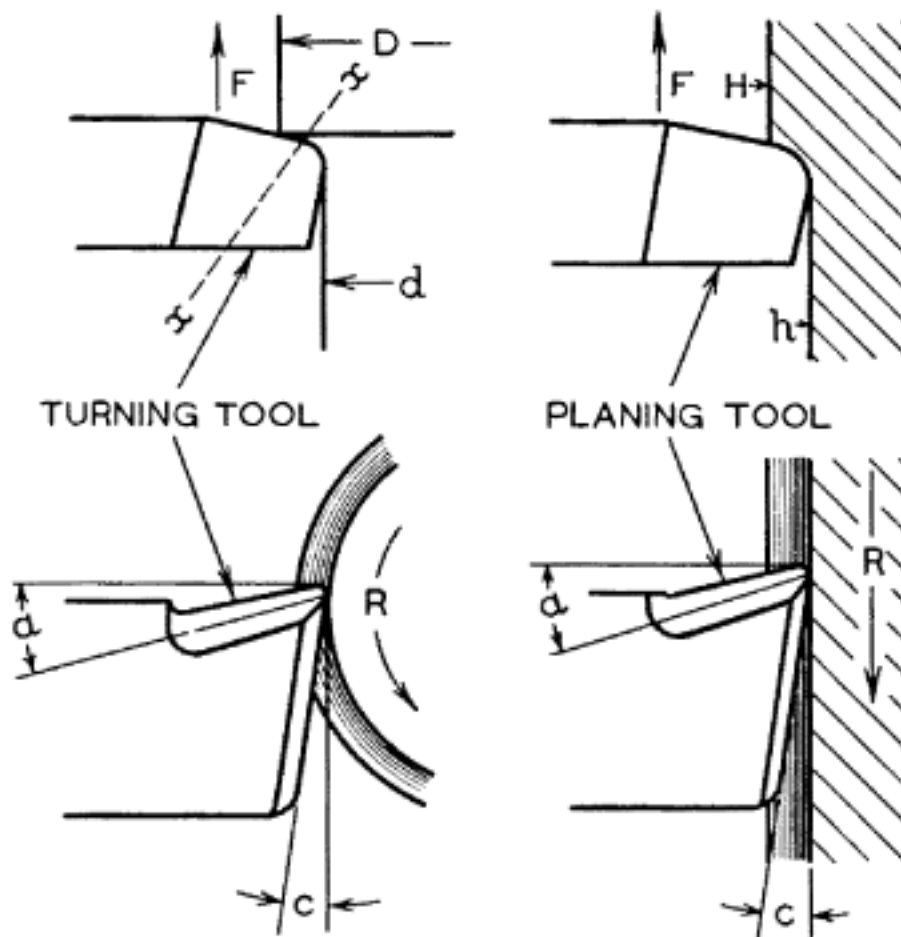


Fig. 1-1. Principles of turning and planing.

intended to represent the combined back and side angles or what is known as the compound angle in some plane $x-x$.) If this inclination α , is excessive, the tool point will be weakened. On the other hand, if there is no inclination, the tool will still remove metal, but not as effectively as a tool with rake, especially in cutting iron and steel. Below the cutting edge, some relief angle c , is essential so that this end or side surface of the tool will not rub against the work and prevent the cutting edge from entering it freely, particularly when the tool is adjusted inward for taking a deeper cut. This relief is not confined to the end alone but is also required along the side extending as far back as the length of the cutting edge. The factors governing the amount of rake angle α , relief angle c , and the general shape of the cutting edge will be considered later.

Principles of Planing and Shaping

A comparison of the turning-tool diagrams (Fig. 1-1, left) with the planing tool (Fig. 1-1, right) will show that planing a flat surface is similar in principle to turning a cylindrical surface, the chief difference being that a straight-line movement is required for planing and a rotary or circular motion is used for turning. (The planing tool, when planing a horizontal surface, is held in a vertical position. It is shown in a horizontal position to obtain a more direct comparison with the turning tool.) The planing tool is held stationary while cutting and the part to be planed is given a straight-line motion R , during the cutting stroke, followed by an intermittent, lateral tool-feeding movement in direction F , during the return or non-cutting stroke. (If a shaper, which is a small type of planing machine, is used, this order is reversed; the tool moves, while the work remains stationary.) As the tool is moved laterally, prior to each cutting stroke, successive strokes reduce the surface of the work from H to h , the amount depending upon the requirements.

In planing, as in turning, the tool surface against which the chip bears usually slopes backward or has a certain rake angle α , to reduce the cutting resistance. Some relief angle c , in back of the cutting edge, is necessary to permit this edge to cut freely. If either angle α or c is excessive, the point of the tool will be weakened and it may not be strong enough to resist the pressure required for cutting iron or steel. This is true of turning, planing, and other classes of metal cutting tools. The shapes and angles of planing tools are dealt with in Volume 2, in the chapters on Shapers and Planers.

Principles of Milling

The milling cutter represented by the upper left diagram, Fig. 1-2, has equally spaced teeth around its circumference so that as many cuts are taken per cutter revolution as there are teeth, and the cutting action is continuous. The cutter is rotated by the spindle of the milling machine and the part to be milled is given a feeding movement, generally in direction F , or *against* the cutter rotation R . (Sometimes the feeding movement is in the opposite direction, or *with* rotation R .) As the work feeds

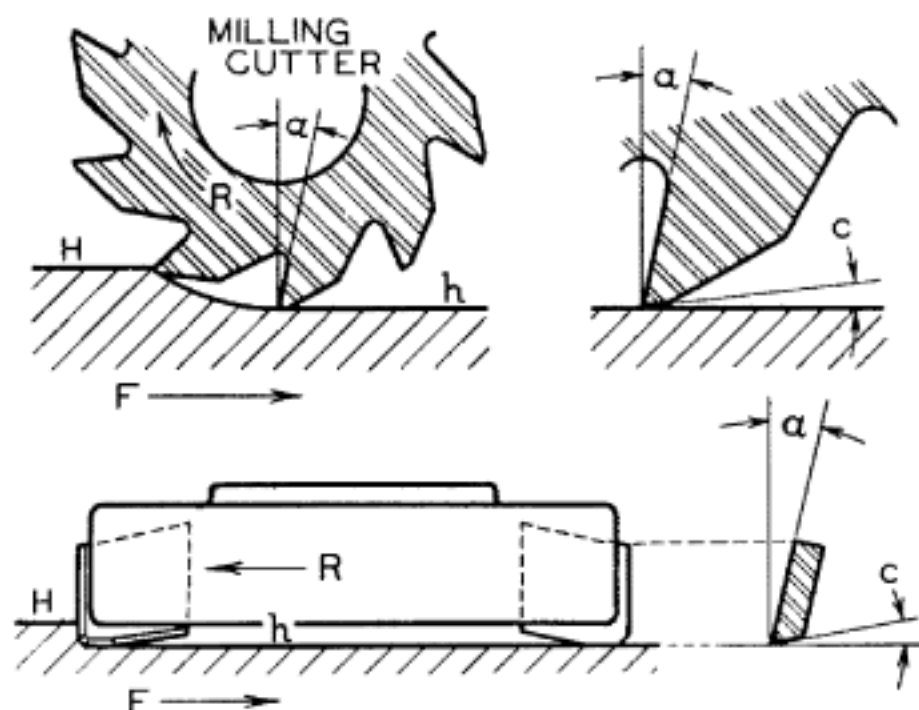


Fig. 1-2. Principles of plain milling and face milling.

past the revolving cutter, the surface H , is reduced to some height h , as may be required.

The milled surface will be flat if the cutter is cylindrical, but, frequently, cutters of other forms are used, as explained later in Chapter 6, Volume 2, the section on Milling Machine Operations. The chip-bearing surfaces of the milling cutter teeth may have some rake angle α , to lessen cutting resistance and they must have relief c , to permit free cutting action. Thus we see that each tooth depends for its cutting action upon a relief angle and usually a rake angle also, just as with the turning and planing tools. (The actual relief angle c , applies to a narrow top edge on each tooth as indicated by the enlarged view, at upper right, of one tooth. Normally, the cutter is sharpened by grinding this narrow edge or "land" as it is called.)

The lower diagram in Fig. 1-2 illustrates the action of a face milling cutter. This general type of cutter is designed especially for milling flat surfaces. To simplify the diagram, a cutter is shown with two cutting blades only. In actual practice, however, there would be a number of blades equally spaced around the circumference of the cutter body to obtain more continuous cutting action. As the cutter rotates in direction R , the work feeds past it as indicated by arrow F , so that each blade, as it sweeps around, reduces the surface from some height H , to h . The feeding movement of the work is not always in a straight line but may be rotary. This, however, does not affect the operating principle. Each blade is held in a cutter body at an angle as indicated by the detail view to the right. This inclined position gives each blade a certain rake angle, α . The lower

end of each blade also has a relief angle c . The face milling cutter is another illustration of the fact that metal cutting operations with different types and designs of tools are all based upon the same fundamental principles.

Principles of Boring

The machining of internal cylindrical surfaces, when a single-point tool is used, is called "boring." Figure 1-3 illustrates the process. The part to be bored rotates in direction R , while the boring tool, with its top surface located in approximately the same plane as the center of the hole, feeds in direction F , or parallel to the axis of the hole, for cylindrical boring. As the tool advances, it increases the diameter of the hole, or bore, from d to D . The cutting tool, when applied to iron or steel, normally would

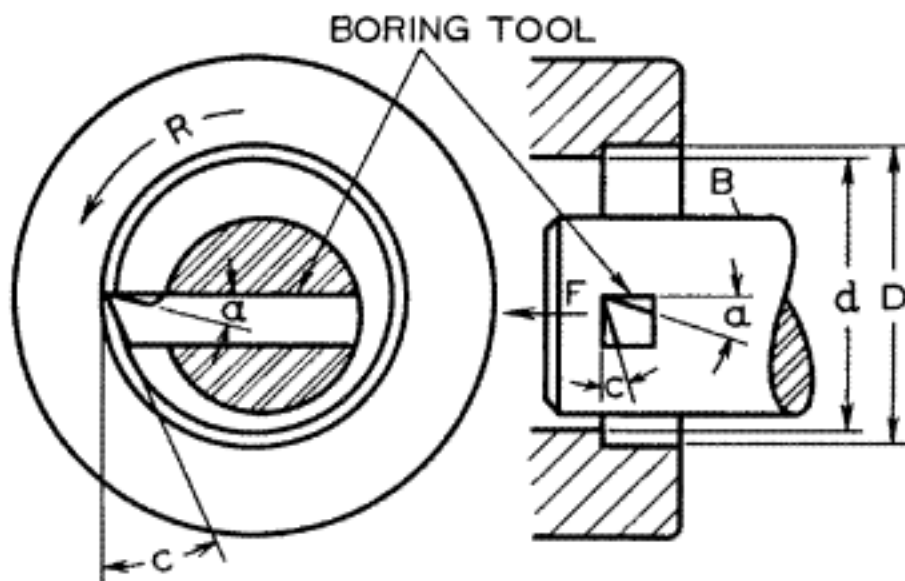


Fig. 1-3. Principles of enlarging or boring a hole.

have some slope or rake angle a on the chip-bearing surface, and a relief angle c , is essential. Note that there is both backward and side slope (or rake), and also relief at both the extreme end and along the leading side, or below all of the cutting edge. A somewhat greater relief angle may be required for boring than for turning—especially if the hole is small—but the cutting action is based upon the same fundamental principle in each case. A boring tool, of the general type shown in the diagram, is merely a small turning tool supported by a bar B , which, in turn, is rigidly supported at its outer end, and is long enough to permit boring a hole as deeply as may be required.

Principles of Drilling

When a hole is drilled by using a twist drill, the cutting point, or end, is ground to provide a certain amount of relief c , in back of each cutting

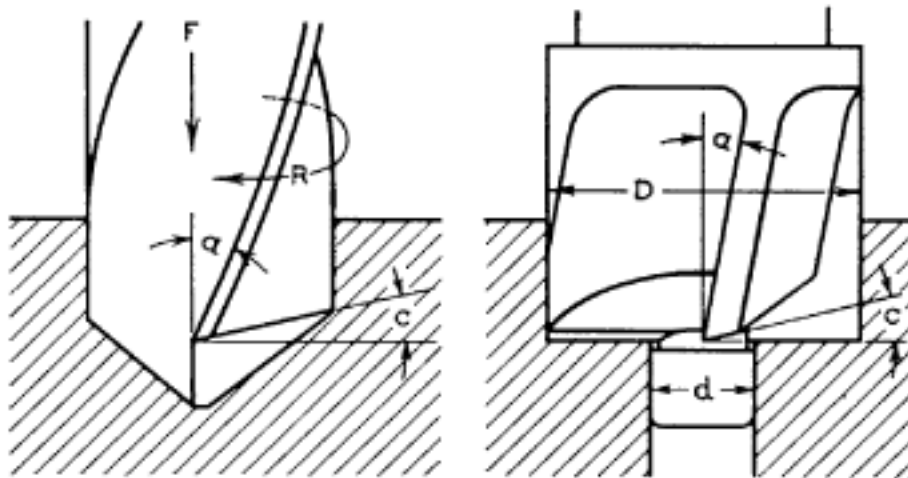


Fig. 1-4. (Left) Cutting end of twist drill. (Right) Counterbore for enlarging a previously drilled hole.

edge (left, Fig. 1-4) and extending along the entire length of the two edges. The twisting, or helical-shaped flute, or groove through which the chips pass also provides rake angle α above each cutting edge. This rake, or backward slope, is away from the cutting edge and is also applied to the chip-bearing surface as in the case of the other tools. It will be understood that the drilling of a hole is done either by feeding a rotating drill in direction F , and into stationary work as when using a drilling machine, or by rotating the work itself instead of the drill as, for example, when using a lathe or turret lathe for drilling.

The diagram, Fig. 1-4, right, illustrates a counterbore. This is a tool for enlarging part of a hole from some diameter d , to D . A pilot, or extension, fits into the smaller hole and insures cutting the enlarged part concentric with it. Again we have on the counterbore, relief angle c , at the lower end of each cutting tooth, and also rake angles α . While the counterbore and the twist drill are in appearance unlike a milling cutter or turning tool, all of these tools conform to the same fundamental operating principles.

Principles of Broaching

A broaching tool (Fig. 1-5) has a series of teeth so arranged that they cut the metal when the broach is given a straight-line movement, as indicated by arrow R . (In some cases the workpiece is moved instead of the broach, but the operating principle is the same.) The broach cuts away the metal since its teeth progressively increase in height, as shown on the exaggerated scale in the lower diagram. Thus, cutting action is distributed over a series of teeth and each tooth takes a light cut.

Each tooth is similar to a planing tool, and the cutting of metal in broaching is based upon the same fundamental principles as in planing, turning, or milling. The rake angle α , as in the case of other metal cutting tools, improves the cutting action. The top, or land, of each tooth has a

clearance, or relief angle c . When the broach is initially ground to the size and shape required, a narrow part of this land is often ground without any relief so that precise measurements may be taken on this surface. Later, the relief angle is finish ground, leaving a very small portion of the land behind the cutting edge without a relief angle in order not to destroy the size and shape of the broach. This straight section of the land sometimes helps to stabilize the broach when it is cutting and it provides a slight burnishing or finishing action on the surface of the workpiece. This lack of clearance, or relief, on broach teeth may seem opposed to the principle that relief is an essential requirement; however, the width of the straight portion is usually less than .030 inch. It does not interfere

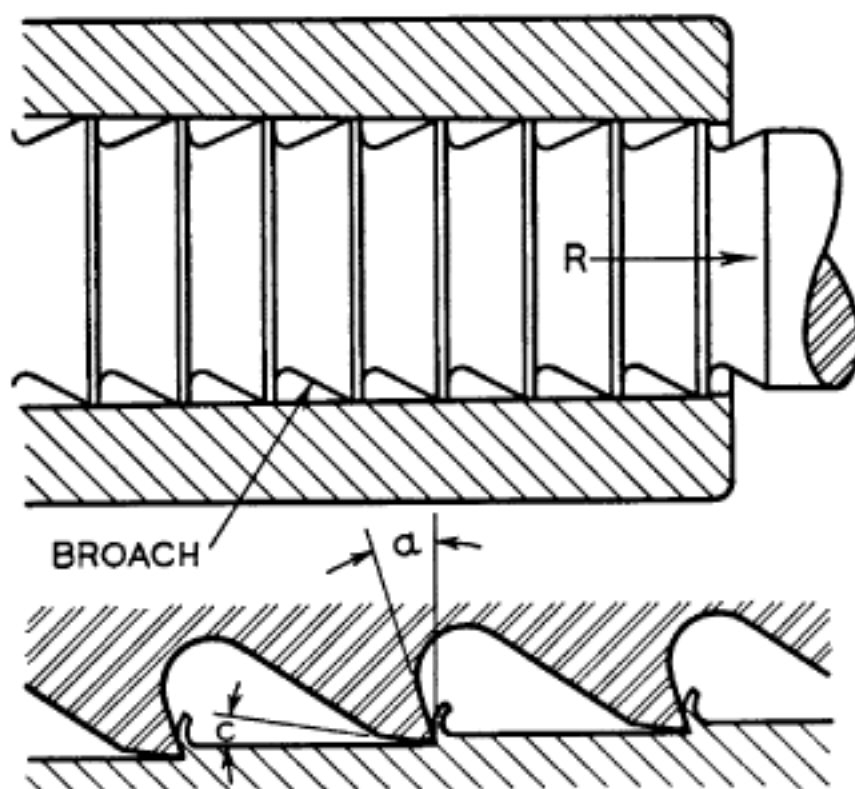


Fig. 1-5. (Above) Cutting action of a broaching tool. (Below) Enlarged scale drawing of broach teeth showing height increases.

with the cutting action because each tooth removes only a small amount of metal and the cutting edges of the broach are never adjusted inward or sunk more deeply into the metal while cutting.

The diagrams show that the cutting action of the broach follows the same principle as the turning tool, planing tool, milling cutter, and other metal cutting tools. Thus, it can be seen that the basic requirements of tool shapes are not complicated. In actual shop practice, however, it is essential to use tools which more or less vary in shape to suit different cutting conditions. The different factors to be taken into account will be referred to later.



Courtesy of Cincinnati Milacron

Fig. 1-6. Principle of cylindrical grinding, "traverse" method.

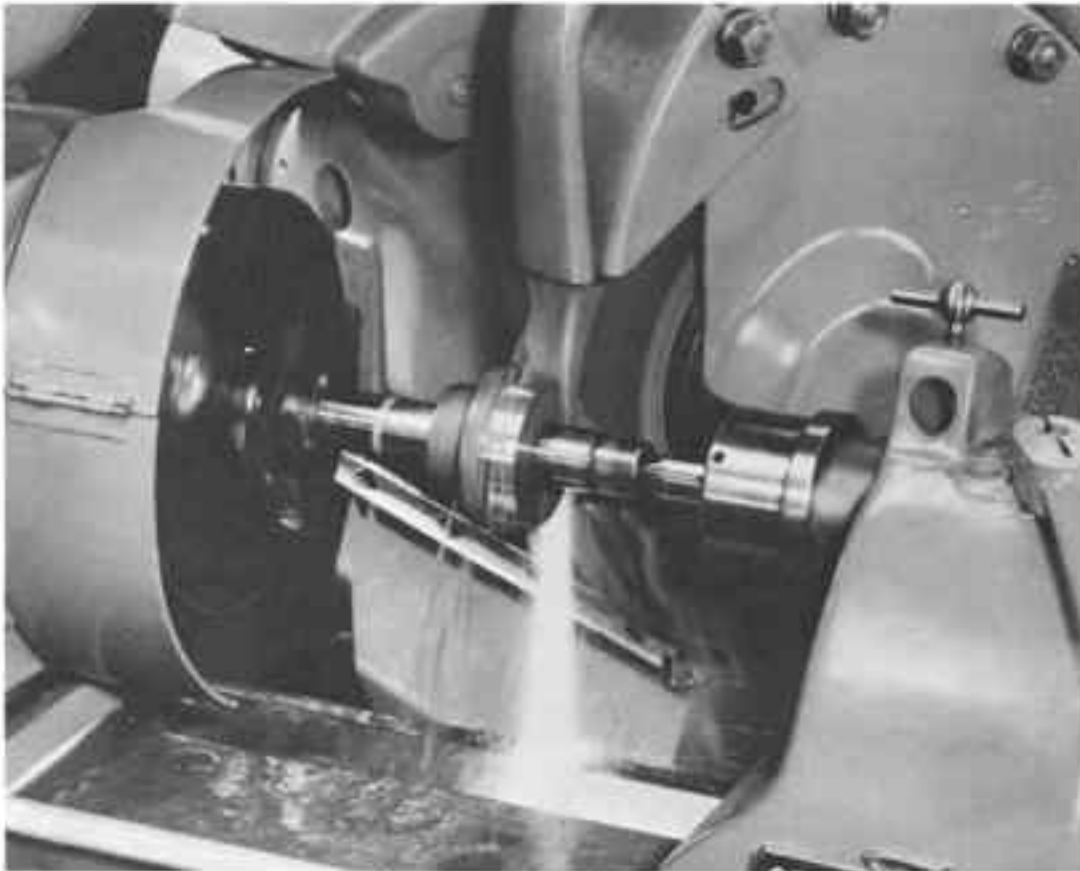
Precision Grinding

Grinding wheels are just as much metal cutting tools as are single- or multiple-point cutting tools which are used to alter the shape of a workpiece. Some grinding wheels are used to rough grind castings or steel bar stock in foundries or steel mills. These wheels must be strong enough to withstand rough usage. In the machine shop, however, grinding wheels are usually mounted on the spindles of precision machine tools in order to machine parts to close tolerances and to produce fine surface finishes. Grinding wheels are also used in the machine shop to grind cutting tools on pedestal grinders or on special cutter- and tool-grinding machines. Precision grinding machines are designed to control the path followed by the work and the wheel, with great precision. These machines thus have the capability of producing parts with a high degree of dimensional accuracy.

This capability is further enhanced by the nature of the cutting action of the grinding wheel which produces a multitude of very small chips.

These small chips permit very small amounts of metal to be removed from the surface of the work, thus favoring both dimensional accuracy and surface finish. Another important characteristic of the grinding wheel is that it will readily penetrate hardened metals, such as hardened tool steel, enabling these materials to be machined economically on precision grinding machines. Some heavy-duty surface-grinding machines have been designed to achieve rather high, metal removal rates. This process is called abrasive machining.

Cylindrical grinding machines have been designed to produce ground cylindrical, tapered, and formed surfaces. When traverse grinding (Fig. 1-6), the rotating workpiece is traversed past the grinding wheel which is rotating at a very high speed. At the end of each traverse stroke there is a dwell before the direction of the traverse is reversed. The purpose of the dwell is to give the grinding wheel time to cut at the end of the stroke. Generally, the dwell is also utilized to feed the grinding wheel into the workpiece for the succeeding traverse. This feed is called "infeed." Another method of cylindrical grinding is called "plunge grinding" (Fig. 1-7). This method is limited, however, to surfaces on the workpiece that are no longer than the width of the grinding wheel. The grinding wheel is fed, or plunged, into the rotating workpiece without any traverse motion.



Courtesy of Cincinnati Milacron

Fig. 1-7. "Plunge grinding" method of cylindrical grinding principle.

Surface grinding is accomplished by traversing a non-rotating workpiece, usually in a horizontal position, past the grinding wheel as shown in Fig. 1-8. The grinding wheel is generally fed a small amount perpendicular (crosswise) to the direction of the traverse movement of the work



Courtesy of the Galmeyer & Livingston Company

Fig. 1-8. Principles of surface grinding.

at the end of each traverse stroke. The surface resulting from this action is a plane, or flat surface. Special methods are used to grind profiled surfaces such as threads and gear teeth.

Basic Measuring Instruments

Measuring tools are used in machine construction, tool and die making, and machine tool operation in order to secure the required sizes and degree of accuracy of the parts. There are many different types and designs of measuring tools, some of which are very sophisticated. This section will deal with the measuring tools normally used by machinists and toolmakers in the everyday performance of their trade. A representative sample of these tools is shown in Fig. 2-1.

All precision measuring tools must be handled with care in order to avoid the slightest damage to them. Often measurements accurate to one-thousandth (.001) inch, or even to one ten-thousandth (.0001) inch, must be made. Careless handling of precision measuring tools will destroy their accuracy, and hence their usefulness, in making exact measurements. For example, a square that is not a true square will result in spoiled work. Precision measuring tools, therefore, must frequently be checked in order to verify their accuracy. With many of these tools the workman must depend upon the sense of feel in order to make accurate measurements. This requires that they be correctly grasped by the hands in order to utilize the sensitive nerves that are located in the tips of the fingers. The proper manipulation of precision tools is shown in many of the illustrations in this chapter.

The Precision Steel Rule

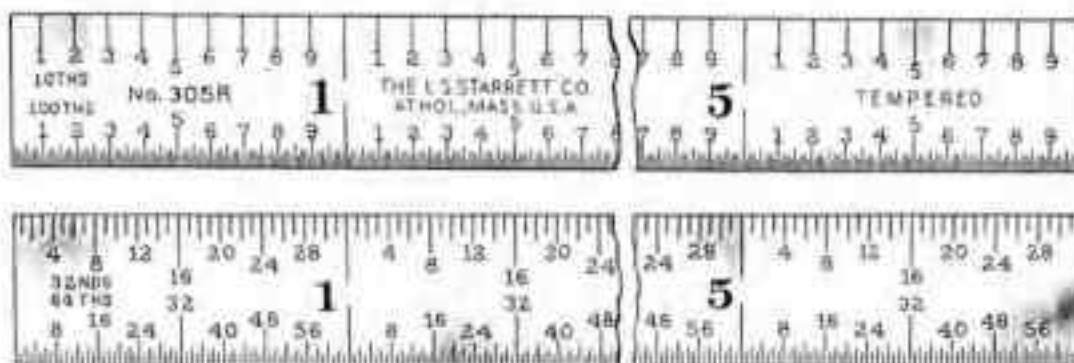
The rule is a basic length-measuring tool. It may be used directly in measuring a length; or, it may be used indirectly as when the diameter of a cylindrical object is measured with outside calipers and this measurement is then transferred from the calipers to the rule. Only precision-made, hardened steel rules are acceptable for use in the machine shop or in the tool and die shop. The lines or graduations on these rules are machine divided on specially designed, and very precise, graduating machines. Precision steel rules may be obtained as spring temper, semi-spring temper, semi-flexible, or full flexible. They range in size from one-quarter inch in length for measuring in grooves, recesses, and key-seats to twelve feet in length for large work. The most frequently used lengths are the six-inch and twelve-inch rules.

Two systems of graduations (Fig. 2-2) are used in the United States on steel rules. They are the decimal inch system and the fractional inch



Courtesy of The L. S. Starrett Company

Fig. 2-1. Basic machinists' measuring tools including a precision surface-plate.



Courtesy of The L. S. Starrett Company

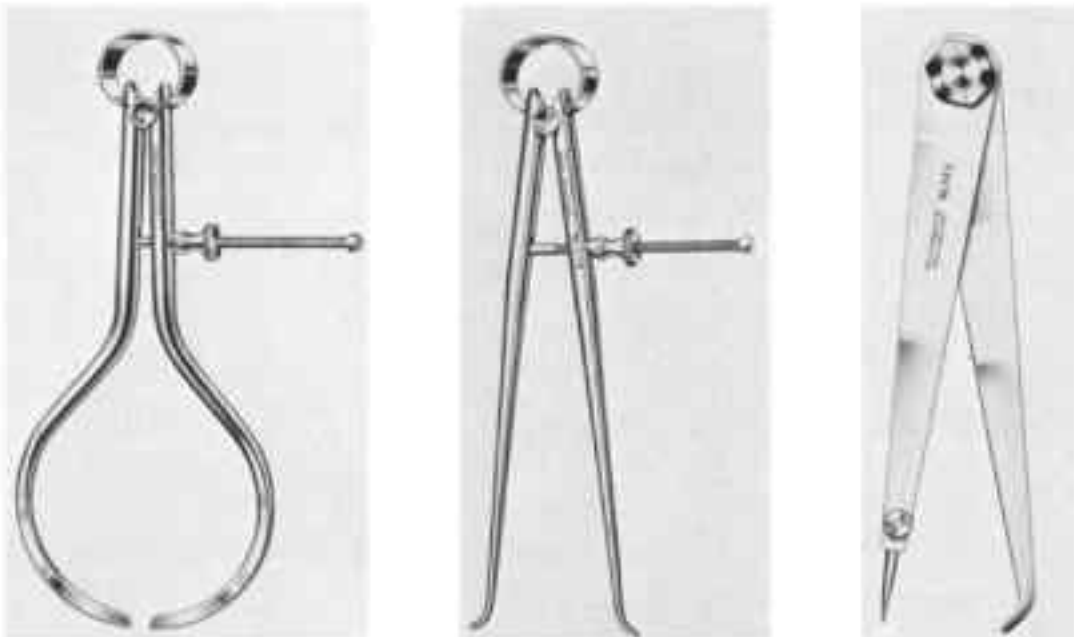
Fig. 2-2. (Upper view) Steel rule with decimal inch graduations; (Lower view) Steel rule with fractional inch graduations.

system. In the decimal inch system, the inch is divided into one tenth (.100) inch, one-fiftieth (.020) inch, and one-hundredth (.010) inch. (See upper view, Fig. 2-2.) In the fractional inch system (lower view, Fig. 2-2), the inch is successively divided by two, yielding the following graduations: $\frac{1}{2}$ or .500 inch, $\frac{1}{4}$ or .250 inch, $\frac{1}{8}$ or .125 inch, $\frac{1}{16}$ or .0625 inch, $\frac{1}{32}$ or .03125 inch, and $\frac{1}{64}$ or .015625 inch. One advantage of the fractional inch system is that the smallest graduation that can be clearly distinguished with normal eyesight, and without causing undue eyestrain, is $\frac{1}{64}$ or .0156 inch as compared to $\frac{1}{32}$ or .03125 inch for the decimal inch system.

Decimal equivalents are the decimals corresponding to the fractions in the fractional inch system. The decimal equivalent can be calculated from the fraction by dividing the denominator into the numerator. For example, to find the decimal equivalent of $\frac{7}{32}$ inch, divide 7 by 32, thus $7 \div 32 = .21875$ inch. Tables of decimal equivalents are usually available, making these calculations unnecessary. Their use is recommended to prevent possible mistakes that sometimes are made when decimal equivalents are found by calculation. A table of decimal equivalents is given in the Appendix of this text.

Calipers

There are three basic kinds of calipers: 1. Outside calipers, used primarily to measure outside diameters; 2. Inside calipers, used primarily



Courtesy of The L. S. Starrett Company

Fig. 2-3. (Left) Outside calipers; (Center) Inside calipers; (Right) Hermaphrodite caliper.

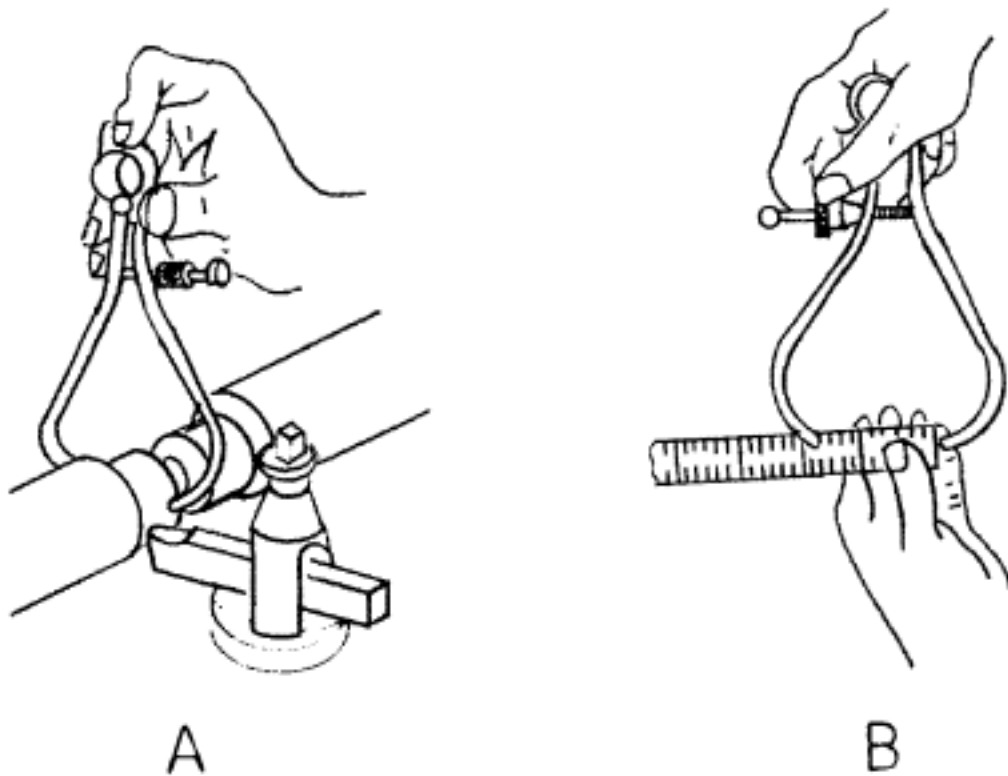


Fig. 2-4. Correct method of using outside calipers.

to measure hole diameters; and 3. Hermaphrodite calipers, used to measure the distance between a surface and a scribed line or to scribe a line from a surface. Two types of construction, firm joint and spring, are used in calipers. Firm joint calipers utilize the friction between the legs to maintain their setting. A firm joint hermaphrodite caliper is shown in Fig. 2-3. Spring calipers have a curved spring at the upper, or pivot, end which forces the caliper legs against the adjusting screw nut, thereby maintaining the caliper adjustment.

Accurate measurements are obtained with outside calipers by holding them as shown at A, in Fig. 2-4, and passing them over the workpiece with a very light contact pressure that can just be felt by the fingertips. The caliper reading is then transferred to a rule (Fig. 2-4, B). Accurate caliper readings, however, cannot be made when the work is rotating. In measuring a hole, inside calipers are adjusted to the correct diameter of the hole (see A, Fig. 2-5) by holding one caliper leg against the side of the hole with a finger of one hand, while the caliper, held in the other hand, is carefully passed through the hole. Moving the free leg back and forth in the X direction (A, Fig. 2-5), while slowly passing through the hole in the Y direction, will assist in finding the smallest true diameter of the hole through which the caliper legs can pass. Again, a very light feel will determine this setting. The inside caliper measurement is then transferred to a steel rule (B, Fig. 2-5) or to an outside micrometer caliper (D, Fig. 2-5). When done with great care, measurements accurate to

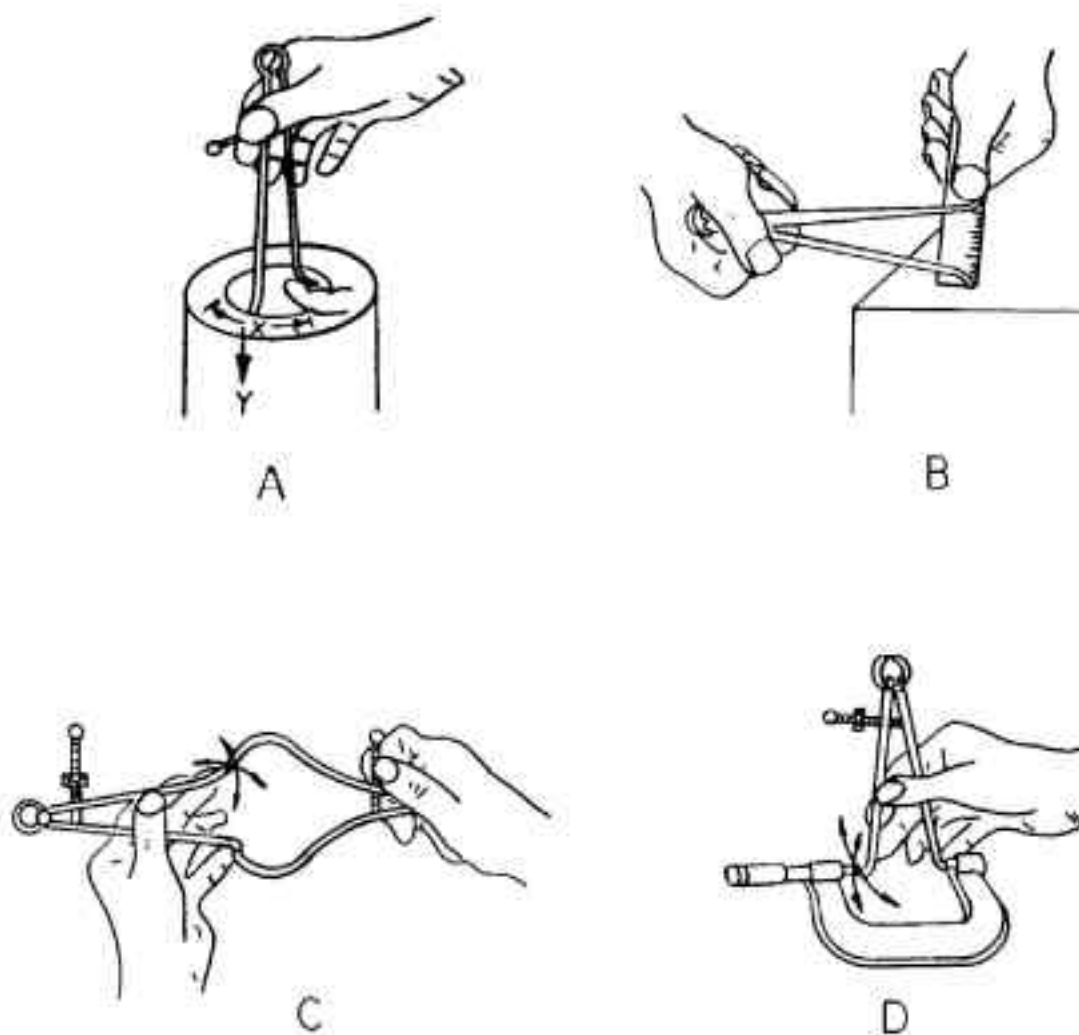
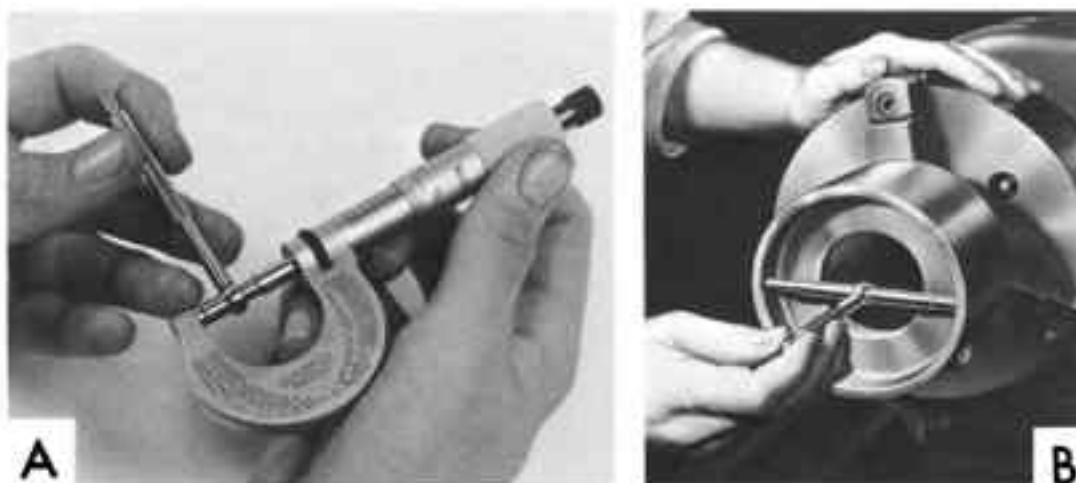


Fig. 2-5. Correct method of using inside calipers.



Courtesy of The L. S. Starrett Company

Fig. 2-6. Telescoping gages used to measure hole diameters.

"tenths" (.0001 inch) can be made, using both inside calipers and outside micrometer calipers. Measurements can also be transferred between inside and outside calipers as shown at C, in Fig. 2-5.

Telescoping-Gage and Small Hole Gages

The telescoping gage, Fig. 2-6A, has two contact plungers which expand outwardly across the diameter of a hole when the knurled nut at the end of the gage stem is released. When both plungers have contacted the hole, Fig. 2-6B, this nut is again tightened and the telescoping gage measurement is transferred to outside micrometer calipers. Small hole gages (Fig. 2-7) have two contact points that are expanded by turning the knurled nut on the end of the stem until the gage can pass through



Courtesy of The L. S. Starrett Company

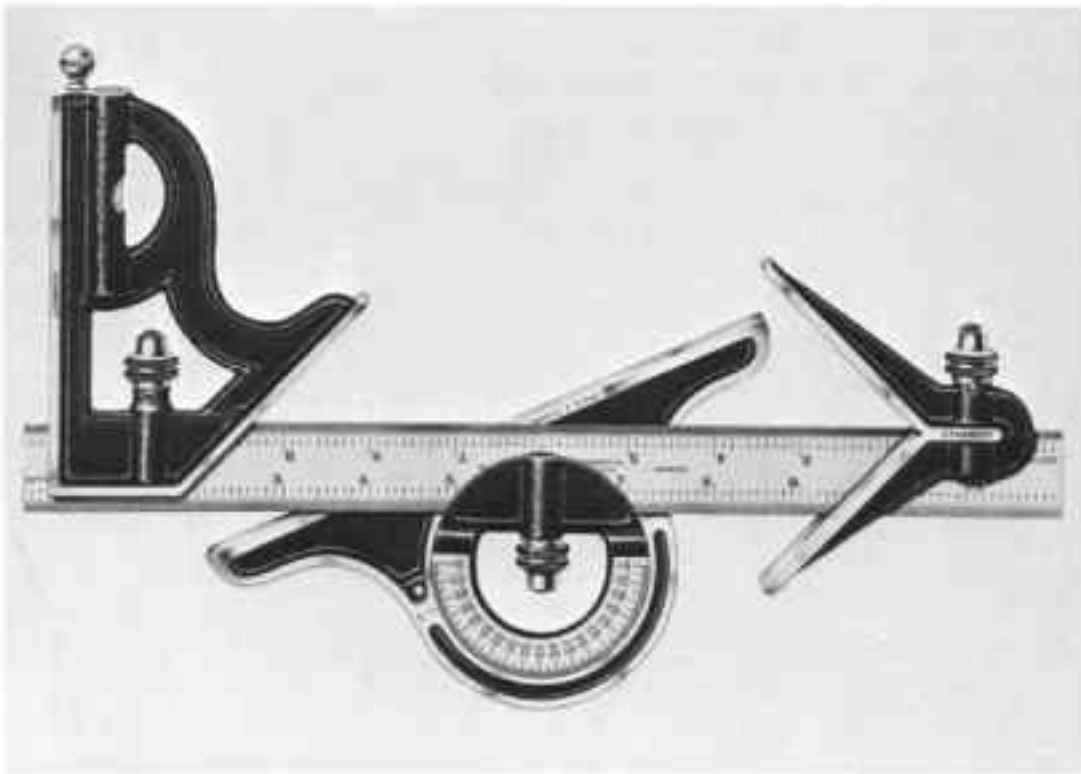
Fig. 2-7. Small hole gage-setting measured by outside-micrometer caliper.

the hole with a very light contact "feel." The hole measurement is then transferred to outside micrometer calipers as shown in the illustration.

The Square and the Bevel Protractor

A frequently encountered angle in machine shop work and in tool-making is the right angle, or 90 degree angle. The square is the basic tool used to measure and test for this angle. Other angles are measured with a protractor, illustrated in Figs. 2-8 and 2-16.

A combination set is shown in Fig. 2-8. Attached to a precision steel rule, from left to right, are a square head, a bevel protractor, and a center head. These heads can be changed to any desired position along the length of the rule or they may be removed completely. The square

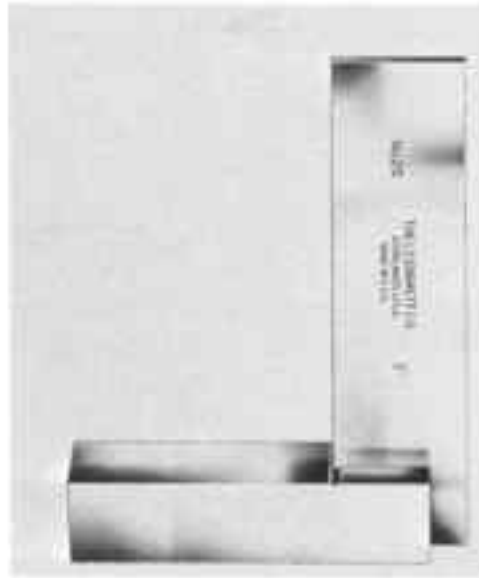


Courtesy of The L. S. Starrett Company

Fig. 2-8. Combination set including square head, bevel protractor, and center head.

head and the bevel protractor both contain spirit levels which are often of assistance in making measurements. The square head has a short edge machined at 45 degrees with respect to the right angle. The use of the center head will be described in the next chapter.

For very accurate determinations of the right angle, a master precision square, Fig. 2-9, is used. The beam and the blade of this square are hardened, ground, and lapped to insure straightness, parallelism, and the exact, right-angle relationship. It should be very carefully handled at all times. Squares may be checked by using two toolmakers buttons that are screwed to an angle plate as shown in Fig. 2-10. The lower button is screwed tight, while the other button is tightened also, but less firmly. The square and the angle plate are then set on a precision-surface plate and finally, the blade of the square is placed against the buttons. A thin piece of paper, or paper "feeler," is placed between the blade and each button. The upper button is then lightly tapped with a piece of soft metal, say a 1-inch dia. $\times$ 6-inch piece of soft brass, until both paper feelers are tight. The square is then positioned on the other side of the buttons and checked with paper feelers as before. If both paper feelers are tight the square is accurate. In making this test it is very important that both toolmakers buttons have exactly the same diameter. The thin paper



Courtesy of The L. S. Starrett Co.

Fig. 2-9. Master precision-steel square.

feelers are often much more sensitive than eyesight when making precise angular measurements.

Extremely accurate right-angle measurements can be made with the cylindrical square shown in Fig. 2-11. It is used on a precision-surface plate. This square is a perfectly true cylinder having one end lapped exactly at right angles to the cylindrical surface. The other end is lapped to a fixed angular relationship to the sides. When the angular end is resting on the surface plate and the work is in contact with the cylindrical surface, the deviation from the true square condition of the part can be measured by rotating the square until the light visible between the part and the square is shut out. The deviation is read by reading up the top-most dotted curve in contact with the work to the number which shows the deviation in steps of .0002 inch.

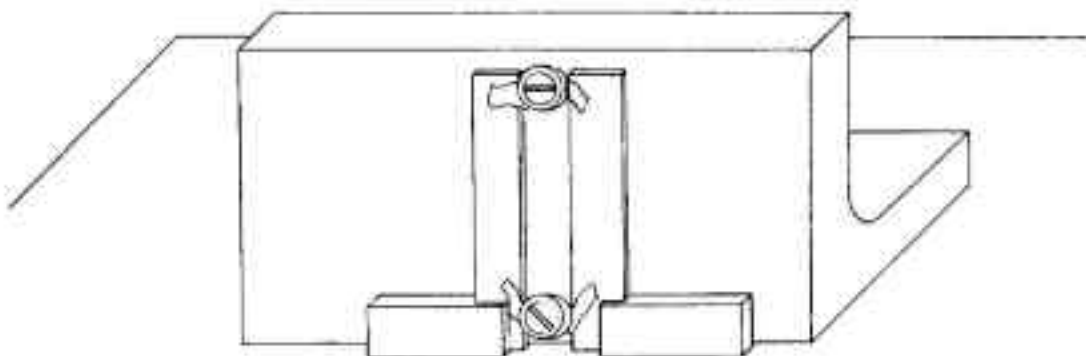
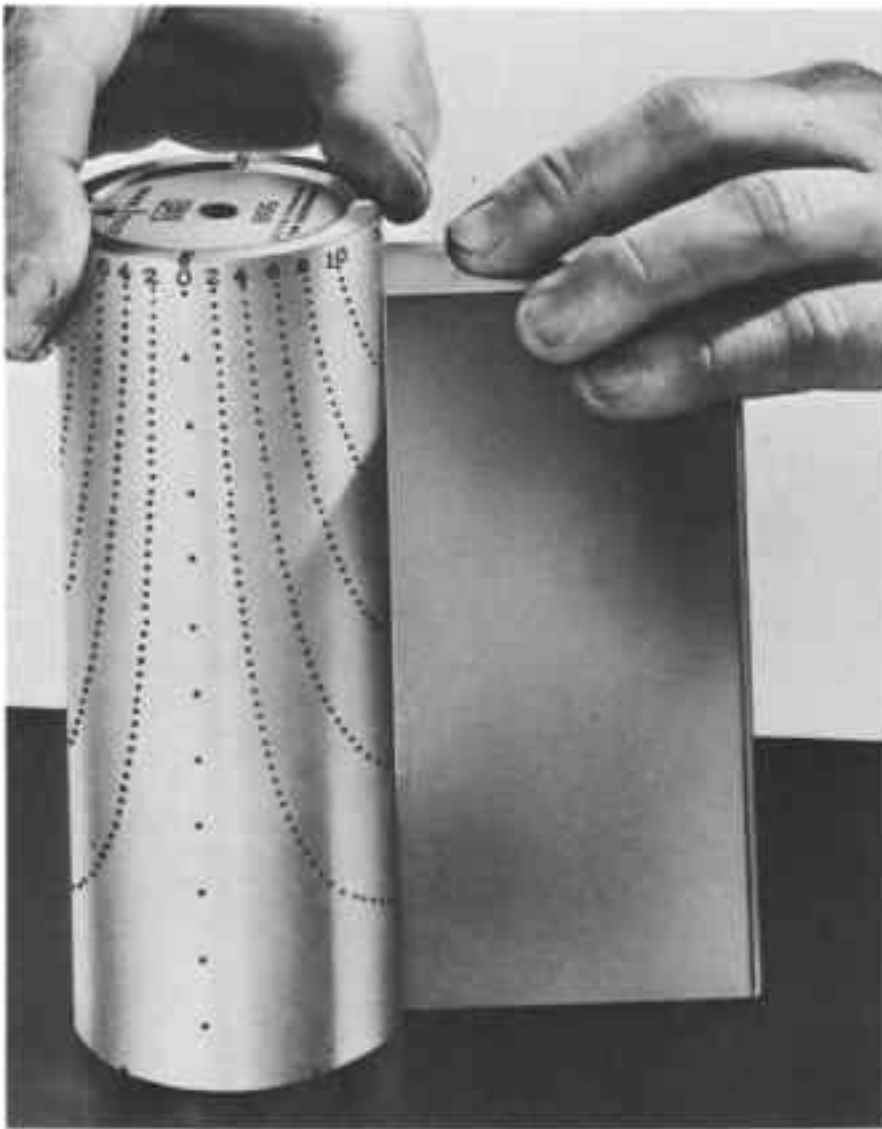


Fig. 2-10. Method for checking a square.



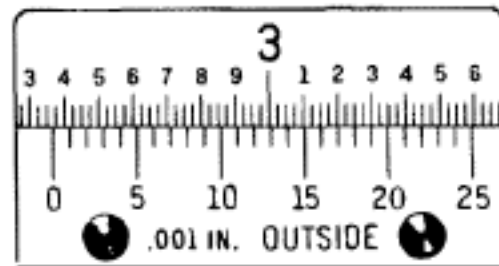
Courtesy of the Brown & Sharpe Manufacturing Co.

Fig. 2-11. Cylindrical square used to make precise measurements at right angles.

Vernier Measuring Instruments

The vernier is an auxiliary scale that is attached to calipers, height gages, depth gages, protractors, and other measuring instruments. It allows exceedingly accurate measurements to be made. The vernier measuring instrument is, in effect, a graduated steel rule to which a sliding member containing the vernier scale is attached. There are two types of vernier scales commonly used in the machine shop; one has a 25-division vernier scale and the other has a 50-division vernier scale.

The primary scale of a 25-division vernier is divided into inch graduations (large numbers), .100-inch graduations (small numbers), and .025-inch graduations (no numbers). The vernier scale has 25 divisions, and its length is equal to the length of 24 small divisions on the primary



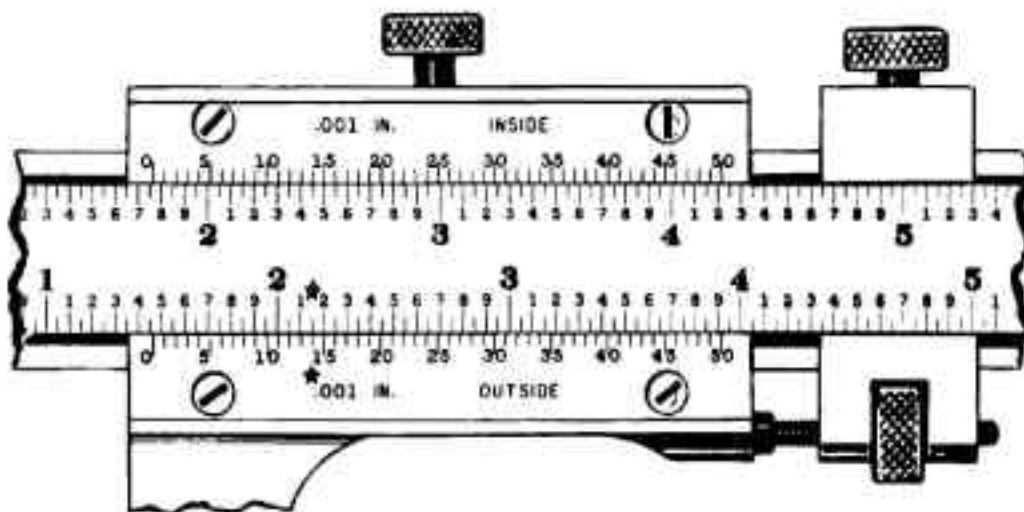
Courtesy of the Brown & Sharpe Manufacturing Co.

Fig. 2-12. 25-division vernier scale.

scale. Thus, the length of the vernier scale is $24 \times .025$ or $.600$ inch. Since the vernier scale has 25 equally spaced divisions, the length of each division must be $.600 \div 25$, or $.024$ inch. Therefore, each division on the vernier scale is $.001$ inch ($.025 - .024$) shorter than each small division on the primary scale. When the zero graduations on both scales are aligned with each other, the first two lines on the primary and the vernier scales are $.001$ inch apart; the second two lines will be $.002$ inch apart; the third pair are $.003$ inch apart; etc. It is evident that in order to move the vernier exactly $.002$ inch from this position, it is only necessary to slide it along the primary scale until the second line on the vernier coincides with the corresponding line on the primary scale. To read a 25-division vernier follow the steps given below and refer to Fig. 2-12:

- | | |
|--|-------------------|
| 1. Read the whole inches on the primary scale | 2.000 (not shown) |
| 2. Read the nearest small number to the left of the 0 indicator line and multiply times .100, or $3 \times .100$ | .300 |
| 3. Count the number of small graduations beyond the previous reading and multiply times .025, or $2 \times .025$ | .050 |
| 4. Find the graduation on the vernier scale that is in exact alignment with a graduation on the primary scale and multiply times .001, or $18 \times .001$ | .018 |
| 5. Add total to obtain reading. | 2.368 inches |

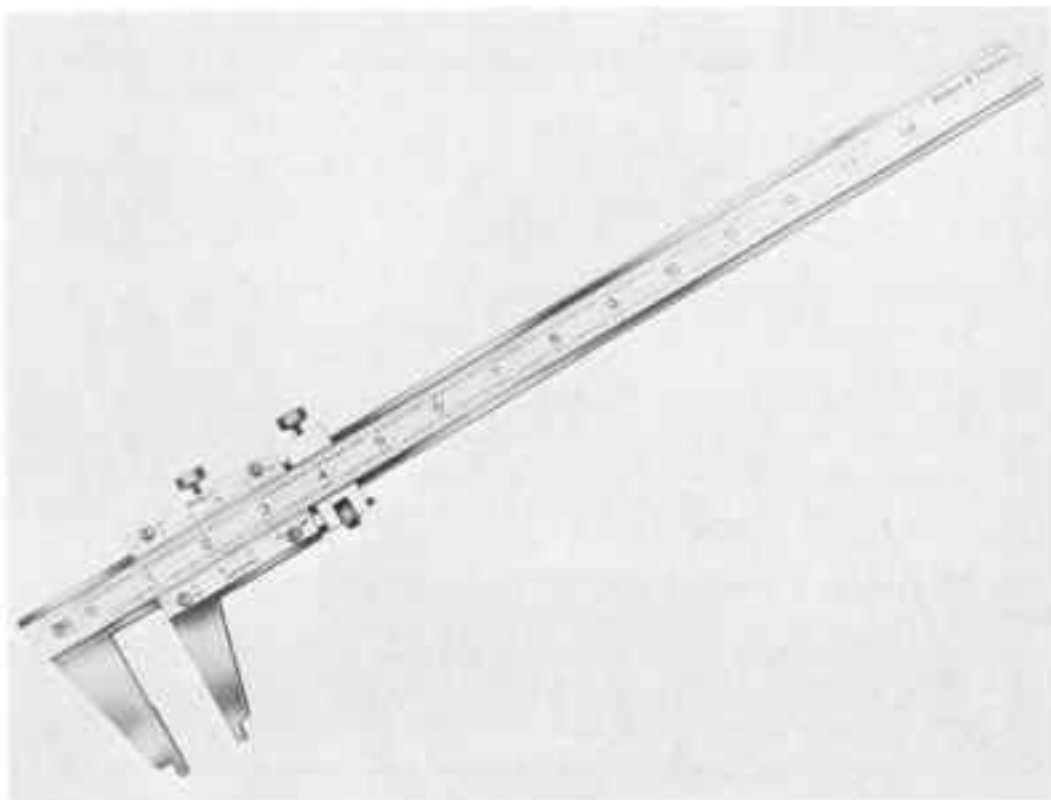
The 50-division vernier, Fig. 2-13, has more widely spaced graduations which are easier to read. The primary scale is divided into inch graduations, $.100$ -inch graduations, and $.050$ -inch graduations. The vernier scale is divided into 50 equally spaced graduations and its length is equal to the length of 49 divisions on the primary scale, or $49 \times .050 = 2.450$ inches. Each graduation on the vernier scale is then $2.450 \div 50$, or $.049$ inch long. Thus, the difference between the length of a graduation on the primary scale and on the vernier scale is again $.001$ inch. To read a 50-division vernier follow the steps below and refer to Fig. 2-13:



Courtesy of The L. S. Starrett Company

Fig. 2-13. 50-division vernier scale.

- | | |
|--|-------|
| 1. Read the whole inches on the primary scale | 1.000 |
| 2. Read the nearest small number to the left of the 0 indicator line and multiply times .100, or $4 \times .100$ | .400 |



Courtesy of the Brown & Sharpe Manufacturing Co.

Fig. 2-14. Vernier caliper.

3. Count the number of small graduations beyond the previous reading and multiply times .050, or $1 \times .050$.050
4. Find the graduation on the vernier scale that is in exact alignment with a graduation on the primary scale and multiply times .001, or $14 \times .001$.014
5. Add total to obtain reading. 1.464 inches

Vernier calipers capable of measuring both inside and outside diameters are shown in Fig. 2-14. Vernier height gages (Fig. 2-15A, and Fig. 2-27) are used to measure vertical distances above a reference plane; usually a precision-surface plate. They are also used to scribe lines at a given distance above a reference plane when making precise layouts on work-pieces. A vernier depth-gage is shown at B in Fig. 2-15.

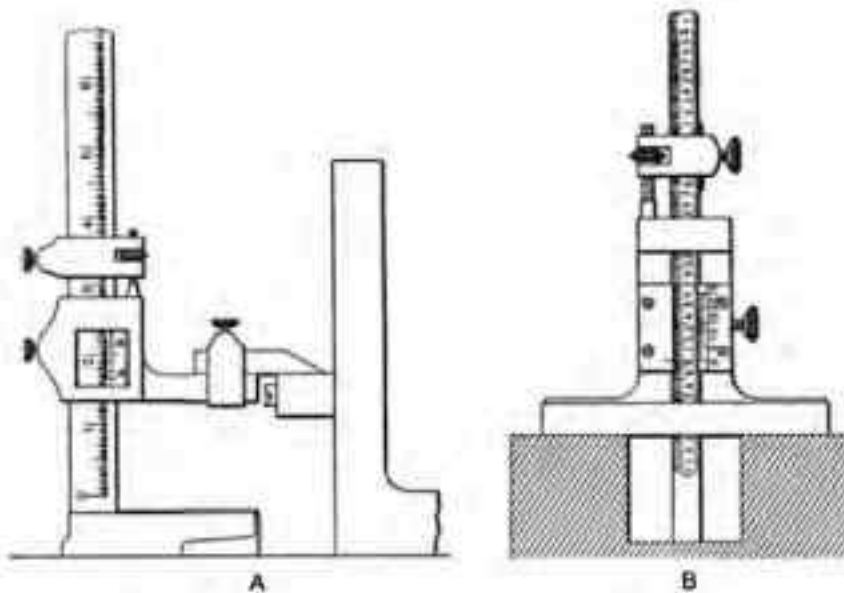
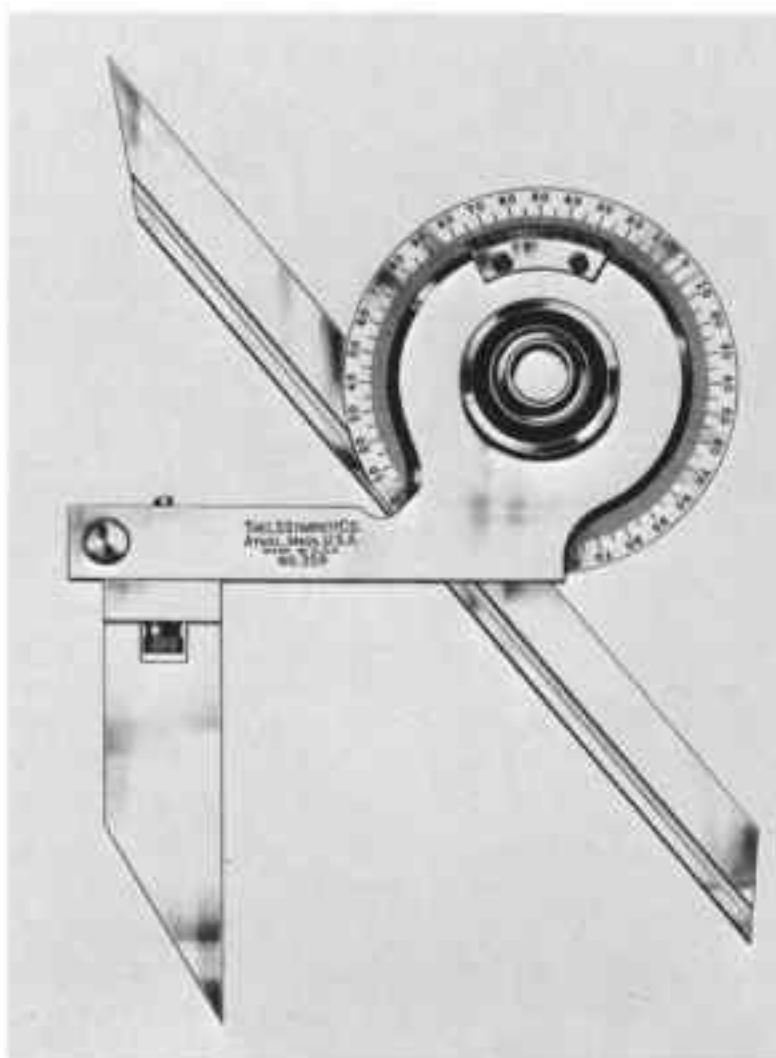


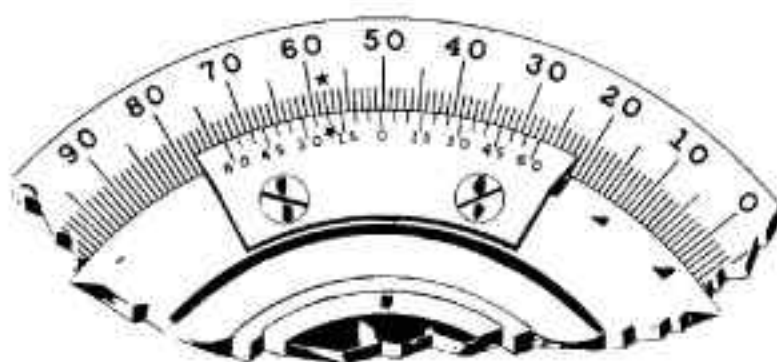
Fig. 2-15. A. Vernier height-gage. B. Vernier depth-gage.

Vernier bevel protractors, as shown in Fig. 2-16, are used to measure angles to an accuracy of five minutes ($05'$). Since there are sixty minutes ($60'$) in one degree, this is equal to one-twelfth ($\frac{5}{60} = \frac{1}{12}$) of a degree. An enlargement of the protractor scale and vernier is shown in Fig. 2-17. The main protractor scale is divided into degrees, with every ten degrees numbered. The vernier scale is actually *two* vernier scales, each having twelve divisions on either side of the zero graduation. The left-hand scale is used when the vernier zero graduation is moved to the left of the zero on the primary scale, while the right-hand scale is used when the movement is to the right. The total length of each vernier scale is equal to 23 degrees on the primary scale. Since there are twelve divisions on the vernier scale the length of each graduation is equal to $23 \div 12$ or $1\frac{11}{12}$ degrees. Since one degree equals $60'$, $1\frac{11}{12}$ degree equals $1\frac{11}{12} \times 60$ or $55'$. Thus, the length of each graduation on the vernier scale is equal to



Courtesy of The L. S. Starrett Co.

Fig. 2-16. Vernier bevel protractor.



Courtesy of The L. S. Starrett Co.

Fig. 2-17. Vernier bevel protractor scale.

$60' + 55'$ or $115'$. One graduation on the vernier scale is, then, $5'$ less in length than two graduations on the primary scale; two graduations on the vernier are $10'$ less than four graduations on the primary; etc.

To read the vernier bevel protractor, first read the number of whole degrees passed by the vernier zero and then count, in the same direction, the number of graduations between the vernier zero and that line which exactly coincides with a graduation on the primary or degree scale; this number multiplied by 5 will give the number of minutes to be added to the whole number of degrees. In Fig. 2-17, the vernier zero has passed to the left of 50° , and the fourth line to the left of the vernier zero coincides with a line on the degree scale. Hence, the reading is $50^\circ + 4 \times 5'$ or $50^\circ 20'$.

Micrometer Measuring Instruments

The micrometer screw principle is used on a variety of measuring instruments which includes outside micrometer calipers, inside micrometer calipers, and micrometer depth gages. The precision obtainable by all micrometer measuring instruments is dependent upon the micrometer screw; therefore, the thread of this screw is made with the greatest possible care and degree of precision. For this reason, micrometer screw threads are very seldom made longer than one inch and micrometer measuring

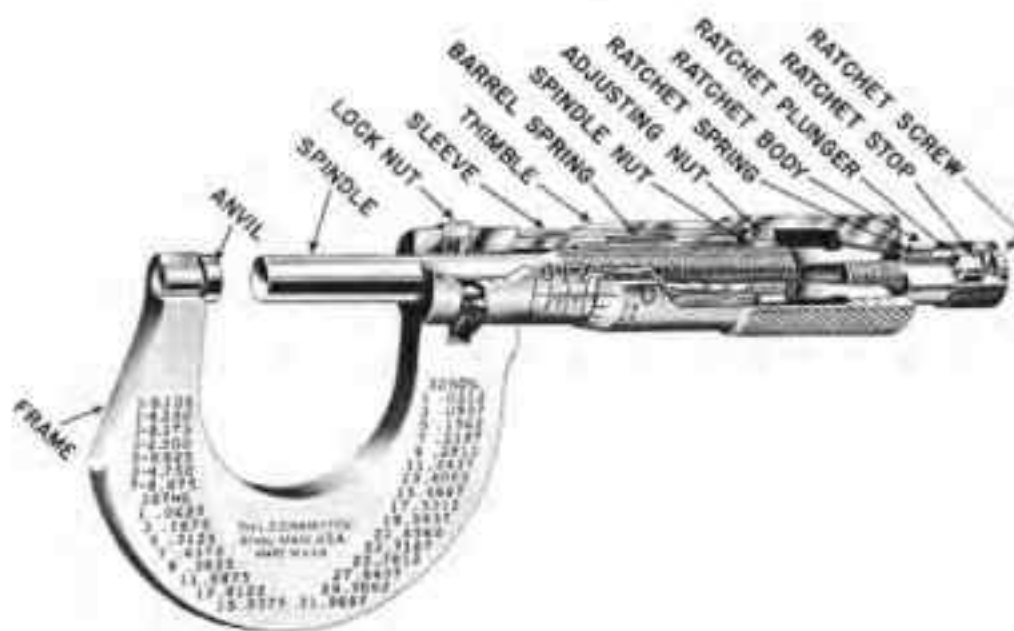


Courtesy of The L. S. Starrett Company

Fig. 2-18. Outside micrometer caliper set including length standards.

instruments are designed to accommodate a one-inch movement of the micrometer spindle. For example, outside micrometer calipers are made with frames having sizes of one-inch increments. Figure 2-18 shows three outside micrometer calipers of the following sizes: 0 to 1 inch, 1 to 2 inches, and 2 to 3 inches. Inside micrometer calipers and depth micrometer calipers have separate stems for each one-inch interval within the range of the instrument.

Figure 2-19 illustrates a sectional view of a one-inch, outside micrometer caliper with all of the parts named. The principal parts are the frame, sleeve, thimble, spindle, and anvil. The frame supports the components of the micrometer caliper and keeps them in correct alignment.



Courtesy of The L. S. Starrett Co.

Fig. 2-19. Sectioned view of 0-1 inch outside-micrometer caliper.

The sleeve houses the spindle nut and it has graduations marked lengthwise on its outside surface. These graduations are one inch long, and they are divided into .100-inch and .025-inch intervals. The thimble is attached to the spindle by a taper and a screw (see ratchet body in Fig. 2-19). The thimble rotates and moves lengthwise over the sleeve. One end of the thimble is beveled. The bevel on the thimble has twenty-five equally spaced graduations around its circumference. The spindle has the micrometer thread on one end and one of the two measuring surfaces on the face of the other. The other measuring surface is on the face of the anvil. These two measuring surfaces are made to be exactly parallel to each other regardless of the position of the spindle.

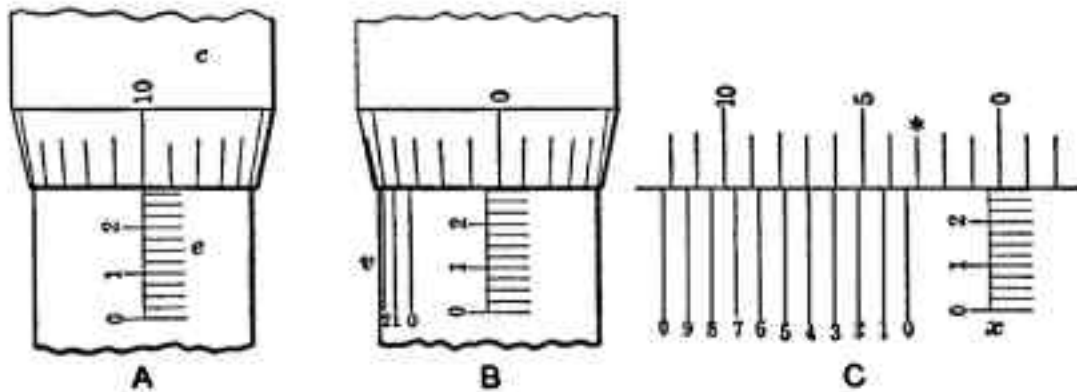
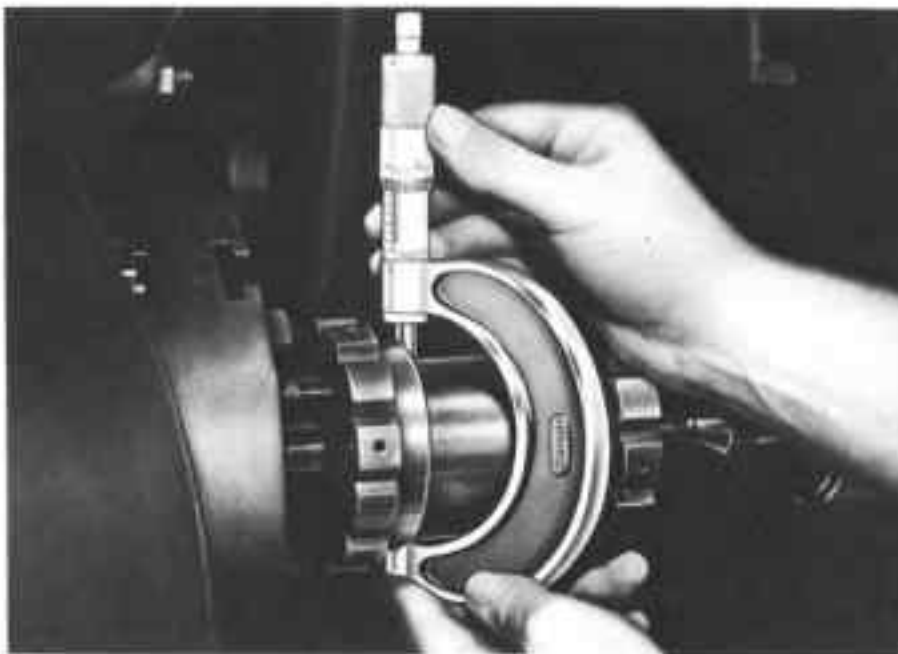


Fig. 2-20. Micrometer graduations.

The micrometer screw thread has 40 threads per inch. Its lead, or distance that the thread advances per revolution, is equal to $1 \div 40$ or .025 inch or the distance between the smallest graduations on the sleeve. The beveled edge of the thimble will move lengthwise one graduation on the sleeve for each revolution, since it is attached to the screw. Turning the thimble through one graduation on the thimble scale will cause the thimble and spindle assembly to rotate exactly $\frac{1}{25}$ of a revolution, since there are 25 equally spaced graduations around the thimble. When the spindle rotates $\frac{1}{25}$ of a turn, it moves lengthwise $\frac{1}{25}$ of its lead, or $\frac{1}{25} \times .025 = .001$ inch.

To read a micrometer scale follow the steps below and refer to A in Fig. 2-20.



Courtesy of The L. S. Starrett Company

Fig. 2-21. Measuring part held in a lathe using outside-micrometer calipers.

1. Read the largest number visible on the sleeve scale and multiply by .100, or $2 \times .100$.200
2. Count the number of small graduations beyond the graduation corresponding to the number read above, and multiply by .025, or $3 \times .025$.075
3. Read the graduation on the thimble scale and multiply by .001, or $10 \times .001$.010
4. Add to obtain reading. .285 inch

Some micrometers have a vernier scale, v , marked on the sleeve as shown at B, Fig. 2-20. The vernier scale, in conjunction with the regular micrometer scale, enables the micrometer to be read to .0001 inch. The relation between the graduations on the vernier and those of the regular micrometer is more clearly shown in C, in Fig. 2-20. The vernier has ten divisions which are the same length as nine graduations on the thimble. For convenience in reading, each graduation on the vernier scale is num-



Courtesy of The L. S. Starrett Company

Fig. 2-22. Measuring with large outside-micrometer caliper.



Courtesy of The L. S. Starrett Company

Fig. 2-23. The use of inside-micrometer calipers in measuring a large bore.

bered. The difference in width of a thimble division and a vernier division is equal to one-tenth of the thimble division. Therefore, a movement of the thimble equal to this difference results in a movement equal to $\frac{1}{10}$ of $\frac{1}{25}$ of a revolution, or $\frac{1}{10} \times \frac{1}{25} = \frac{1}{250}$ of a turn of the thimble and spindle assembly. This will cause the spindle to move lengthwise $\frac{1}{250} \times .025$, or .0001 inch.

In order to read a micrometer having a ten-thousandths vernier scale first determine the reading in thousandths, as with an ordinary micrometer, and then find a line on the vernier scale that exactly coincides with one on the thimble; the number on this line represents the number of ten-thousandths of an inch to be added to the number of thousandths obtained by the regular micrometer scales. For example, at C, in Fig. 2-20,

the micrometer scales read .275 inch. On the vernier scale, the fourth graduation coincides with a graduation on the thimble; therefore, $4 \times .0001$, or .0004 inch is added to the regular micrometer reading. The final reading is, then, $.275 + .0004$, or .2754 inch.

Accurate measurements made with outside micrometer calipers depend upon the way in which they are handled. In order to feel when they are in correct contact with the workpiece they must be held correctly when taking different types of measurements. Any movement of the frame, while manipulating the micrometer screw, should be kept to a minimum—just enough to establish the right feel. Methods of holding the micrometer are best shown by illustrations as follows: 1. Fig. 2-21, when a part is held in a lathe; 2. Fig. 2-7 and A, in Fig. 2-6, when the tool or part to be measured is held in the hand; 3. Figs. 2-1 and 2-22, when measuring with large micrometer calipers. Inside micrometer calipers, as shown in Fig. 2-23, must be manipulated through a hole, or recess, much in the same manner as inside calipers, in order to find the true diameter of the hole. It is advisable to check the accuracy of inside micrometer calipers with outside micrometer calipers before they are used. The use of depth micrometers is shown in Fig. 2-24. The thimble is carefully turned until the measuring rod just touches the surface to be measured.



Courtesy of The L. S. Starrett Company

Fig. 2-24. Using depth micrometers to measure shoulder depth.

The Starrett Mul-T-Anvil micrometer caliper in Fig. 2-25 can make a variety of measurements that cannot be made with a regular micrometer caliper; e.g., it can measure the wall thickness of tubing, the length of a shoulder, and the distance from a hole or slot to an edge. Two anvils are furnished with this micrometer: one is cylindrical in shape and the other is in the form of a stepped flat having a thickness of approximately .125 in. on one end and .060 in. on the other. Other special anvils are available. The anvils can be quickly interchanged by simply loosening the clamping vise jaw. In view A, Fig. 2-25, the Mul-T-Anvil micrometer is measuring the flange thickness on a special screw.

By removing the clamping vise jaw and the clamping screw, the Mul-T-Anvil micrometer can be converted into a micrometer height gage. One of its most useful applications is illustrated in view B, Fig. 2-25, where it is being used to measure the thickness of the workpiece on a surface grinder. Its range of measurement as a height gage can be extended beyond the one inch spindle travel of the micrometer by placing it on precision parallels or precision gage blocks having a known height.

Checking Micrometers

All micrometers should occasionally be checked for accuracy and, if necessary, adjusted to read correctly. Zero-to-one-inch micrometer calipers can be checked by noting if the micrometer reading is zero when the spindle touches the anvil. Both the spindle and the anvil should be cleaned beforehand by pulling a clean strip of paper between the anvil and the spindle. If a zero reading is not obtained, the sleeve should be adjusted by following the instructions supplied with the micrometer by the toolmaker. Larger micrometer calipers are checked in a similar way by placing a standard disc or pin between the anvil and the spindle and reading the micrometer. Standard disc and pin gages have a very precise known diameter or length, which is measured by the micrometer caliper in making the check. They are usually furnished with the larger size micrometers, although they can also be obtained separately. Again, the contact surfaces of the micrometer and the gages must be clean before making this check.

A more precise check can be made on micrometer calipers by measuring over precision gage blocks. (Precision gage blocks are described later on in this chapter.) The measurement obtained can be compared to the known size of the gage blocks. By measuring over a graduated series of different gage blocks, an indication of the accuracy of the micrometer screw can be obtained. Unless the micrometer is abused, the micrometer



Courtesy of The L. S. Starrett Company

Fig. 2-25. Starrett Mul-T-Anvil micrometer caliper. A. Used as a micrometer caliper. B. Used as a height gage to measure part on surface grinder.

screw will wear to the extent of being imprecise on only extremely rare occasions. If this does happen, the micrometer should be replaced. On rare occasions, the adjusting nut on the micrometer screw must be adjusted. Micrometer depth gages are checked by measuring from the top of a stack of precision gage blocks to the surface of a precision surface plate. The measurement obtained is compared to the known length of the gage blocks.

Inside micrometers are usually checked with micrometer calipers that are known to be precise. The inside micrometer is placed against the anvil. Then the spindle of the micrometer caliper is carefully adjusted to read the length of the inside micrometer while it is moved about very slightly to obtain the feel of the correct setting. The readings of the two micrometers are compared to make the check. Sometimes inside micrometers are checked with precision gage blocks and, when available, with precision ring gages.



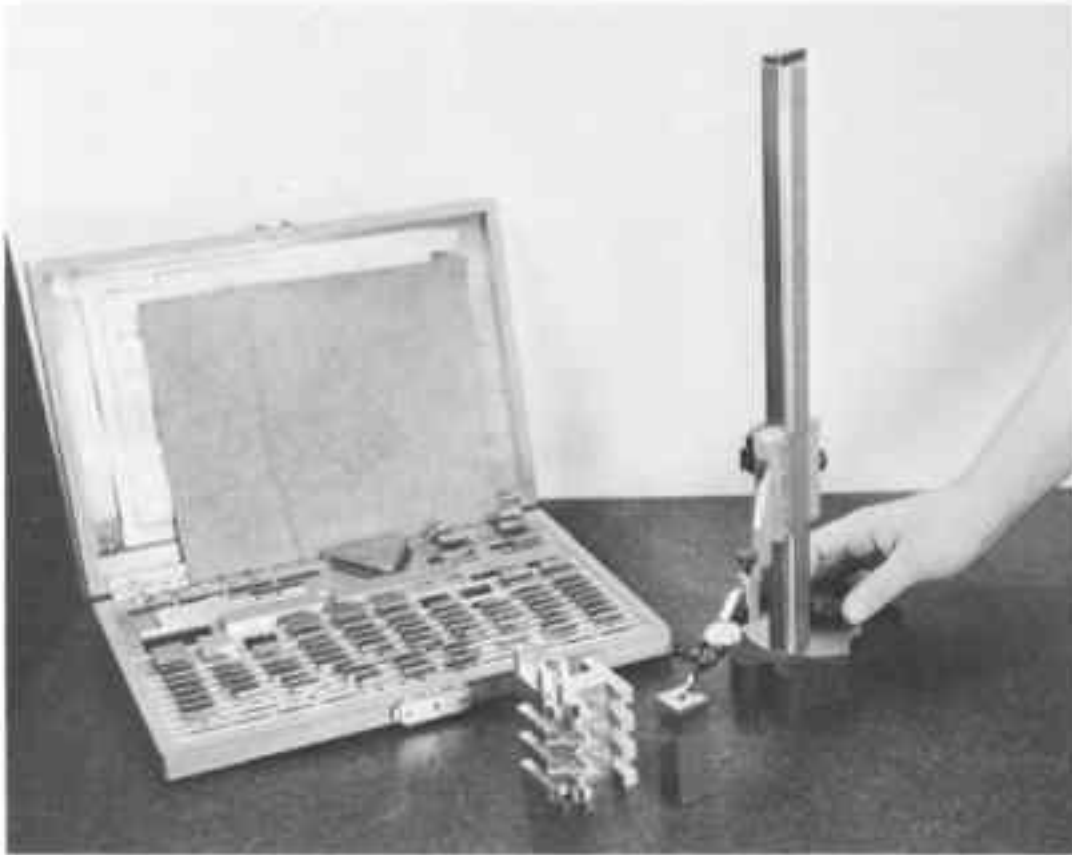
Courtesy of The L. S. Starrett Company

Fig. 2-26. Dial test indicator and planer gage—shown in background—are used to check the size of a machined surface on a planer.

Dial Test Indicators

The dial test indicator (see Fig. 2-26) is a sensitive instrument having a graduated dial and an indicating hand which is connected through a system of multiplying levers, or gears, to a contact point that is fastened to the end of a spindle. The contact point is placed in contact with the workpiece. Any movement of the contact point is transmitted through the spindle, greatly amplified by the internal works, and displayed by a movement of the indicating hand on the dial. Dial test indicators are designed to read to .001 inch, .0005 inch, or .0001 inch. The reading of the divisions is usually marked on the dial face. They are made in a great variety of sizes and shapes. Indicators are widely used in machine shops, and tool and die shops in combination with many other forms of gaging and measuring devices. In the shop, dial test indicators are frequently simply referred to as "indicators."

The dial test indicator in Fig. 2-26 is used to check the depth of cut on a flat casting. The indicator reading is compared to the setting of a planer gage shown in the background. Dial test indicators are used with vernier height gages, Fig. 2-27, to assure uniform contact pressure. The height gage is set by making the indicator read zero on all surfaces to be measured. Among other applications of indicators are these: to locate edges and holes for precision boring, to measure concentricity, to "zero"



Courtesy of The L. S. Starrett Company

Fig. 2-27. Precision gage blocks used with dial test indicator attached to vernier height-gage to measure workpiece.

the position of machine tool slides, and to measure parallelism between surfaces. The operation of using a dial test indicator is usually called "indicating."

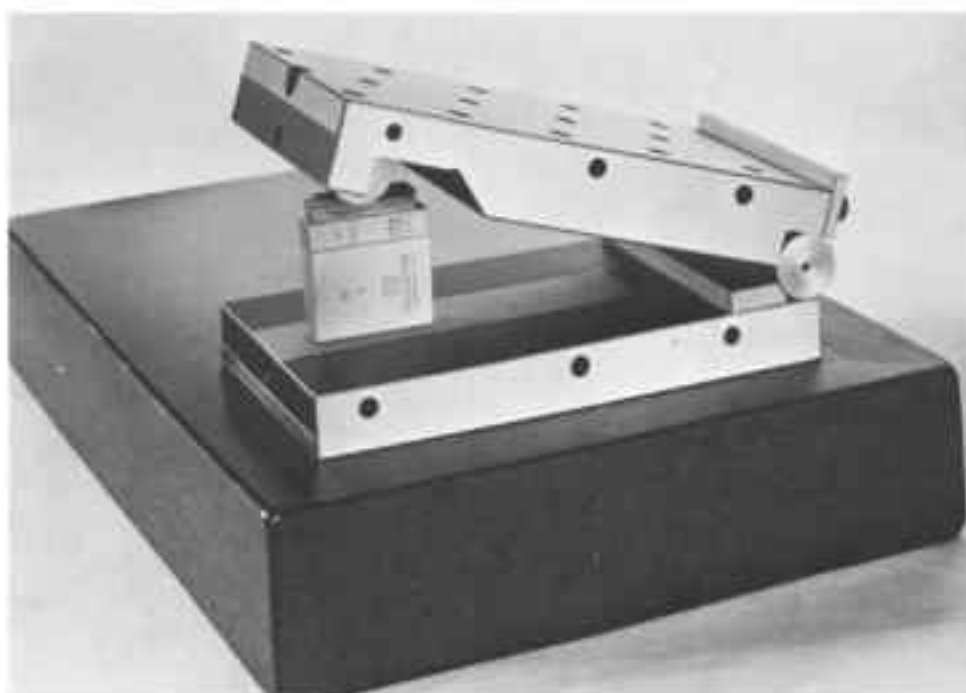
Precision Gage Blocks

Precision gage blocks, Fig. 2-27, are small blocks of steel that have been heat treated to a high hardness. The heat treatment is also designed to obtain a high degree of dimensional stability in the blocks. They are made to have an extremely high degree of dimensional accuracy as shown in Table 2-1. There are several grades of gage blocks which can be categorized as master blocks, inspection blocks, and working blocks. Working blocks are used in the workshop. They are used to calibrate micrometers and other precision measuring tools, to inspect dies, tools, and other precision machine parts. In Fig. 2-27 a gage block is shown on a precision surface plate. It is used with the height gage and the dial test indicator to measure the height of a surface on the workpiece by comparing the height of the gage block and the workpiece with the indicator. Gage blocks are also used to measure angles in conjunction with sine bars and

Table 2-1. Accuracy Standards for Precision Gage Blocks

Classification	English Measure Standards			Surface Finish	Metric Measure Standards		
	Length	Flatness	Parallelism		Length	Flatness	Parallelism
U.S. Federal AA	± .000002	.000004	.000003	.5	± .00005	...	...
British Reference Grade	± .000002	.000003	.000003		± .00005	.00008	.00008
U.S. Federal A	+ .000006 - .000002	.000004	.000004	.8	+ .00015 - .00005	...	...
British Calibration Grade	± .000005	.000003	.000003	...	± .00012	.00008	.00008
German Grade O	...	...	...	...	± .00014	.00010	.00010
U.S. Federal B	+ .000010 - .000006	.000006	.000005	1.2	+ .00025 - .00015	...	...
British Inspection Grade	+ .000007 - .000003	.000005	.000005	...	+ .00020 - .00010	.00010	.00010
German Grade I	...	...	...	...	± .00030	.00015	.00015
U.S. Federal	...	...	...	...	...	...	...
British Workshop Grade	+ .000010 - .000005	.000010	.000010	...	+ .00025 - .00010	.00025	.00025
German Grade II	...	...	...	...	± .00070	.00025	.00025
German Grade III	...	...	...	...	± .00140	.00050	.00050

U.S. Federal Specification GGG-G-15.—British Standards 888-1950.—German Standards DIN 861.



Courtesy of the Brown & Sharpe Manufacturing Co.

Fig. 2-28. Precision gage blocks and sine plate used to measure angles.

sine plates, as shown in Fig. 2-28. Angles are sometimes machined on sine plates which are set by using gage blocks. Precision, angular gage blocks are also made for making angular measurements.

Gage blocks are sold in sets. A set of gage blocks is shown in Fig. 2-27. The sizes for an 83-piece, gage-block set are given in Table 2-2. By combining two or more gage blocks, a large range of extremely accurate dimensions can be obtained. The blocks can be combined in the following way when there is in the set no single block of the exact size that is wanted. Two or more blocks are assembled, or combined, by wringing the blocks together to form the equivalent of a single block of a given dimension. First, the joining surfaces of the blocks are cleaned. One block is then placed over the other at an angle and the two are twisted, or wrung, together. When properly wrung together, the blocks will adhere to each other so strongly that a very considerable force is required to separate them. They are best separated by twisting them apart in a manner similar to the way in which they were assembled.

In selecting gage blocks for a given dimension, the least number of blocks that will give the required dimension should be used. This is accomplished by successively eliminating the smallest remaining dimension.

Example 2-1.

Determine the gage blocks required to obtain 3.6742 inches. (Refer to Table 2-2.)

Table 2-2. Sizes for an 83-piece Gage-Block Set

First: .0001 Series (9 Blocks)												
	.1001	.1002	.1003	.1004	.1005	.1006	.1007	.1008	.1009			
Second: .001 Series (49 Blocks)												
.101	.102	.103	.104	.105	.106	.107	.108	.109	.110	.111	.112	.113
.114	.115	.116	.117	.118	.119	.120	.121	.122	.123	.124	.125	.126
.127	.128	.129	.130	.131	.132	.133	.134	.135	.136	.137	.138	.139
.140	.141	.142	.143	.144	.145	.146	.147	.148	.149			
Third: .050 Series (19 Blocks)												
.050	.100	.150	.200	.250	.300	.350	.400	.450	.500	.550	.600	
.650	.700	.750	.800	.850	.900	.950						
Fourth: 1.000 Series (4 Blocks)												
	1.000	2.000	3.000	4.000								
	Two .050 Wear Blocks											

Step 1. Eliminate the .0002 inch by selecting a .1002 block. Subtract .1002 from 3.6742.

$$\begin{array}{r} 3.6742 \\ - .1002 \\ \hline 3.5740 \end{array}$$

Step 2. Eliminate the .004 inch by selecting a .124 block. Subtract .124 from the remainder in Step 1.

$$\begin{array}{r} 3.574 \\ - .124 \\ \hline 3.450 \end{array}$$

Step 3. Eliminate the .450 inch with a .450 block. Subtract .450 from the remainder in Step 2. This obviously leaves 3.000 which is obtained with a 3-inch block.

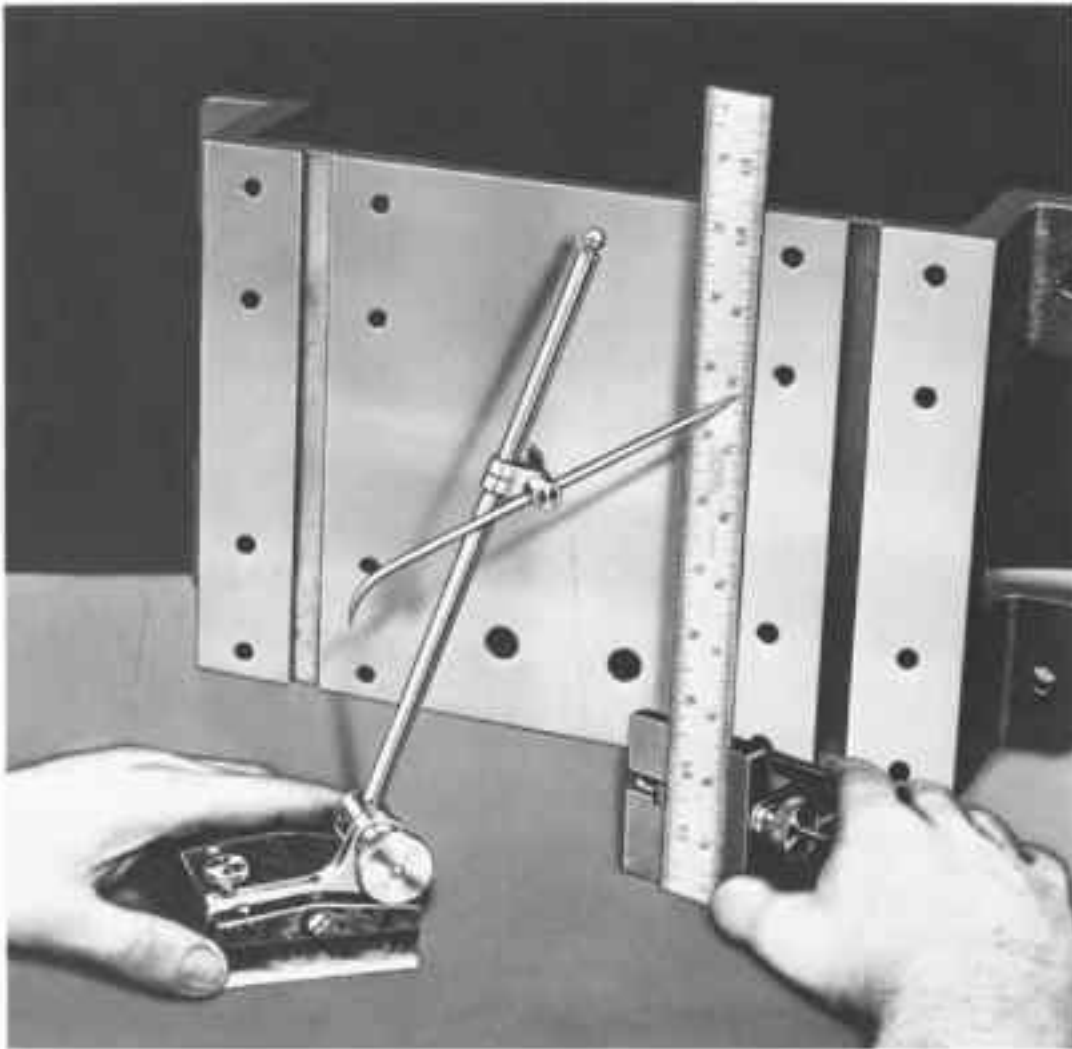
Step 4. Check the blocks to determine if they will result in the required dimension.

$$\begin{array}{r} .1002 \\ .124 \\ .450 \\ \hline 3.000 \\ 3.6743 \text{ inches} \end{array}$$

Surface Gage

The surface gage, Fig. 2-29, is one of the most useful tools employed by the machinist and the toolmaker. On certain classes of work it is almost an indispensable tool. For example, surface gages can be used to scribe layout lines on workpieces in order to locate the centers of holes, to provide reference lines for setting up workpieces on the machine, or to outline surfaces to be machined. In Fig. 2-29, the scriber point of the surface gage is being positioned by a rule in preparation for scribing a line on the workpiece. The rule is conveniently held in a vertical position by a rule holder; a surface plate provides a reference plane for transferring the measurement from the rule holder to the workpiece by means of the surface gage. Located at the end of a rocker plate, a vertical thumb-screw provides a very fine adjustment to the spindle and allows accurate settings of the scriber to be made.

A dial test indicator, which can be used to transfer precise measurements, can be attached to the spindle or to the scriber. The scriber point can be used to check the position of layout lines when setting up a workpiece on the machine. The bent point of the scriber can be used to lay out lines that are close to the base of the surface gage. It is also used to test the parallelism between a surface on the workpiece and the top of a surface plate or machine tool table. This can be done whether the surface of the workpiece has been finished smooth or if it is still rough. The use



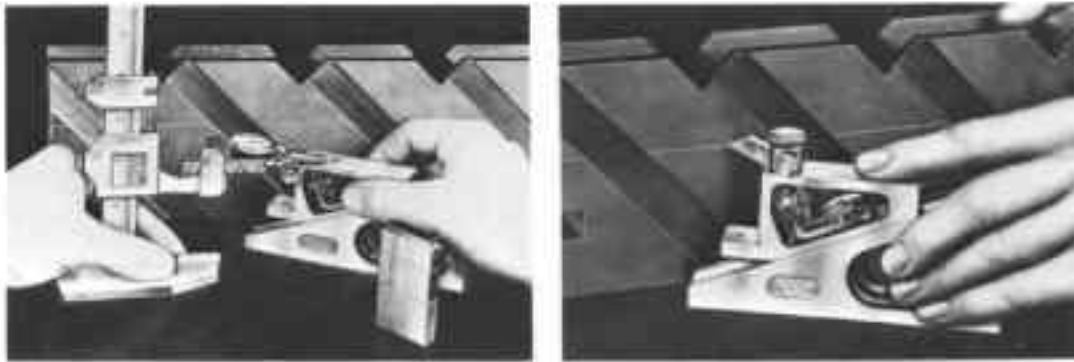
Courtesy of The L. S. Starrett Company

Fig. 2-29. Surface gage used to transfer a measurement from a 12-inch rule. The rule is held in a rule holder.

of the surface gage to test a cast surface for parallelism is especially effective because the surface gage can average out the irregularities on such surfaces.

Universal Precision Gage (Planer Gage)

A universal precision gage is shown in Fig. 2-30. This gage was developed from a planer gage by the incorporation of a fine adjustment mechanism. Originally, planer gages were used for setting planer and shaper cutting tools to establish the correct depth of cut. The universal precision gage can still be used for this purpose. Many other uses have been found for this tool in doing precision machine-tool work. The slide can be precisely adjusted to any vertical position by measuring across the base and the top of the slide with a micrometer caliper, or by transferring the measurement from a stack of gage blocks with a height gage



Courtesy of The L. S. Starrett Company

Fig. 2-30. (Left) Setting a universal precision gage using a height gage, dial test indicator, and precision gage blocks. (Right) Scribing a layout line with a universal precision gage.

and a dial test indicator, as shown in the illustration at left. A scriber can then be attached to the universal precision gage in order to scribe layout lines. Other uses include measurements of parallel faces in a slot, as an adjustable parallel, for setting up and leveling workpieces, and as a height gage for transferring measurements from gage blocks by using a dial test indicator mounted on the universal precision gage.

Parallels

While not primarily a measuring tool, parallels are indispensable tools in machine shop and toolroom work. They are used for the setup of workpieces on machine tools, when inspecting parts, and when making layouts on parts. Parallels are rectangular bars of hardened or unhardened steel that have two sides ground exactly parallel; some parallels have two pairs of sides ground parallel. Frequently parallels are made in pairs, each pair being ground together so that they are exactly the same size. Very large parallels are usually made of alloy cast iron; highly precise parallels are also made from granite because this material is very stable dimensionally. Hardened steel precision parallels have two pairs of sides parallel, the sides being square and parallel to "tenth" (.0001 in. or 0.002 mm) tolerances. These parallels are made in fractional inch sizes ranging from $\frac{1}{4} \times \frac{3}{8} \times 6$ inches to $1\frac{1}{2} \times 3 \times 12$ inches; metric sizes range from 6 x 10 x 150 mm to 38 x 76 x 300 mm. Special sizes are often used, when needed.

Although not classified as parallels, 1-2-3 blocks serve the same purpose. Made of hardened or unhardened steel to "tenth" tolerances, their dimensions are 1 x 2 x 3 inches. Usually their weight is reduced by drilled holes, which may or may not be tapped.

Adjustable parallels are precision tools that could be classified as gages or measuring tools. They may be used as precision parallels; however, one of the most important applications of these tools is illustrated in Fig. 2-31,



Courtesy of The L. S. Starrett Company

Fig. 2-31. Measuring the width of a slot using adjustable parallels and micrometer calipers.

where it is being used to measure the width of a slot. The adjustable parallel is inserted in the opening and expanded to fit the slot. The size of the slot is then measured with a micrometer caliper by measuring over the adjustable parallel. Each adjustable parallel has its own size range; e.g., $\frac{3}{8}$ to $\frac{1}{2}$ inch, $\frac{1}{2}$ to $1\frac{1}{16}$ inch (9.5 to 12.7 mm, 12.7 to 17.5 mm). A set of six parallels that are available commercially will cover all sizes from $\frac{3}{8}$ to $2\frac{1}{4}$ inches, or 9.5 to 57.1 mm.

Metrication

The metric system adopted by all countries is the International System of Units, or SI. In machine shop practice the application of this system is primarily concerned with linear measurements. The metric units of linear measurement recommended by SI are the meter, kilometer, and millimeter, although the centimeter may also be used in some instances. For all precision machine work the millimeter will be used exclusively. Dimensions less than one millimeter are always expressed as a decimal fraction; common fractions ($\frac{1}{8}$, $\frac{1}{4}$, etc.) are not used as they are with customary inch units.

There are exactly 25.4 millimeters in an inch; thus, one inch is equal to 25.4 millimeters, exactly. Conversely, one millimeter is equal to .0393700787 inch, which is usually rounded off to .03937 inch.

To convert inches and millimeters from one to another, use the following rules:

multiply inches by 25.4 to obtain millimeters
multiply millimeters by .03937 to obtain inches.

Another method of converting is to use Table 2-3. When the value wanted cannot be found directly in this table, find the values of an equivalent sum; then convert the equivalent values and find their sum to obtain the converted value. This procedure is illustrated in the following examples.

Example 2-2:

Convert .4474 inch to millimeters.

Observe that $.4474 = .4 + .04 + .007 + .0004$. Find the millimeter equivalent of each term of the sum and add.

$$\begin{array}{rcl} .4 & \text{in.} & = 10.16000 \text{ mm} \\ .04 & \text{in.} & = 1.01600 \text{ mm} \\ .007 & \text{in.} & = .17780 \text{ mm} \\ .0004 & \text{in.} & = .01016 \text{ mm} \\ \hline .4474 & \text{in.} & = 11.36396 \text{ mm} \end{array}$$

Example 2-3:

Convert 80.92 mm to inches.

$$\begin{array}{rcl} 80. & \text{mm} & = 3.14961 \text{ in.} \\ .9 & \text{mm} & = .03543 \text{ in.} \\ .02 & \text{mm} & = .00079 \text{ in.} \\ \hline 80.92 & \text{mm} & = 3.18583 \text{ in.} \end{array}$$

The converted dimension must be rounded-off to be consistent with the degree of precision required by the job. In Example 2-2, when working to the millimeter equivalent of a "thousandths" of an inch (.001 in.) the answer would be 11.36 mm; when working to "tenths" (.0001 in.) the answer is 11.364 mm. In Example 2-3 the answer is 3.186 in. and 3.1858 in. when working in "thousandths" and "tenths" respectively. When converting precise dimensions, however, in some cases the converted dimension must not exceed and in others not be less than the original dimension. For example, if the dimension in Example 2-3 is a shaft diameter, the converted dimension may be required not to exceed the original dimension, in which case it would be 3.185 in. when working to "thousandths." If it is a hole diameter the converted dimension may be required not to be less than the original dimension, in which case it should be 3.186 in. as before. In Example 2-2, the shaft and hole diameters would be 11.36 mm

Table 2-3. Inch—Millimeter and Inch—Centimeter
Conversion Table *
(Based on 1 inch = 25.4 millimeters, exactly)

INCHES TO MILLIMETERS											
in.	mm	in.	mm	in.	mm	in.	mm	in.	mm	in.	mm
10	254.0000	1	25.4000	.1	2.5400	.01	.2540	.001	.0254	.0001	.00254
20	508.0000	2	50.8000	.2	5.0800	.02	.5080	.002	.0508	.0002	.00508
30	762.0000	3	76.2000	.3	7.6200	.03	.7620	.003	.0762	.0003	.00762
40	1,016.0000	4	101.6000	.4	10.1600	.04	1.0160	.004	.1016	.0004	.01016
50	1,270.0000	5	127.0000	.5	12.7000	.05	1.2700	.005	.1270	.0005	.01270
60	1,524.0000	6	152.4000	.6	15.2400	.06	1.5240	.006	.1524	.0006	.01524
70	1,778.0000	7	177.8000	.7	17.7800	.07	1.7780	.007	.1778	.0007	.01778
80	2,032.0000	8	203.2000	.8	20.3200	.08	2.0320	.008	.2032	.0008	.02032
90	2,286.0000	9	228.6000	.9	22.8600	.09	2.2860	.009	.2286	.0009	.02286
100	2,540.0000	10	254.0000	1.0	25.4000	.10	2.5400	.010	.2540	.0010	.02540
MILLIMETERS TO INCHES											
mm	in.	mm	in.	mm	in.	mm	in.	mm	in.	mm	in.
100	3.93701	10	.3937	1	.03937	.1	.00394	.01	.00039	.001	.00004
200	7.87402	20	.7874	2	.07874	.2	.00787	.02	.00079	.002	.00008
300	11.81102	30	1.1811	3	.11811	.3	.01181	.03	.00118	.003	.00012
400	15.74803	40	1.5748	4	.15748	.4	.01575	.04	.00157	.004	.00016
500	19.68504	50	1.9685	5	.19685	.5	.01969	.05	.00197	.005	.00020
600	23.62205	60	2.3622	6	.23622	.6	.02362	.06	.00236	.006	.00024
700	27.55906	70	2.7559	7	.27559	.7	.02756	.07	.00276	.007	.00028
800	31.49606	80	3.1496	8	.31496	.8	.03150	.08	.00315	.008	.00031
900	35.43307	90	3.5433	9	.35433	.9	.03543	.09	.00354	.009	.00035
1,000	39.37008	100	3.9370	10	.39370	1.0	.03937	.10	.00394	.010	.00039
* For inches to centimeters, shift decimal point in mm column one place to left and read centimeters, thus: 40 in. = 1016 mm = 101.6 cm For centimeters to inches, shift decimal point of centimeter value one place to right and enter mm column, thus: 70 cm = 700 mm = 27.55906 inches											

and 11.37 mm respectively. This procedure should, however, be applied only when conditions affecting the dimensions are known; otherwise, use the procedure first described.

Metric Steel Rules

The graduations on metric rules are in millimeters on one scale and one-half millimeters on the other. Usually each ten millimeter length on the rule is numbered on consecutive sequence. On the steel rule in Fig. 2-32, the direction of the scales on the two sides are reversed for the convenience of the user. To compare the smallest graduation on metric and inch rules: 1.0 mm is equal to .039 in., as compared to .031 in. for $\frac{1}{32}$ in.; 0.5 mm is equal to .0197 in., as compared to .0156 in. for $\frac{1}{64}$ in. Metric rules are made in lengths of 150 mm (approximately equivalent to a 6 inch rule), 300 mm, 500 mm, and 1000 mm. Steel rules are available having metric graduations on one side and inch graduations on the other side.

Metric Micrometers

Two metric micrometer calipers are shown in Fig. 2-33. They are identical except that the one at B has a vernier scale while the other does not. The micrometer in view A has graduations that will read to an accuracy of 0.01 mm (.00039 in.); the addition of the vernier scale on the micrometer in view B allows it to be read in 0.002 mm (.00008 in.) increments. The usual range of each metric micrometer is 0 to 25 mm, corresponding to 0 to .984 inch; metric micrometers are available in sizes up to 600 mm and larger, if required.

As shown in Fig. 2-34, the reading line on the sleeve is graduated in one millimeter (1.00) increments above the line, and below the line each millimeter is divided in half by a graduation. Therefore, the reading line is divided into one-half millimeter increments by the graduations above and below the reading line. Every fifth line above the reading line is numbered from 0 to 25, indicating five millimeter (5.00) intervals between the numbered lines.

The pitch of the spindle screw on metric micrometers is one-half millimeter (0.50 mm). One revolution of the spindle will, therefore, advance the spindle 0.50 mm toward or away from the anvil. Since the thimble moves with the spindle, it will also move 0.50 mm, which is equal to the distance between graduations on the reading line.

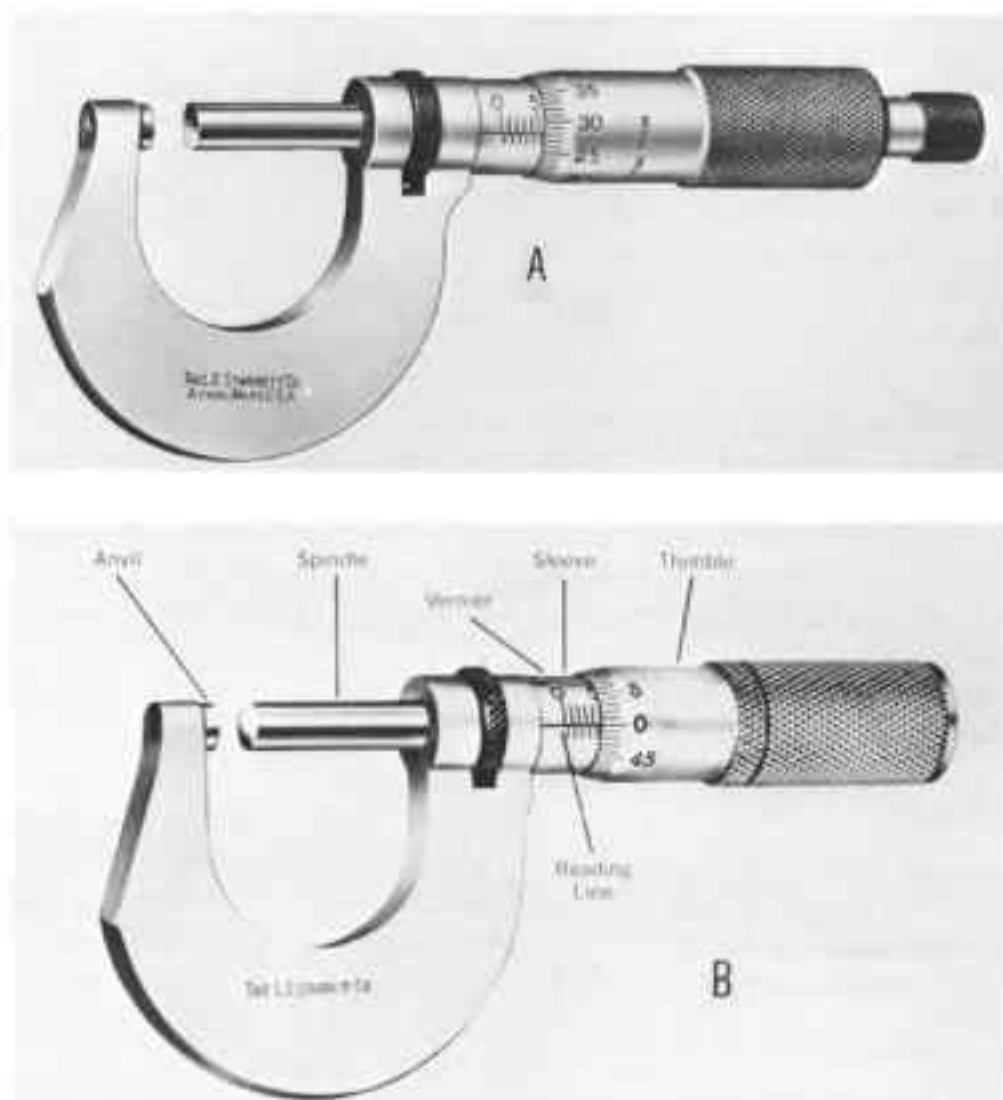
The beveled edge on the thimble is graduated into 50 divisions, every fifth line being numbered. Each division represents a rotation of $\frac{1}{50}$ of a revolution by the thimble and the spindle screw. Since one complete revolution of the thimble and spindle screw causes a movement of 0.50 mm by the spindle, $\frac{1}{50}$ of a revolution will cause a movement of $\frac{1}{50} \times 0.50$ mm, or 0.01 mm. Thus, one graduation on the thimble equals 0.01 mm, two graduations 0.02 mm, three graduations 0.03 mm, etc.

To read the metric micrometer, simply add the number of millimeters



Courtesy of The L. S. Starrett Company

Fig. 2-32. Steel rule with metric graduations.

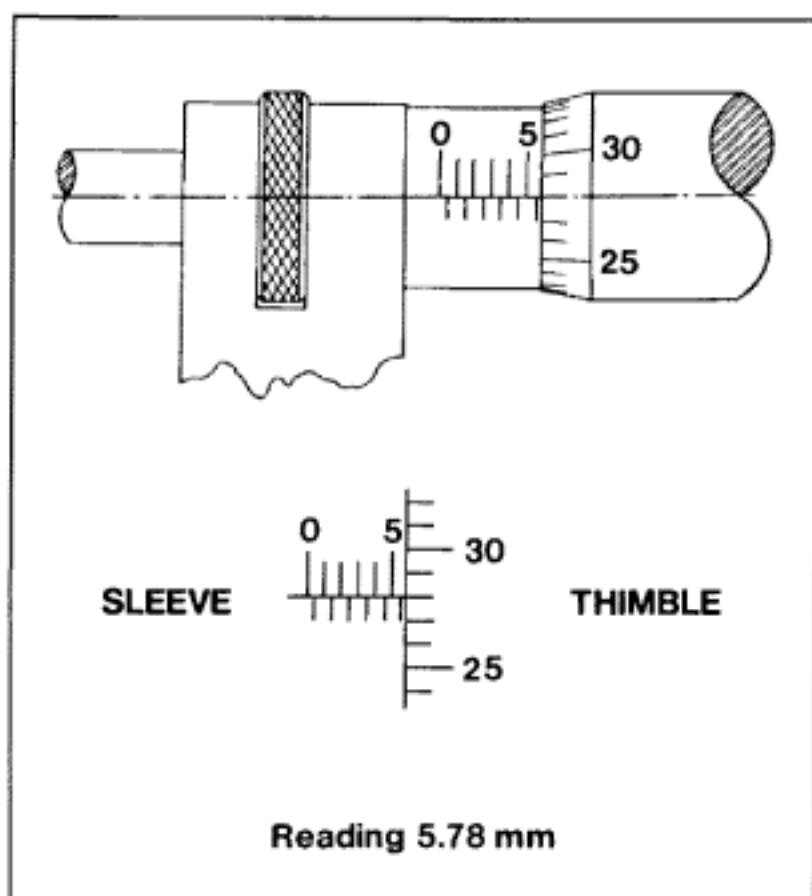


Courtesy of The L. S. Starrett Company

Fig. 2-33. Metric 0-25 mm micrometer calipers. A. Without vernier scale; graduated to read to 0.01 mm. B. With vernier scale; graduated to read to 0.002 mm.

and half-millimeters visible on the sleeve to the number of hundredths of a millimeter indicated by the thimble graduation coinciding with the reading line on the sleeve. To illustrate this procedure refer to the reading on the micrometer scale in Fig. 2-34 and follow the steps below.

1. Read the large number visible on the reading line and express as whole millimeters. 5.00
2. Count the number of graduations visible above and below the reading line that are visible beyond the large number and multiply by 0.5, or 1×0.5 .50



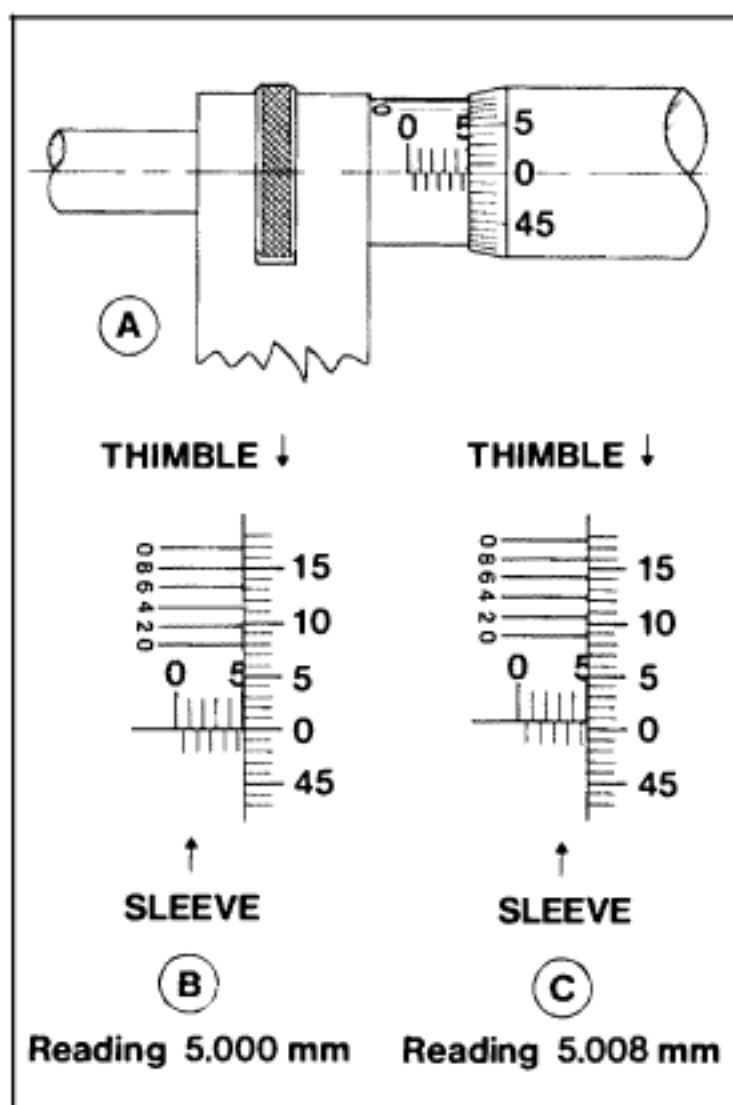
Courtesy of The L. S. Starrett Company

Fig. 2-34. Reading a metric micrometer scale graduated to read to 0.01 mm.

3. Read the graduation on the thimble coinciding with the reading line and multiply by 0.01, or 28×0.01 .28
4. Add to obtain the reading. 5.78 mm

Metric vernier micrometers (Fig. 2-35) are used like those graduated in hundredths of a millimeter (0.01 mm) except that an additional reading of two thousandths of a millimeter (0.002 mm) is obtained from the vernier scale. To read a vernier micrometer, first read it as before to obtain a reading to the nearest 0.01 mm. Then find the graduation on the vernier scale that coincides with a line on the thimble scale. If this line is marked 2, add 0.002 mm to the reading; if it is marked 4, add 0.004 mm, etc. In view B, Fig. 2-35, the micrometer reading is exactly 5 mm. The reading in view C is obtained as follows:

1. Read the large numbers visible on the reading line. 5.000
2. Count the graduations beyond the large number and multiply by 0.5. .000



Courtesy of The L. S. Starrett Company

Fig. 2-35. Reading a metric vernier micrometer graduated to read to 0.002 mm.

3. Read the graduation on the thimble coinciding with the reading line. In this case the thimble has not moved to the first line beyond 0. .000
4. Read the line on the vernier scale coinciding with a graduation on the thimble .008
5. Add to obtain the answer. 5.008 mm.

Metric Vernier Measuring Instruments

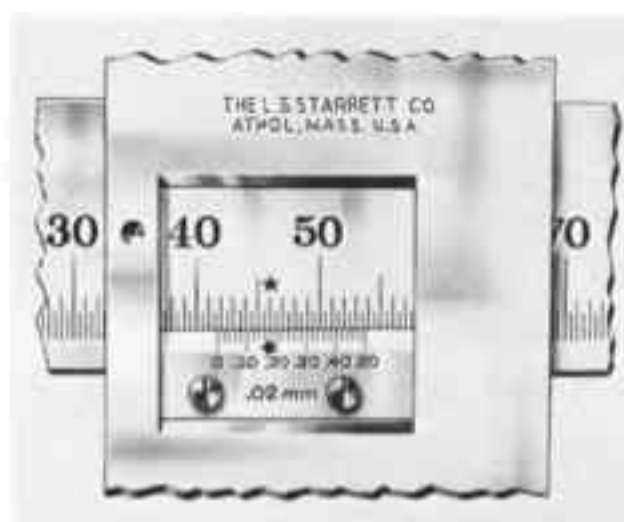
Metric vernier height gages, vernier calipers, and other vernier measuring instruments read to an accuracy of 0.02 mm, which is equal to .00079 inch. Like inch reading verniers, there are two types of metric vernier

scales, 25-division and 50-division scales. Some vernier measuring instruments have two sets of graduations, one in inch units and the other in millimeters.

A 25-division vernier is shown in Fig. 2-36. The primary scale is graduated in 0.50 mm increments, and every twentieth graduation is numbered 10 mm, 20 mm, 30 mm, etc. The vernier scale is graduated into 25 graduations, each representing 0.02 mm. Every fifth vernier graduation is numbered 0.10 mm, 0.20 mm, 0.30 mm, 0.40 mm, and 0.50 mm. To read a 25-division metric vernier, follow the steps below and refer to Fig. 2-36.

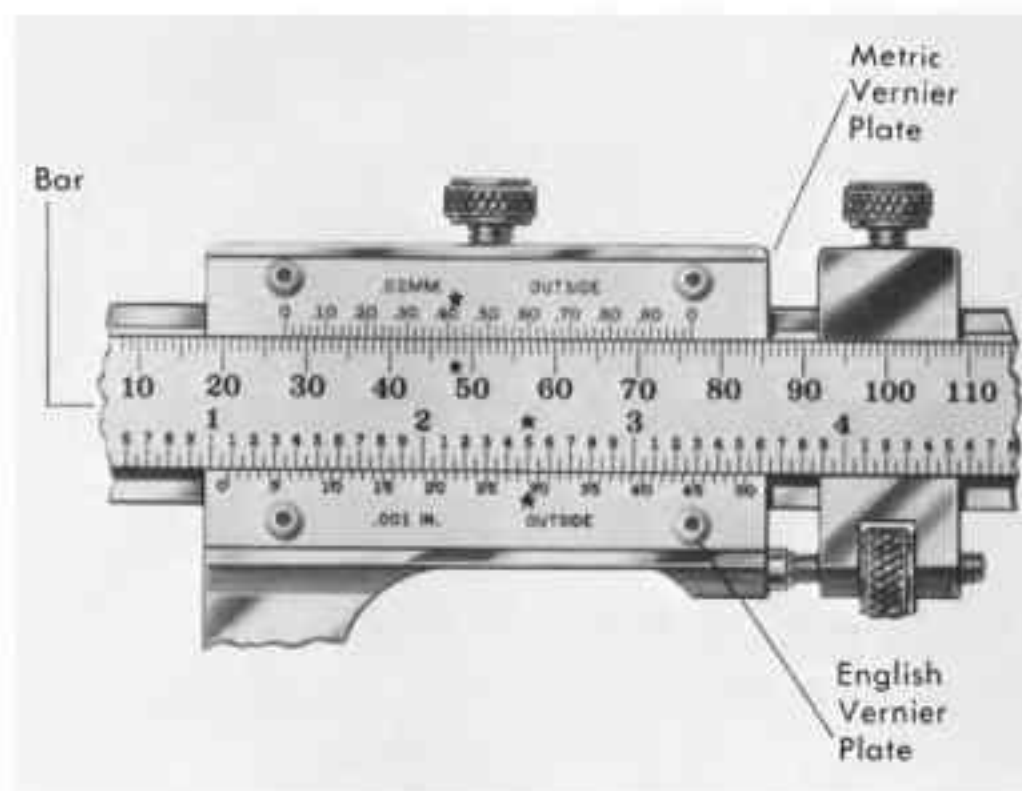
1. Read the numbered graduation on the primary scale. 40.00
2. Count the number of graduations between the numbered graduation on the primary scale and the zero graduation on the vernier scale, and multiply by 0.50; 3×0.50 1.50
3. Find the graduation on the vernier scale that coincides with a graduation on the primary scale; then read the vernier scale. Each vernier graduation represents 0.02 mm. In the illustration, the vernier scale reading is 0.18 mm, as indicated by the stars. .18
4. Add together to obtain the reading. 41.68 mm

The vernier shown in Fig. 2-37 has two sets of graduations, one in inch units and the other in millimeters. Each vernier scale has 50 divisions. The primary metric scale is graduated in 1.00 mm increments, with every tenth graduation numbered in sequence 10 mm, 20 mm, 30 mm, etc.



Courtesy of The L. S. Starrett Company

Fig. 2-36. Metric 25-division vernier scale.



Courtesy of The L. S. Starrett Company

Fig. 2-37. Vernier scales graduated to read in metric (above), and inch (below), units. Both scales are 50-division vernier scales.

The metric vernier scale has 50 graduations, each representing 0.02 mm. Every fifth graduation is numbered in sequence 0.10 mm, 0.20 mm, . . . 0.80 mm, 0.90 mm, and back to 0. To read a 50-division metric vernier, follow the steps below and refer to Fig. 2-37.

1. Read the numbered graduation on the primary metric scale. 20.00
2. Count the number of graduations between the numbered graduation on the primary scale and the zero graduation on the vernier scale. Each graduation is one millimeter. 7.00
3. Find the graduation on the vernier scale that coincides with a graduation on the primary scale; then read the vernier scale. Each vernier graduation represents 0.02 mm. In the illustration the vernier scale reading is 0.42 mm, as indicated by the stars. .42
4. Add together to obtain the answer. 27.42 mm

Converting 27.42 mm to inch units gives 1.079 inches, which is the reading on the inch scale in Fig. 2-37.

Miscellaneous Tools

A perusal through a toolmakers' catalog will show that there are other tools useful to machinists and toolmakers. It is not possible to catalog all of these tools in this volume. Many of these tools would be used primarily only on a certain type of work; e.g., die makers' squares, deep throat micrometers, etc. Other tools have a more universal application, such as prick punches, center punches, beam trammels, radius gages, metric thread pitch gages, toolmakers' parallel clamps, V-blocks, etc. The application of many of these tools will be illustrated in other chapters in this book.

The Basic Nomenclature of Measurement

The nomenclature of measurement deals with the definitions of the terms encountered in the field of measurement. The topics in the remainder of this book will be more meaningful after a study of these terms.

Nominal Size is the ideal size to which a part, or two mating parts, is to be made. It does not take into account the clearance, or interference necessary in fitting two parts together, so that they will function as desired. Nor does it take into account the deviations in size resulting from the manufacturing process.

Allowance is the intentional clearance between two mating parts so that they will work together as desired when they are assembled. In the case of a press, or shrink fit, allowance is the intentional interference between two parts.

Limits are the maximum and the minimum permissible dimensions of the size of a part. For example, a bore may be specified to have a diameter between 6.750 and 6.756 inches, in which case 6.750 and 6.756 inches are the limits.

Upper Limit specifies the largest size to which a dimension on a part is to be made. In the example above, 6.756 inches is the upper limit.

Lower Limit specifies the smallest size to which a dimension on a part is to be made. In the above example, 6.750 inches is the lower limit.

Tolerance is the specification of the total permissible variation of size of a dimension on a part. The total tolerance in the above example is .006 inch. In some instances this is expressed in terms of plus or minus from a given basic size (basic size is described below). For example, the tolerance in the example above might also be expressed as $\pm .003$.

Basic Size is that size from which the limits of a part are obtained by the application of tolerances. It is sometimes considered as the "ideal" size of the part. For example, if the ideal size or the basic size of a bore is 6.753 inches and the tolerance is $\pm .003$, the limits of the hole would be 6.750 and 6.756 inches.

Unilateral Tolerance refers to a permissible deviation in the basic size in one direction only; i.e., this specification states that the part is to be made with the basic size as the lower limit, or with the basic size as the upper limit. An example of a unilateral tolerance would be $6.750 + .003$. Another example would be $6.775 - .002$.

Bilateral Tolerance refers to a permissible deviation from the basic size in two directions; i.e., this specification would state that the basic size is neither the upper nor the lower limit and that the part may be slightly larger or slightly smaller than the basic size. Bilateral tolerances may be equal or unequal. An example of an equal bilateral tolerance is $6.753 \pm .003$ inches. An example of an unequal bilateral tolerance would be $6.752 \begin{smallmatrix} +.002 \\ -.001 \end{smallmatrix}$ inches.

Precision refers to the degree of dimensional refinement to which a part is made or to which a machine is capable of working. A part which is made to, or a machine which will produce parts, with a very small dimensional tolerance is said to be precise.

Accuracy refers to the confirmation of a part to the requirements specified by the dimensional tolerance. A part that is within the required dimensional tolerance is accurate. If the required tolerance is very small, say $\pm .001$ inch, the part is also precise. If the tolerance on a machined part is large, say, $\pm \frac{1}{16}$ inch, a part is accurate if it falls within this range, yet, at the same time, it would be considered to be imprecise.

Layout Work

Castings, forgings, weldments, and rolled sections are laid out in order to establish reference lines or planes which are used to set up the part on the machine, to outline the surfaces that are to be machined, and to show the approximate amount of metal to be removed from each surface. Castings and forgings in particular, have excess material on all surfaces which are to be machined, including cored holes. One purpose of the layout is to determine if enough metal exists on all of these surfaces to "clean up" or "machine out." Sometimes, a core will shift in making a casting, resulting in insufficient metal to "machine out." When the core shift is only a small amount this can be corrected in the layout; at other times, the casting must be scrapped. However, it is best to know that the casting will be scrap *before* any expense is encountered in machining.

Two other reasons for laying out workpieces that are to be machined are to locate and outline the position of holes that are to be drilled and to outline keyways that are to be cut. Although castings, forgings, and weldments are usually laid out before machining any surface, additional layout work is often done after one or more machining operations have been finished. In some cases, usually involving rolled material, the initial layout operation is performed after several machining operations are completed. Not all workpieces are laid out. Parts made from bar stock and those parts having a relatively simple shape are often machined directly, without making a layout.

The layout is usually made from a reference surface which is a plane. Surface plates, layout plates, layout tables, floor plates, and the top of machine-tool tables are used in making layouts. They must be kept clean and free from scratches, nicks, and burrs that would impair the accuracy of the reference surface and the layout work done upon it. Other tools used to do layout work include precision steel rules, layout rules, squares, bevel protractors, surface gages, height gages, scribes, hermaphrodite calipers, dividers, prick punches, center punches, hammers, V-blocks, parallels, angle plates, and a variety of clamps. Layout machines (see Figs. 3-16 and 3-17) have been developed that greatly facilitate the layout operation.

Painting for Layout

The first step in making any layout is to paint the surface. Surfaces that have a smooth finish are usually painted with a blue layout dye.

Brushed onto a clean surface it dries rapidly, providing a blue background that makes scribed lines stand out sharply and clearly. The surfaces of rough castings and forgings are usually painted with a specially prepared white paint. This paint is composed of a suitable white pigment mixed with alcohol, which evaporates rapidly upon application, enabling the paint to dry in a short time. This mixture should be thin so that it will spread easily on the metal surfaces. White blackboard chalk can be used when no paint is available; however, it is not permanent and is not a good substitute for layout paint or dye.

Layout of Hole Location for Drilling

The procedure for laying out holes that are to be drilled in a thick plate is shown in Fig. 3-1, A through F. The dimensions of the hole locations are given in F. The surface finish of the plate is smooth; free from nicks and burrs. After painting the surface that is to be laid out, the workpiece is clamped to the angle plate with one edge resting on the layout table as in A. For clarity, the clamps are not shown. Using a precision steel rule that is held in an upright position, the surface gage, shown in A, is set so that the scriber point is exactly 5 inches ($1\frac{3}{4}" + 3\frac{1}{4}"$) above the lower edge of the workpiece. With this setting the surface gage is then used to scribe a horizontal line as shown in B. In a similar manner, a second horizontal line is scribed $1\frac{3}{4}$ inches above the lower edge of the workpiece. With a steel rule positioned as shown in C, the

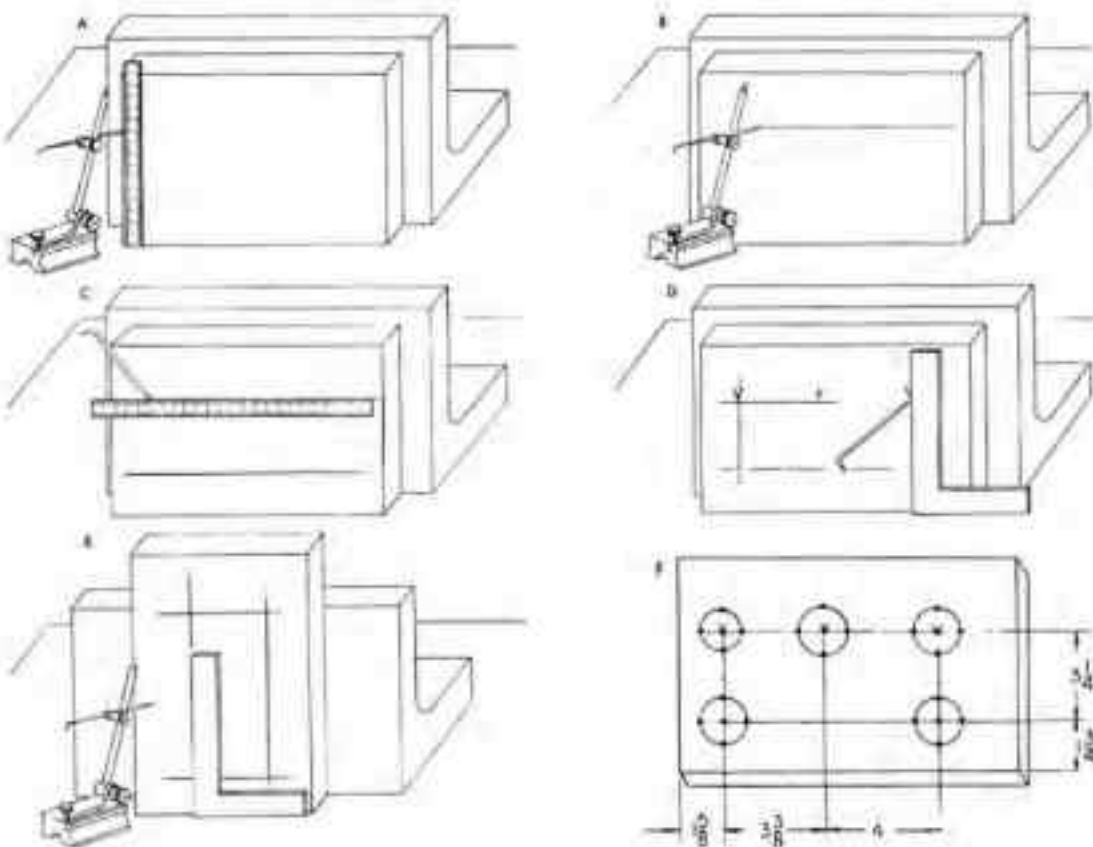


Fig. 3-1. Steps in laying out holes for drilling.

$1\frac{5}{8}$ -inch, $3\frac{3}{8}$ -inch, and the 4-inch distances are laid out with a scribe. The location of each hole is marked by two scribed lines which come together to a point. The two lower holes are laid off by scribing two vertical lines with a scribe and a square, as shown at D. An alternate method is illustrated in E, where the workpiece is turned 90 degrees and reclamped on the angle plate. The workpiece is correctly positioned by aligning the scribed line with the blade of a square. The $1\frac{5}{8}$ -inch, $3\frac{3}{8}$ -inch, and 4-inch dimensions are then laid out with a surface gage.

A prick punch is carefully positioned at the center of each hole, as determined by the layout lines, and a mark is made by tapping the punch with a hammer. Using a rule, dividers are then set to the radius of the hole to be drilled, and a circle, equal in diameter to the hole diameter, is scribed at each hole location. (See Fig. 3-2.) The center of each hole is punched again with a center punch and a hammer, and four "witness marks" are punched on the scribed circles as shown in F, Fig. 3-1. The purpose of the witness marks is to provide points of observation when starting to drill the hole with the point of the twist drill. The scribed lines can be rubbed off by the chips but the witness marks will remain. The circle and witness marks are usually omitted for holes that are approximately $\frac{1}{4}$ inch, or less, in diameter.

There are other procedures that can be used to lay out holes, two of which are shown in Fig. 3-3. At the left, a combination square and rule is used, while at the right, the layout is made with hermaphrodite calipers. Often, combinations of all of these procedures are used to lay out a part.

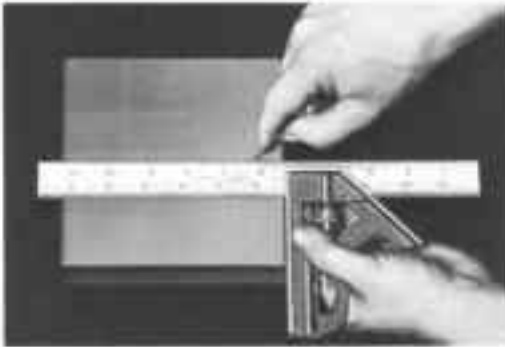


Courtesy of The L. S. Starrett Company

Fig. 3-2. Scribing hole diameters on a hole layout using spring dividers.

Layout of Cylindrical Part

Layout operations on cylindrical parts usually require that the point of the surface-gage scribe be set on the exact center of the workpiece. The procedure for doing this is shown in Fig. 3-4, A through D. This procedure is preliminary to finding the exact center of the end faces of the workpiece, prior to center drilling the ends in preparation for turning the part between centers in a lathe. (See D.) It is also required when holes and keyways are to be laid out, as shown in E and F.



Courtesy of The L. S. Starrett Company

Fig. 3-3. (Left) Using combination square and scribe to lay out plate. (Right) Using hermaphrodite calipers to lay out plate.

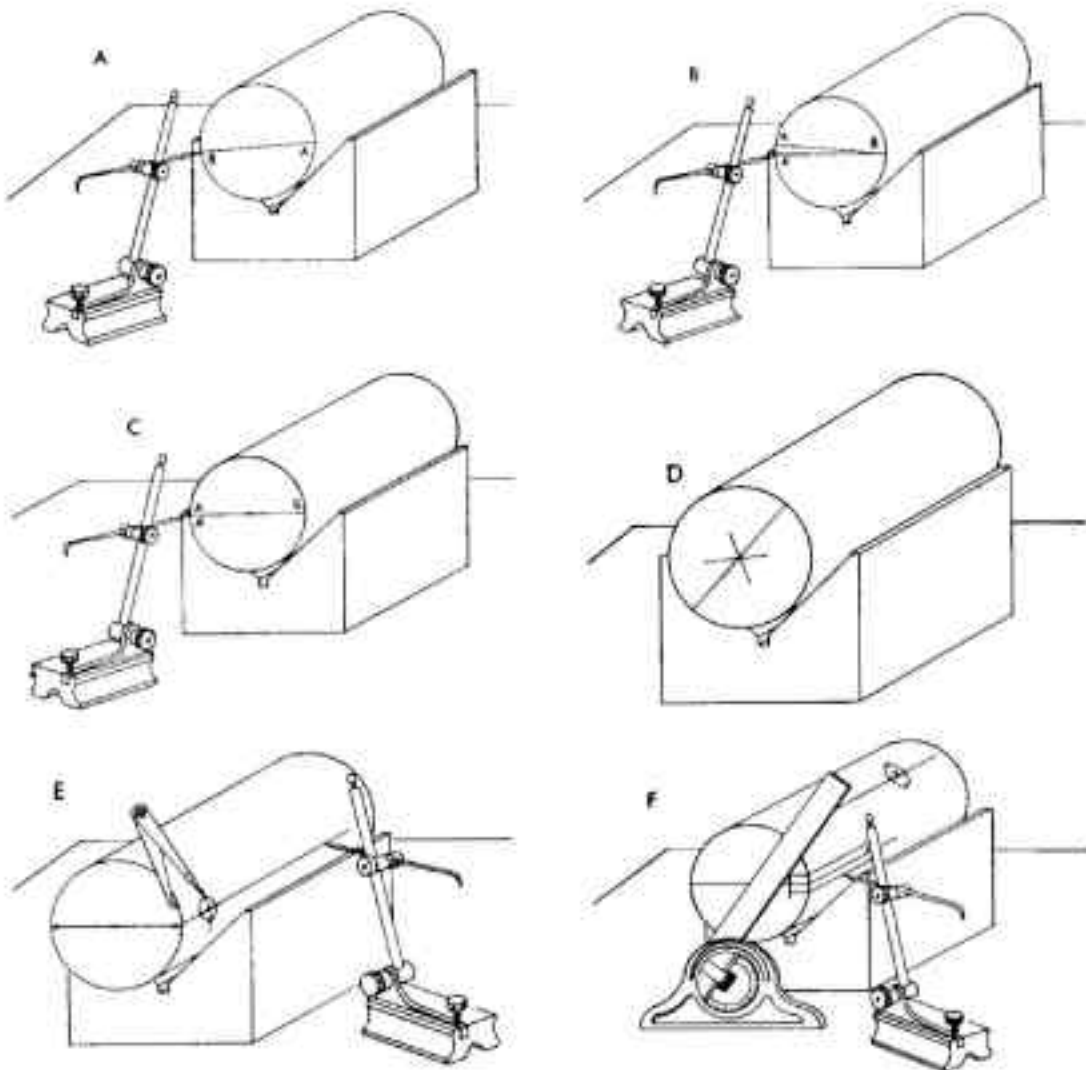


Fig. 3-4. Laying out cylindrical work; A through C. Method of setting surface-gage scribe on centerline of work axis. D. Laying center of end face. E. Laying out for drilling hole perpendicular to axis. F. Laying out for drilling holes perpendicular to axis and at an angle to each other.

The part is first painted in the areas to be laid out. After the paint has dried, the part is placed in a V-block which is standing on a layout table. The procedure for laying out the cylindrical part for the different configurations mentioned above will be described in the following steps, and by referring to the diagrams in Fig. 3-4.

1. Adjust the position of the surface gage scribe to the estimated center of the work.
2. Scribe a line from *A* to *B*. (See diagram A, Fig. 3-4.)
3. Rotate the work approximately 180 degrees or one-half turn, until the *B* end of line *AB* is aligned with the surface-gage scribe.
4. Scribe line *BA'*. (See diagram B, Fig. 3-4.)
Note: *A'B* may come out above *AB*, instead of below *AB*, as shown.
5. Adjust the surface-gage scribe one-fourth the distance from *A'* to *A*. (If *A'B* coincides with *AB* as in diagram C, Fig. 3-4, the surface-gage scribe is set at the exact center of the work, thus steps 5 and 6 are not required.)
6. Repeat steps 2 through 5 until *AB* coincides with *A'B* as shown in diagram C. Two or three trials should be all that are normally required to obtain this setting of the surface-gage scribe.
- 7a. To locate the center of the end face for drill and countersinking in preparation for turning between centers in a lathe, scribe two additional lines with the surface gage as shown in D, Fig. 3-4. The intersection of any two of these lines will be the exact center. The third line serves as a check on the accuracy of the work. If all three lines intersect at the same point, the work is accurate.
- 7b. The layout at E, Fig. 3-4 is for a hole that is to be drilled both perpendicular to, and through, the axis of the work. The scribed line is extended along the cylindrical surface of the work with the surface gage scribe set "on center." The distance from the end face to the center of the hole is laid out with hermaphrodite calipers, as shown, or with a scribe and a rule. This location is prick punched, and a circle, equal to the hole diameter, is scribed with dividers. The center of the hole is center punched and witness marks are punched around the circle. Often, two very light prick punch marks are made on the scribed line on the end face, as shown, which serves as a guide when setting up the part on the drill press.
- 7c. The procedure for laying out a keyway at an angle to a hole is illustrated in F, Fig. 3-4. The hole is laid out as explained in step 7b. A bevel protractor is then set to do the required angle. The scribed line on the end face corresponding to the hole is then aligned with the blade of the protractor, as shown. Thus, the work is positioned in the V-block for laying out the keyway. A

line is scribed across the end face and along the cylindrical portion of the work with the surface gage set "on center." Using a rule, the scribe of the surface gage is set above center; a distance equal to one-half of the width of the keyway. A line is then scribed with the surface gage as shown in Fig. 3-4, F. Similarly, a line is scribed below the center to outline the sides of the keyway. The depth of the keyway is marked, and using a scribe and a square, a vertical line is scribed on the end face. The length of the keyway is laid out with a scribe and a rule. A scribe mark and a prick punch mark are then made at a distance equal to one-half of the width of the keyway, from the end. Using dividers set in the prick-punch mark, a half circle is then scribed at the end of the keyway layout to complete the job.

Procedures other than those described in step 7a can also be used to locate the center of the end face. When the cylindrical surface is rough, the use of the hermaphrodite calipers will tend to give more accurate results than do other methods as the caliper leg will contact a greater length of the cylindrical surface as it is moved along, thereby averaging out small irregularities. The method of using hermaphrodite calipers is shown in A, Fig. 3-5. When the cylindrical surface is fairly true, a center head, shown in Fig. 3-5, B, is a fast and accurate method of performing this operation. Two or three lines are scribed with the blade held in different positions on the work. The intersection of these lines is the center of the end face. A center punch mark *C*, is made to mark the center. In Fig. 3-5, D, a bell center is shown. The conical surface locates the punch at the center of the end face without the necessity of making a layout. The punch is struck with a hammer, to mark the center. Bell centers should only be used when the workpiece is fairly round.

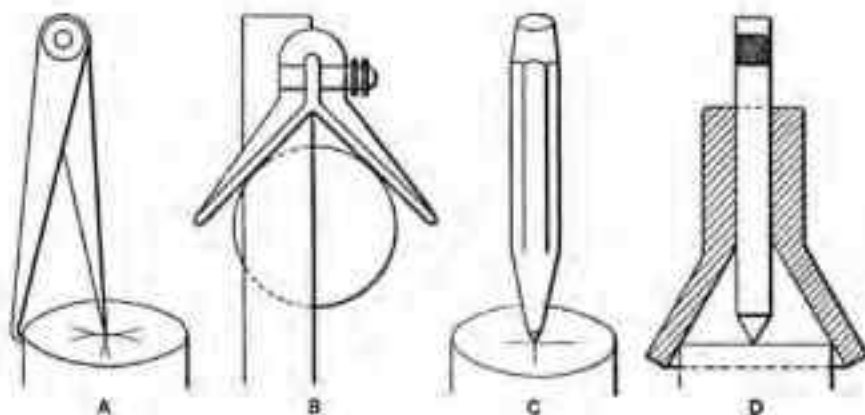


Fig. 3-5. Laying out center, on end face of cylindrical work. A. Using hermaphrodite calipers; B. Using center head; C. Using center punch; D. Using bell center.

Layout of Equally Spaced Holes on Hole Circle

A form of construction frequently encountered is the placement of holes in a circular pattern, as shown in Fig. 3-6. The circle on which the

hole is located is called a *hole circle*. The center of the hole circle is often located in the center of a larger central hole. In such a case, it will be necessary to press a wooden plug or stick (see Fig. 3-6) into the central hole. Prior to pressing the plug or the stick into the hole, a large-head nail is driven into the center of the stick. The center of the central hole can now be found with hermaphrodite calipers, or by using a surface gage and a rule. A light prick-punch mark made at the intersection of the scribed lines on the nail head locates the center of the central hole. This prick-punch mark anchors one leg of the dividers, or trammel points, while the other leg is swung in an arc to scribe the hole circle.

The next operation is to set the dividers to a distance D (see Fig. 3-6), and to scribe the center of each hole. Usually, the position of one of the holes is defined by the drawing; however, in some cases, no starting location is given, as in the case of the ring in Fig. 3-6. In either case, a prick-punch mark is made at the center of the first hole. The dividers, set at the distance D , are then positioned in this prick-punch mark and a short arc is scribed across the hole circle. A prick-punch mark is made at the intersection of the arc and the hole circle. This procedure is repeated until the centers of all of the holes have been located. To complete the layout, circles are scribed with dividers and witness marks are punched.

One of the problems that can be time consuming is the determination of the distance D , at which the dividers must be set to obtain equally spaced holes around the hole circle. A unique situation arises when the number of equally spaced holes is six. In this case, the distance D is equal to the radius of the hole circle. When there are relatively few holes equally spaced around the hole, circle D can be determined by trial-and-error. When there are many holes, however, the trial-and-error method is slow

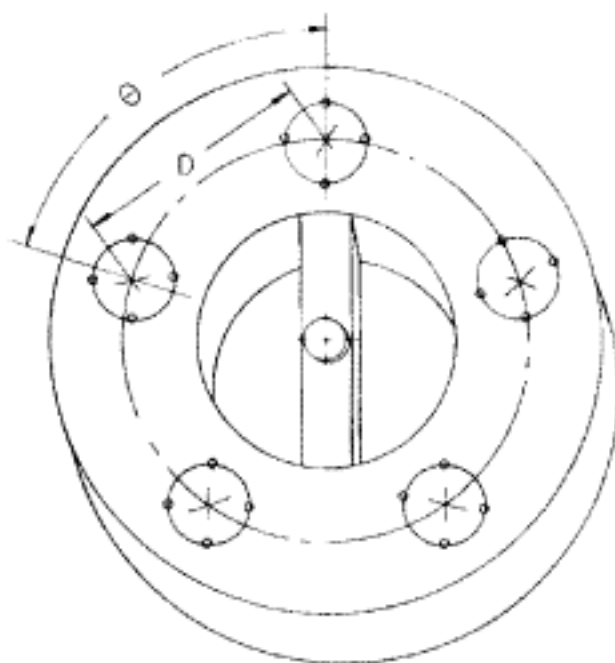


Fig. 3-6. Laying out holes on bolt circle.

and inaccurate. A better way is to calculate D , as explained in the two steps below (see Fig. 3-6):¹

1. Calculate the included angle θ , which is the angle between two adjacent holes on the hole circle. This is done by dividing the number of holes in the hole circle into 360° or

$$\theta = \frac{360^\circ}{\text{No. holes in hole circle}}$$

2. Calculate D by multiplying the diameter of the hole circle by the sine of one-half of the angle θ .²

$$D = \text{Diameter of hole circle} \times \sin \frac{\theta}{2}$$

Example 3-1:

Twenty equally spaced holes are to be placed around a 10-inch diameter hole circle. Calculate the distance between each adjacent hole.

$$\begin{aligned} 1. \theta &= \frac{360^\circ}{\text{No. holes in hole circle}} \\ &= \frac{360^\circ}{20} \\ &= 18^\circ \end{aligned}$$

2. (From a table of trigonometric functions, $\sin 9^\circ = .15643$)

$$\begin{aligned} D &= \text{Diameter of hole circle} \times \sin \frac{\theta}{2} \\ &= 10 \times \sin \frac{18^\circ}{2} \\ &= 10 \times \sin 9^\circ \\ &= 10 \times .15643 \\ &= 1.5643 \text{ inches} \end{aligned}$$

This is approximately $1\frac{9}{16}$ inches, since the decimal equivalent for $\frac{9}{16}$ inch is .5625 inch.

Layout of Large Castings

Large castings vary so greatly that the layout procedure to be described must be rather general. There are, however, certain specific steps to be taken when laying out any large casting which, when completed, accomplish a desired objective. These steps, given below, can serve as a guide for laying out any large casting:

¹ A table of chord lengths for spacing off circles for a wide range of hole numbers is given in *Machinery's Handbook*, Industrial Press Inc.

² The sine can be found in any table of natural trigonometric functions that are given in various handbooks.

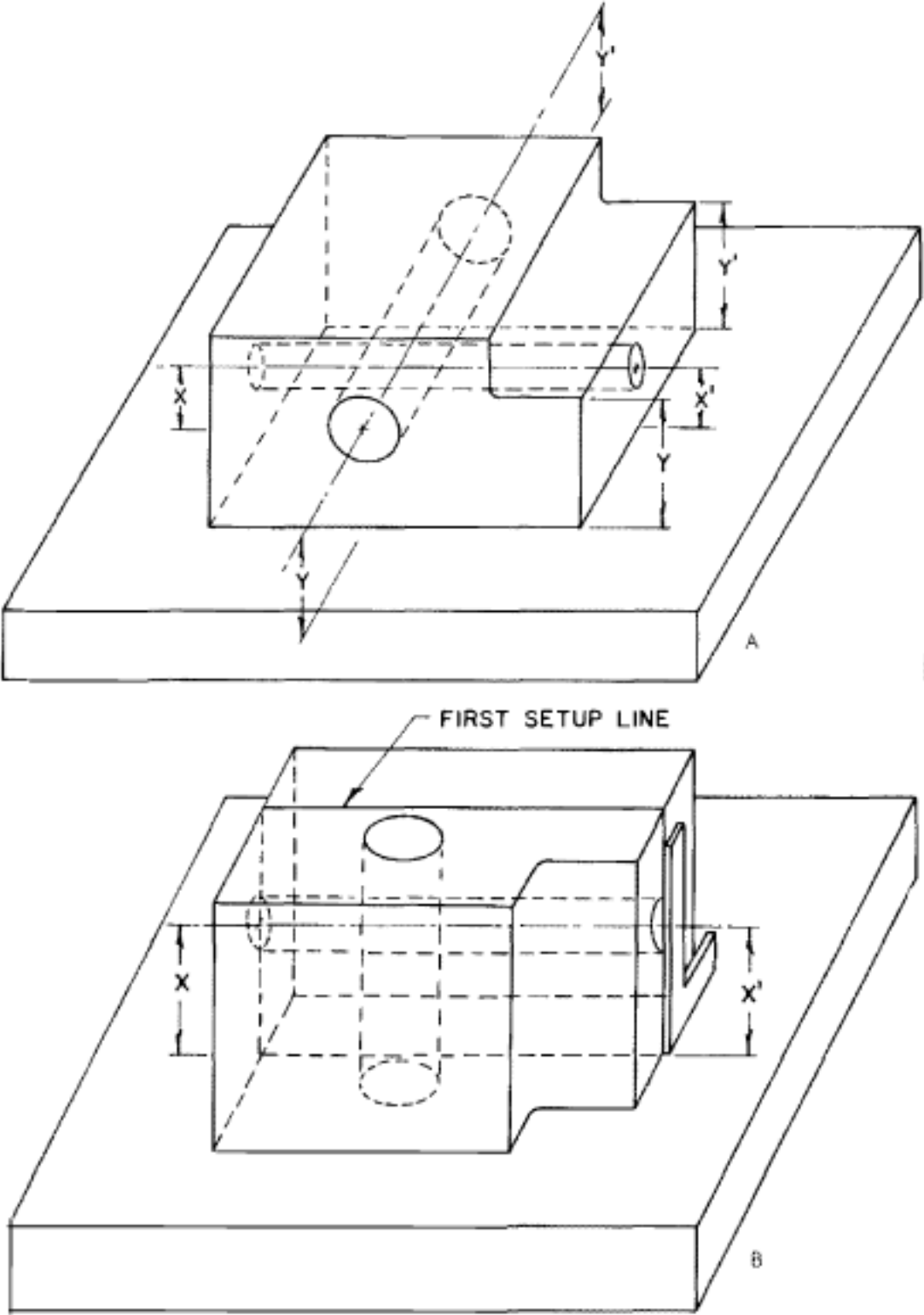


Fig. 3-7. A. Principle of aligning a casting on a layout table for the first setting. Either set of YY' distances can be used, but not both. B. Principle of aligning casting for second setting.

1. Place the work on parallel blocks that have been previously placed in position upon the layout table. Since the surface of the casting is rough, it is usually necessary to add shims to prevent the casting from rocking. Screw jacks are used, sometimes, in place of parallel blocks and shims.
2. Paint the casting with a white layout paint on all surfaces on which layout lines are to be scribed.
3. Establish a reference plane by selecting and leveling two horizontal lines, or planes, that are at right angles to each other on the casting. This is the initial step in actually making the layout. The principle of establishing a reference plane is illustrated in A, Fig. 3-7. A block having two perpendicular axes is shown. Each axis is parallel to the top and the bottom surfaces, or planes, on the block. The pair of axes may or may not actually intersect each other. The top and bottom surfaces, or planes, of the block will be parallel to the top of the layout table when the distances Y and Y' , taken from the top of the table, are equal to each other and when the distances X and X' are equal to each other. It is not necessary for Y and Y' to be equal to X and X' .

In order to establish the reference plane on the casting, a starting point must be selected. Sometimes, a flat surface can be used as a reference plane, in which case this surface is leveled with a surface gage or placed in a perpendicular position with a square. Care should be exercised in selecting plane surfaces as reference planes because they often are made to have "draft," an intentional taper which is provided to enable the pattern to be removed from the mold in the foundry. Cored holes seldom are made with draft.



Fig. 3-8. Checking starting reference point of layout with a surface-gage and a ruler.

The axis of a cored hole, therefore, frequently affords a better reference position than does a plane surface. The axes of two perpendicular cored holes also are frequently used to establish a reference plane on a casting. Figure 3-8 shows the center of a cored hole being checked with a rule and a surface gage. This cored hole passes through the casting and the axis of the cored hole is selected to establish the reference plane. The casting is leveled until the cored hole axis is parallel to the layout table; this is accomplished by making the distance from the table top to the center of the cored hole, equal at each end. If necessary, shims are used to adjust the position of the casting. A similar procedure is used to find, and then to level the second axis, which is perpendicular to the first axis. When adjusting the casting to bring the second axis parallel to the layout table, raise and shim two corners of the casting an equal amount. This is necessary in order to maintain the parallelism of the first axis.

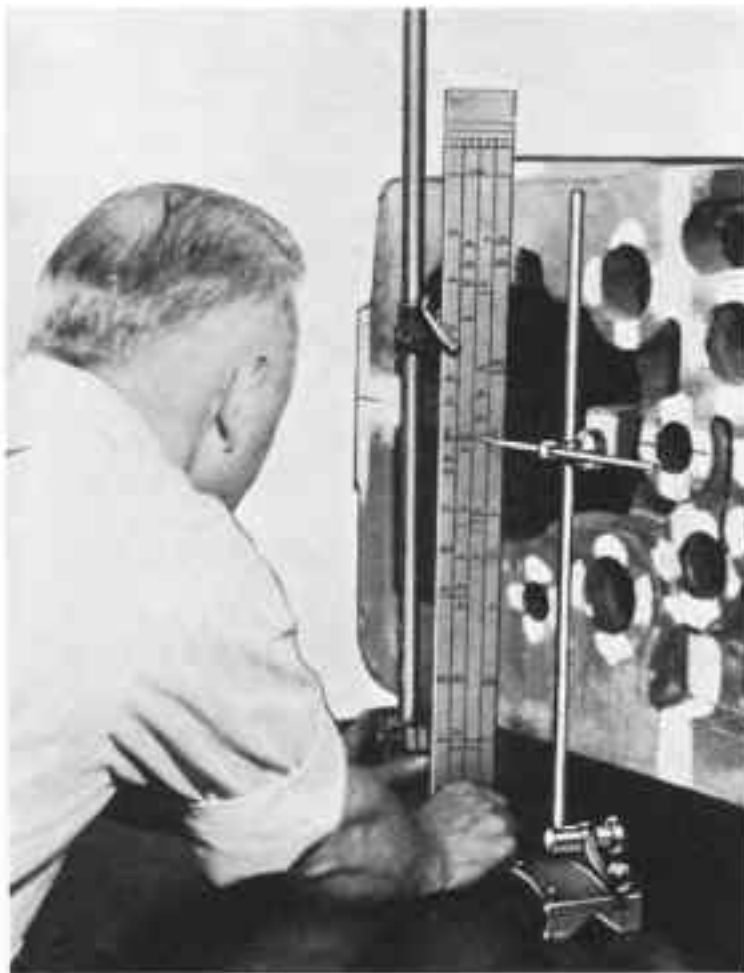
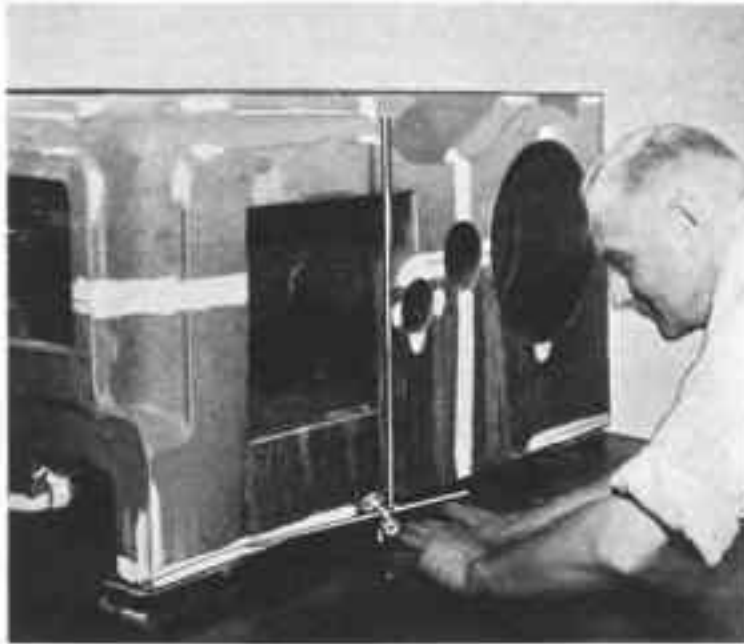


Fig. 3-9. Setting the layout scale to the starting reference point.



Courtesy of the Giddings & Lewis Machine Tool Company

Fig. 3-10. Scribing a finish line for a surface to be milled with a surface gage.

4. Position the rule, or if used, the layout scale, with respect to the casting. A layout scale, Fig. 3-9, has graduations which have been specially cut to mark the dimensions that are to be laid out on a particular casting model. When quantities of the same casting models are to be made, the layout scale will pay for itself by reducing the time required to make the measurement and by reducing errors in reading the rule. When only one or two castings are to be laid out, a precision steel rule is used. In either case, the layout scale, or the rule, is adjusted vertically on the rule holder until one of its principal graduations (say, an inch mark) is exactly aligned with the point of the surface-gage scribe, with the scribe positioned on the reference axis. In Fig. 3-9, two lines are shown scribed across the cored hole containing the reference axis; the layout scale is being positioned with respect to this axis, using the surface gage.
5. Check the amount of metal available for machining on all surfaces that are to be machined. The purpose of this step is to determine if enough metal exists on each surface in order to "machine out" or "clean up." The dimension of each surface is checked with the surface gage, then a line is scribed to mark the location of the surface to be machined. Figure 3-10 shows a finish line being scribed on a casting, to define a surface that is to be milled or planed.

The amount of metal to be machined in cored holes is checked by scribing two lines, one above and one below the axis of each



Fig. 3-11. Prick punching the first setup line.

cored hole. The same cored hole is checked again in the second setup of the casting when the casting is positioned 90° from its present position, as described in Step 7. Thus, the hole will have four scribed lines as shown in Figs. 3-13, 3-14, and 3-15, which will determine if sufficient metal is available to clean up when it is machined.

The layout procedure sometimes will show that there is insufficient metal on a plane surface, or on a hole, to "clean up." In this event, the rule or the layout scale is moved up or down, as required, but far enough so that there will be sufficient metal to "clean up" the defective surface. Every other surface must now be checked again to make certain that this move will not result in the failure of some other surface to be cleaned up. Occasionally, a core will shift sufficiently in making a casting to cause the cast surfaces to be badly misaligned. In such a case the layout will indicate that it is impossible to "machine out" all surfaces. The casting must then be scrapped unless it can be salvaged by welding.

6. Scribe the first setup line around the casting. It should then be prick punched at convenient intervals, as shown in Fig. 3-11. The setup line defines a plane that is parallel to the two original mutually perpendicular axes that were selected in Step 3. This line will determine the alignment of the setup on the machine and it will also indicate a starting point from which other machined surfaces are dimensionally tied. It is sometimes convenient to have the setup line coincide with the center of the casting.
7. Establish the second reference plane. The second reference plane

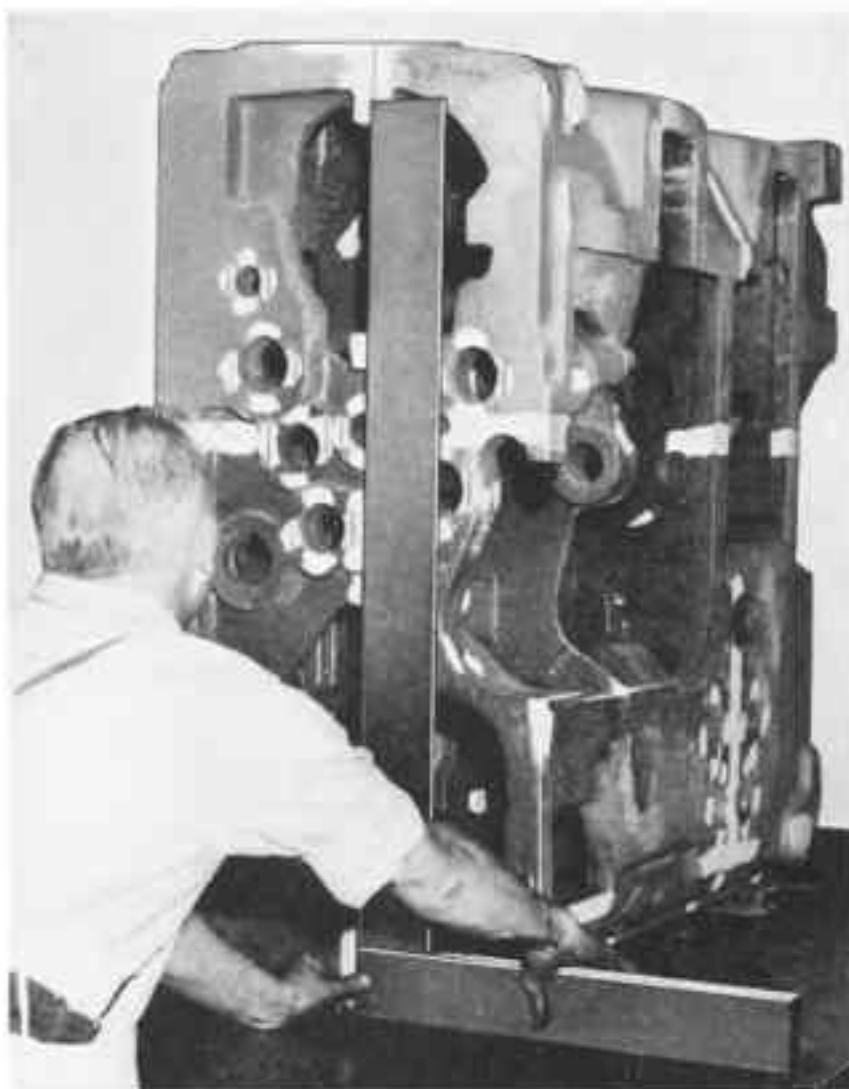


Fig. 3-12. Resetting work to first setup line using a square.

is required to check the amount of material on all surfaces for "clean up," and to scribe a second setup line which is used to align the part in the machine when setting up. The work is reset on the layout table by turning it 90° . The principle of aligning the part on the layout table for the second setting is shown in B, Fig. 3-7. The first layout line (Step 6) is aligned perpendicular to the top of the layout table, using a square, as shown in B, Fig. 3-7 and also in Fig. 3-12. The original axis (the cored hole in the casting illustrated) is then made to be parallel to the top of the layout table by making distance X equal to X' (see B, Fig. 3-7). In practice, this is accomplished with a rule and a surface gage as shown in Fig. 3-8, and described in Step 3.

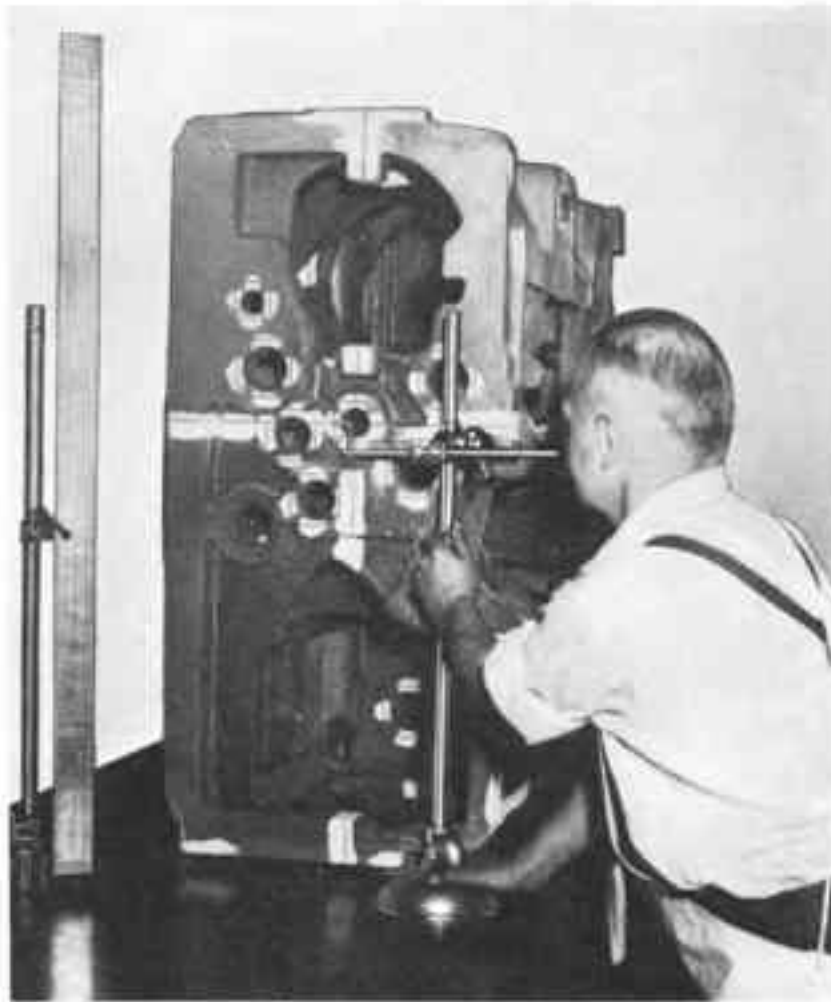


Fig. 3-13. Scribing the second setup line.

8. After the casting has been set up and aligned with the first setup line perpendicular to the table and with a reference axis positioned parallel to the table, the second part of the layout work can proceed. Each surface that requires machining is checked and the necessary layout lines are scribed by repeating Steps 4 and 5.
9. Scribe the second layout line completely around the casting, as shown in Fig. 3-13. This line will also be used in setting up the casting on the machine.

To avoid eccentric cuts with small boring bars, whenever possible the axis of a small diameter hole should be selected as a reference axis. When a small cored hole is not aligned with the axis of rotation of the boring bar, the boring tool will take a heavier cut from one side of the hole than from the other side. This unequal cutting action causes the boring bar to

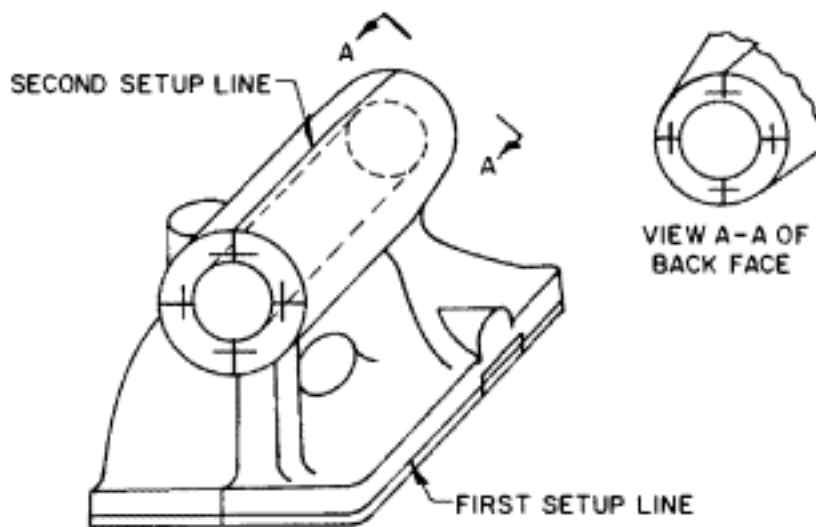


Fig. 3-14. Layout of an engine lathe tailstock. The first setup line is also a layout line for the first machining operation.

deflect more on the heavier side which, as a consequence, will produce a hole that is not in the correct location. To locate the hole correctly, several light cuts are required, which are time consuming. When the axis of the small hole is the reference axis, the layout can be made so that a uniform amount of metal is to be removed from all sides of this hole; then when the cut is taken the boring bar deflection will be uniform around the hole and the hole can be bored in location without difficulty. Eccentricity on larger holes is not quite as serious provided enough metal exists on all sides to "clean up." Larger and stiffer boring bars which do not deflect as easily are usually used to bore large holes.

Sometimes a setup line is also a layout line, as shown in the layout of an engine lathe tailstock in Fig. 3-14. When scribed around the base as shown, instead of around the spindle barrel, the first setup line presents an easier target to see when setting this casting up on the machine. Since it is longer, the setup will also be more precise. The first machining operation will be to machine the base to the setup line, which acts as a layout line. When finish machined, this surface will act as a reference surface for locating the vertical position of the hole while the second setup line is used to locate the axis of the hole.

Many machining operations are performed by adjusting the cutting tool to cut to the layout line with surprisingly precise results. It is not usually difficult to work to a tolerance of ± 0.005 inch (± 0.127 mm) by this procedure and often even a closer tolerance can be obtained. When very precise tolerance must be obtained, the layout line serves only as a guide and the fine adjustments of the machine tool and the cutting tool are used to obtain the required tolerance.

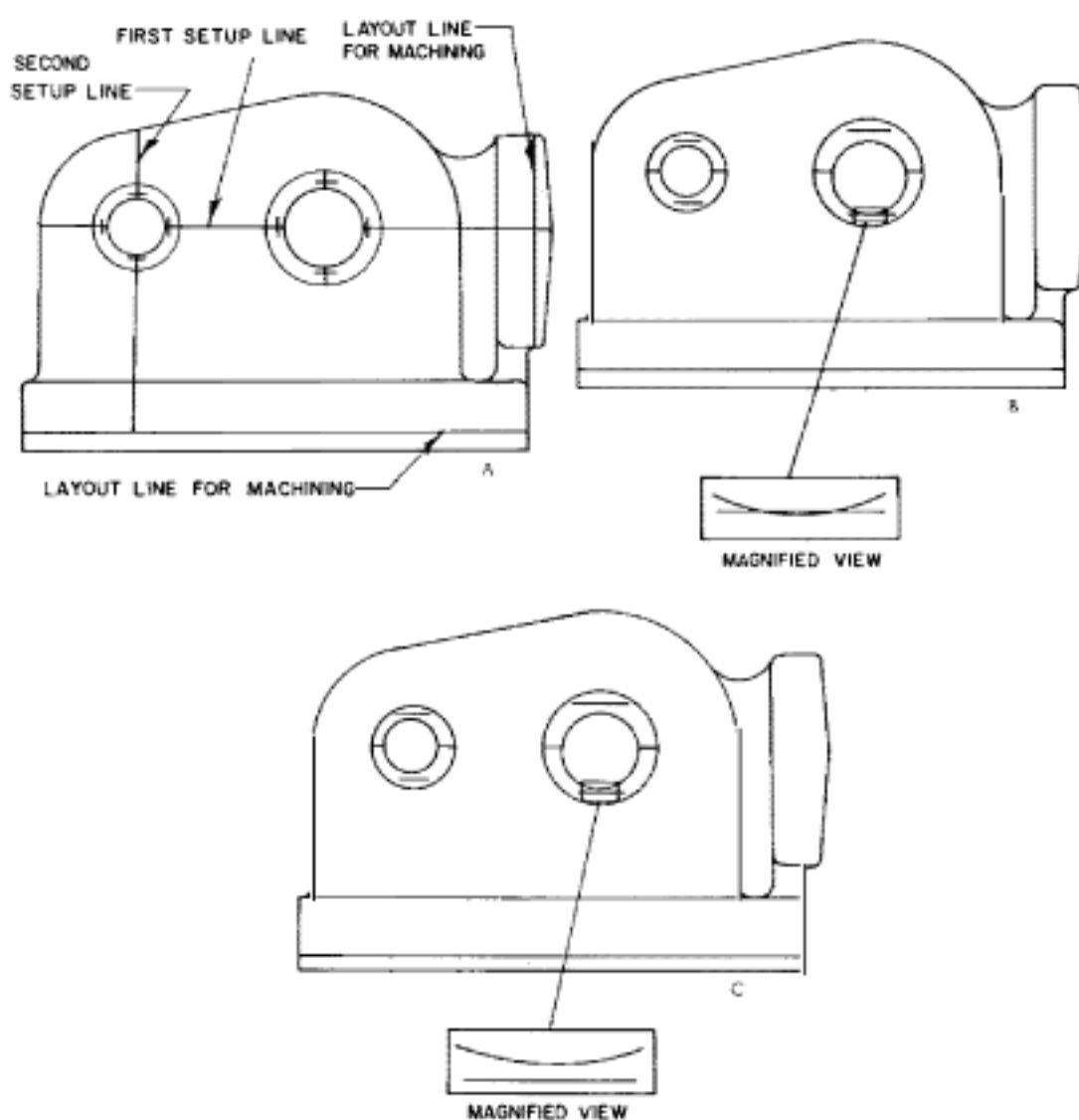
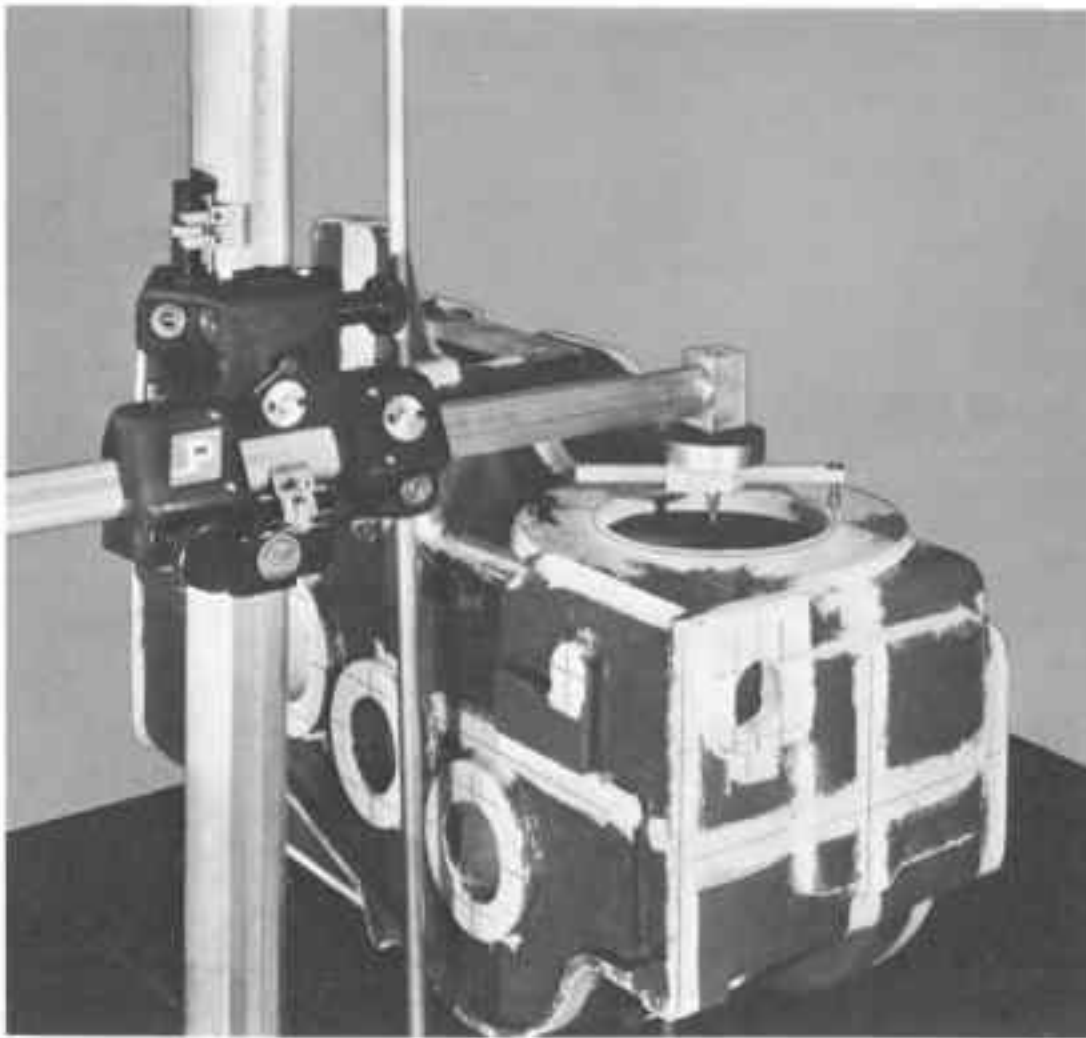


Fig. 3-15. Procedure for adjusting the layout to save a casting. A. Complete layout of a good casting that does not require an adjustment of the layout. B. Casting having a cored hole that will not "clean up," as determined by the layout. C. Adjusted layout to provide sufficient metal on all machined surfaces to "clean up."

Adjusting the Layout

As mentioned before, when the first layout shows that a particular surface will not "clean up" when machined, the casting or weldment can often be saved by adjusting the layout. The layout of the gear housing in Fig. 3-15A shows that this casting will "clean up" on all surfaces to be machined. In this instance, the centerlines of all of the holes were scribed because it was convenient to locate the scribed lines that define the finished diameters from these centerlines. This procedure is used when convenient, but not always. While requiring more work, it will decrease the chance of an error occurring.

Another gear housing casting is shown in Fig. 3-15B which will not "clean up." The core of the larger hole has dropped, leaving insufficient



Courtesy of the Bendix Automation & Measurement Div.

Fig. 3-16. Layout machine used to lay out a casting.

metal in the hole to be machined out. To save this casting the entire layout is lowered, as shown in Fig. 3-15C. All of the surfaces to be machined that are affected by this shift in the layout must be checked to see if they clean up before proceeding further with the layout. When these surfaces have been proven to have enough metal to clean up, the first setup line is scribed and the casting can be positioned to make the second part of the layout.

Layout Machines

The layout work just described can be greatly simplified, and the accuracy greatly increased, by the use of layout machines. A layout machine is illustrated in Fig. 3-16. The layout scriber can be moved



Courtesy of the Bendix Automation & Measurement Div.

Fig. 3-17. Layout machine used to lay out a cylinder block.

horizontally, or parallel to the layout table, by moving the entire machine and by sliding the horizontal arm. It can be moved vertically by moving the arm assembly up or down the column. Vernier scales attached to the column and the arm make possible accurate vertical and horizontal movements. Circles can be scribed with the circular scribing attachment shown in Fig. 3-16.

One of the great advantages of the layout machine is that the need for a second setting is generally eliminated. All surfaces can be laid out in one setting. Figure 3-17 illustrates an automobile cylinder block positioned on a rotary table and being laid out with a layout machine.

Two very typical layout jobs on larger castings are shown in Figs. 3-18 and 3-19. One great advantage of the layout machine is that both setup lines can be scribed perpendicular to each other in one setup, or setting, of the casting on the layout table. Sometimes it is also possible to scribe all of the layout lines in one setting of the casting. A second setting of the casting is required whenever layout lines must be scribed on the surface, or surfaces, of the casting facing the layout table during the first setting,



Courtesy of the Bendix Automation & Measurement Div.

Fig. 3-18. Laying out a large casting with a layout machine.

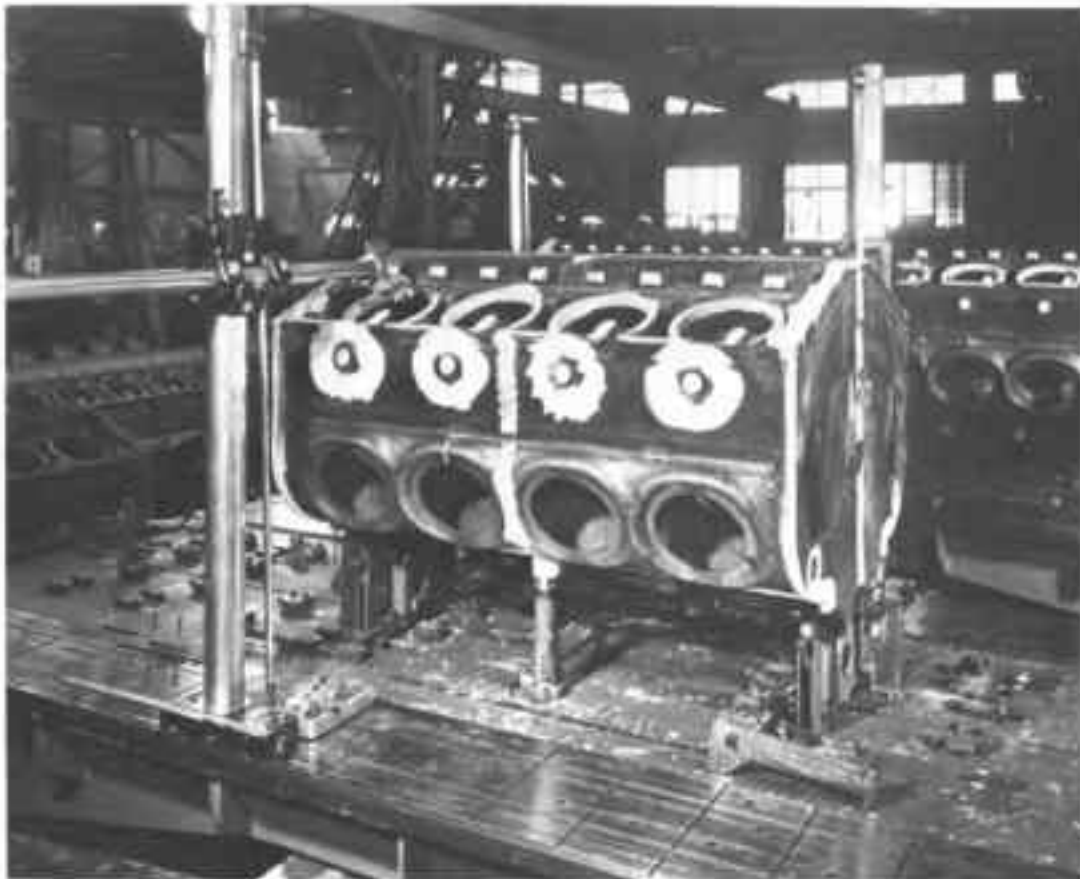


Courtesy of the Bendix Automation & Measurement Div.

Fig. 3-19. Laying out a large casting with a layout machine.

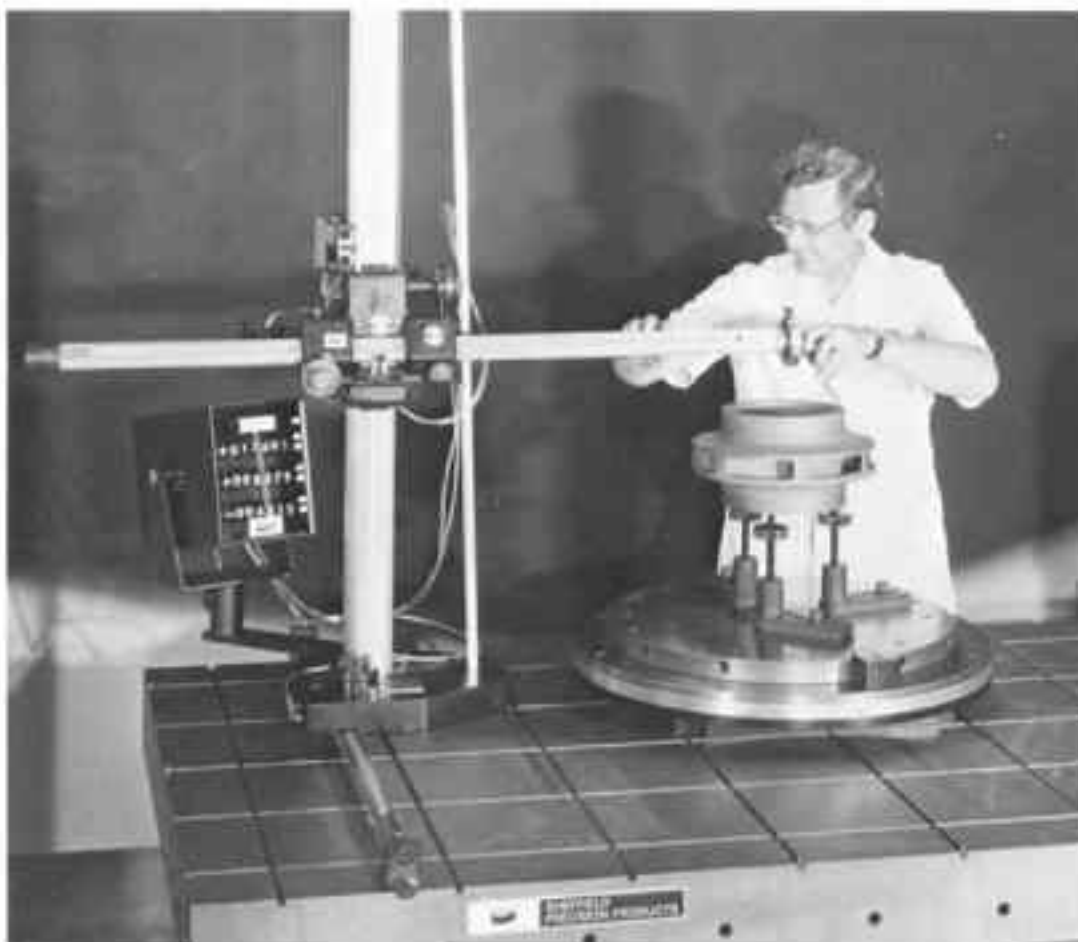
or when a setup line must be continued on these surfaces. The layout machine is used to align the workpiece in making the second setting by using the scribe on the arm of the layout machine to check the alignment of the setup lines previously scribed on the casting.

Production machine tools are usually equipped with work holding fixtures that locate the workpiece so that the cutting tool will machine the surfaces of the part in their exact location without requiring a layout to be made. However, on more complicated and expensive castings, or when costly machining operations are to be performed on the castings, a layout is usually made prior to machining to verify that the casting will "clean up" on all of the surfaces to be machined. A typical production layout operation of this type is shown in Fig. 3-20. The cylinder block in this illustration is mounted on a special layout fixture for this layout operation, which is performed by two layout machines, one placed on each side of the casting.



Courtesy of the Bendix Automation & Measurement Div.

Fig. 3-20. Production layout of castings using two layout machines, one on each side of the casting.



Courtesy of the Bendix Automation & Measurement Div.

Fig. 3-21 Layout machine with a digital readout.

Layout machines may be equipped with a digital readout system, as shown in Fig. 3-21. The numbers on this display are clearly visible, which saves time and reduces the possibility of an error in reading the small scales on the ordinary layout machine. The digital readout can be turned on to read either in inches or in millimeters. The readings can be in increments of .001 or .005 inch, or in 0.01 or 0.005 millimeter.

Drilling Machines, Twist Drills, and Auxiliary Tooling

Drilling machines, or drill presses, as they are also called, are used to open, enlarge, and to finish holes by drilling, reaming, boring, counter-boring, countersinking, and tapping. These operations are also performed on other machines such as lathes, boring mills, milling machines, and precision jig borers. Many different types of drilling machines have been built for handling different classes of work to the best advantage. Some designs incorporate great versatility in handling a wide variety of different workpieces, while others facilitate the mass production of duplicate parts. Many modern drilling machines are numerically controlled; others are automatically cycled. The horsepower of drilling machines may range from a fractional horsepower for drilling small holes to as much as fifty horsepower for driving a heavy-duty, multiple-spindle drill head that is capable of drilling many holes simultaneously.

Upright Drilling Machine

Upright drill presses, illustrated in Figs. 4-1 and 4-2, are general purpose drilling machines; yet, they can be readily adapted to do production work. The upright drill press shown in Fig. 4-1 has a round column and circular table; however, some upright drilling machines, such as the one shown in Fig. 4-2, have a box column and a square table. The circular table can be raised and lowered. It can also be rotated about the column and about its own center. This feature provides a very convenient method of aligning workpieces that are clamped to the table, with respect to the spindle. The workpieces are clamped with bolts anchored in T slots that are machined in the table. An American Standard Morse Taper is used to hold and drive twist drills, drill chucks, drill sleeves, reamers, and other cutting tools. The spindle housing may be raised or lowered and it contains the feed mechanism. An automatic power feed mechanism is provided in order to feed the twist drill through the work. Different spindle speeds are obtained by shifting the control lever that is on the head of the machine.

Sensitive Drilling Machine

The machine in Fig. 4-2 is a sensitive upright drilling machine. This machine is used for drilling smaller holes and it is not provided with a



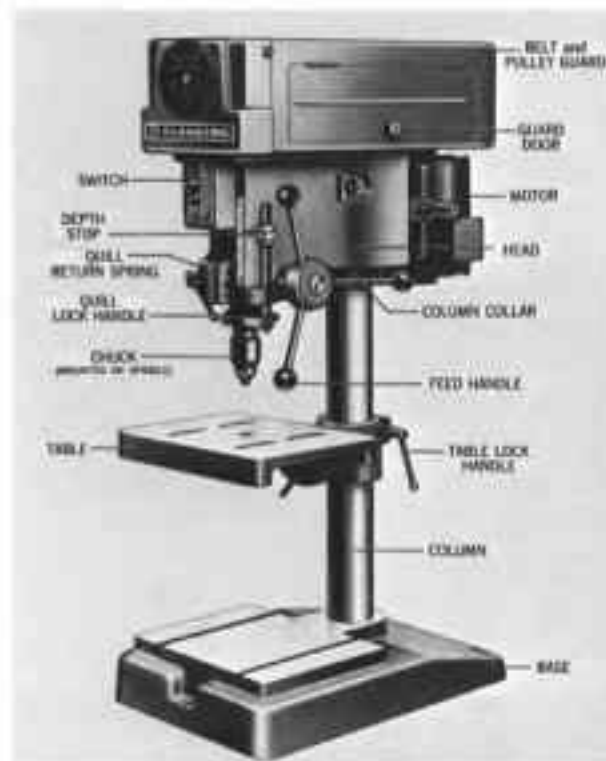
Courtesy of The Fostick Machine Tool Company

Fig. 4-1. Upright drilling machine—round column, circular-table type.



Courtesy of The Fostick Machine Tool Company

Fig. 4-2. Sensitive upright drilling machine—box column, square-table type.



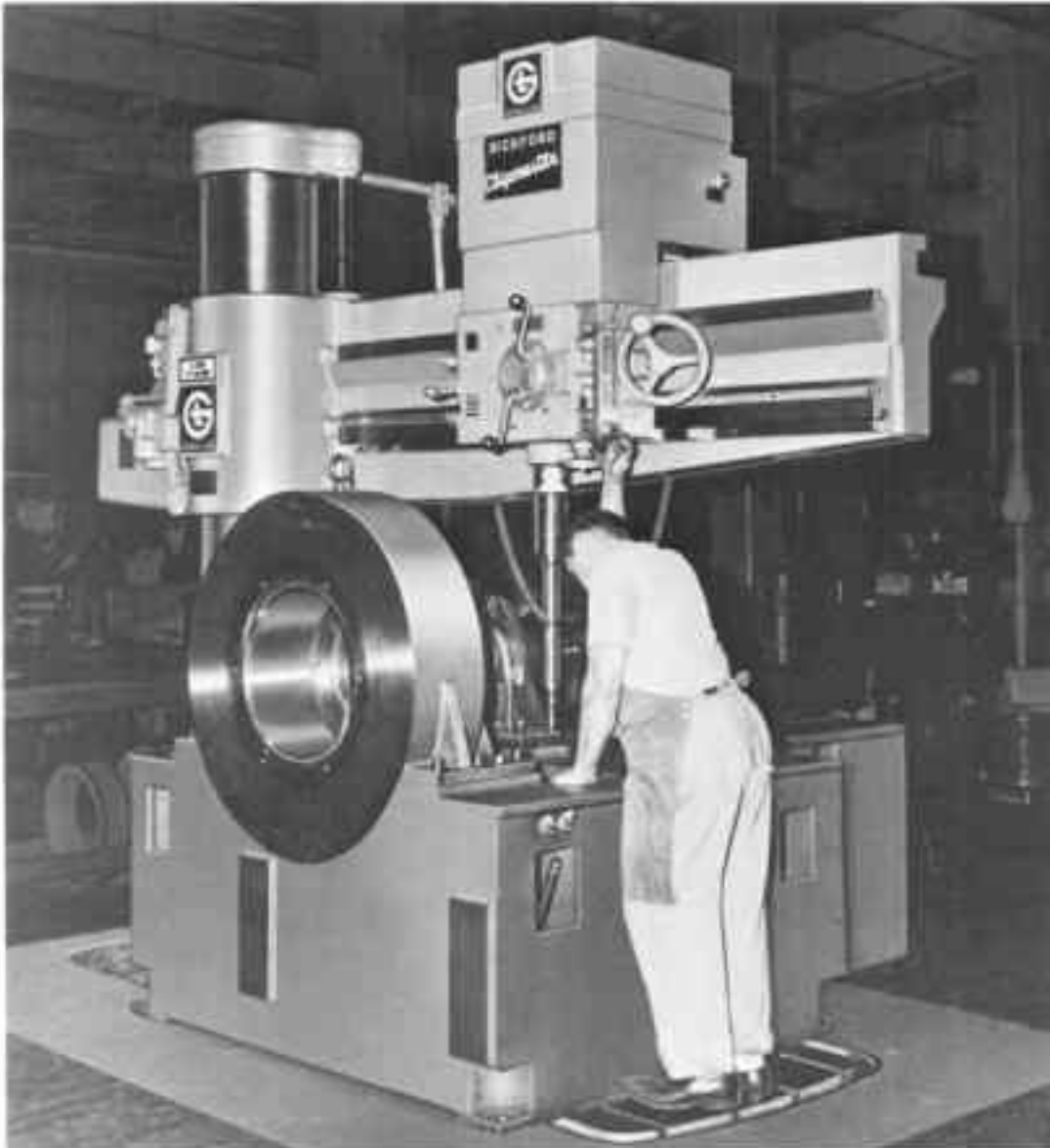
Courtesy of the Clamping Division of The Atlas Press Company

Fig. 4-3. Bench type, sensitive drill machine.

power feeding mechanism. Instead, it is designed to provide a fine sense of feel to the spindle feed lever which aids the operator in feeding the drill through the hole. It is similar in construction to an upright drill press although its features are lighter. A bench-type, sensitive drill press is shown in Fig. 4-3. Very small, sensitive drill presses capable of drilling extremely small holes are built to do instrument work.

Radial Drilling Machine

Radial drilling machines, Fig. 4-4, are very useful for drilling holes in larger workpieces. In this illustration, holes are being drilled in a large machine frame which is resting on the floor plate, or base, of the radial drill press. The arm of the radial drilling machine is supported by a large, round column. The arm can be raised and lowered, and it can be rotated around the axis of the column. The spindle head, or drilling head, is mounted on the arm. It contains the spindle, the spindle speed-change mechanism, and the automatic power feed mechanism. The spindle head can be moved lengthwise along the arm and clamped in any desired location. Thus, the spindle moves in an arc around the center of the column, and radially, in or out from the column. This design affords great flexibility and ease in positioning the drill in the exact location desired



Courtesy of the Giddings & Lewis Machine Tool Company

Fig. 4-4. Radial drilling machine drilling large machine frame which is resting on base of drill.

on the workpiece. The arm of the radial drill press provides a means of extending the spindle over a workpiece in order to drill holes in surfaces that cannot be reached by the ordinary drill press, as shown in Fig. 4-4.

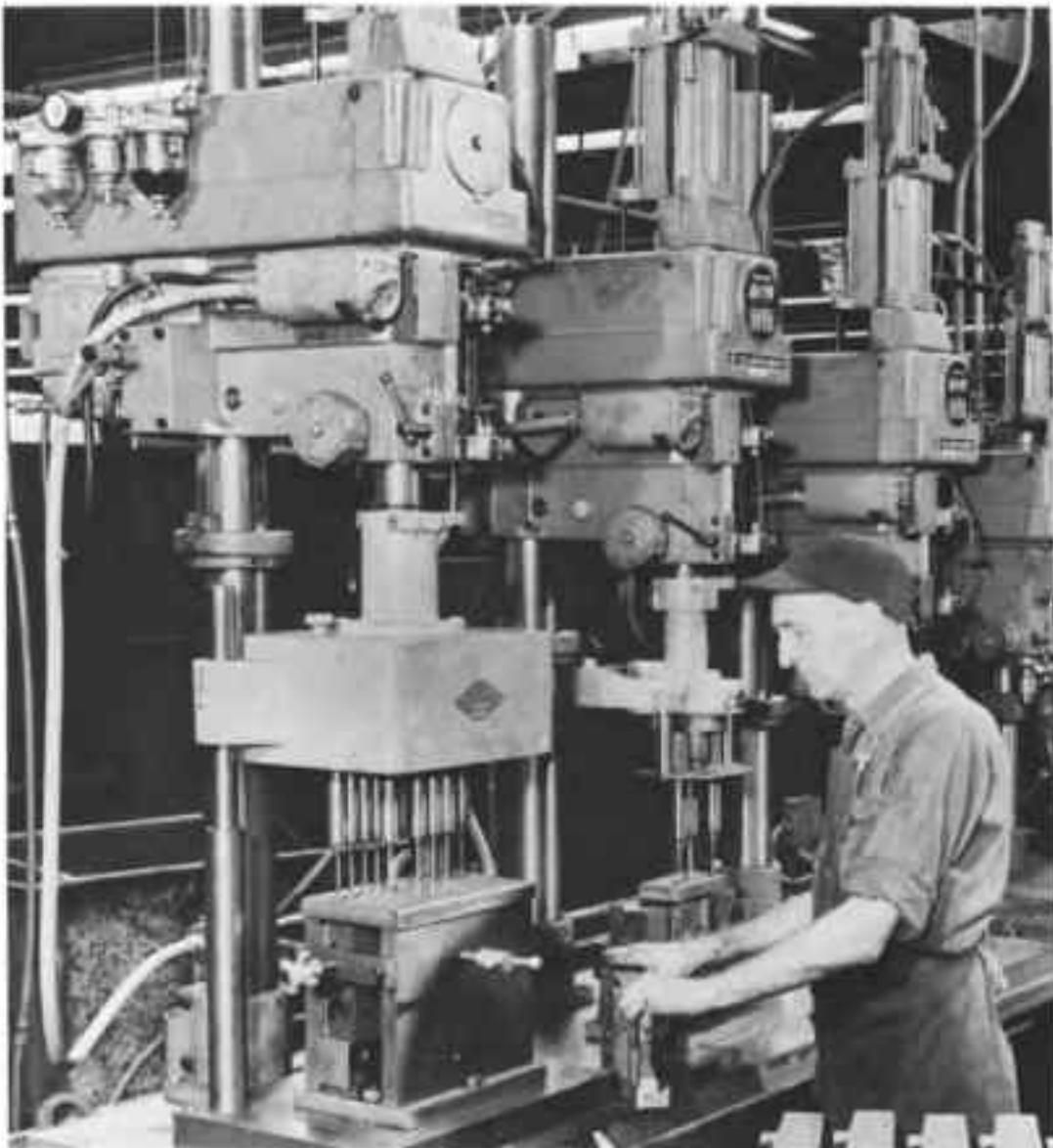
Turret Drilling Machines

Some drilling machines are equipped with a turret having six faces. Each face has a spindle which can hold a drill or other hole-producing

cutting tool. The turret is indexed manually or automatically to bring each spindle into the operating position with respect to the work. Thus, a number of operations can be performed on a hole, or a group of holes, without the need to change cutting tools as would be the case if a single-spindle drilling machine was used.

Multiple Spindle and Gang Drilling Machines

These machines are used for high production work. Multiple spindle machines have two or more spindles built on a single head. All of the drills feed into the work together, resulting in the production of two or more holes, simultaneously. As many as thirty or more spindles have been



Courtesy of the Clamping Division of The Atlas Press Company

Fig. 4-5. Gang drilling-machine equipped with air-hydraulic spindle feeds. Multiple drill-heads are mounted on two of the spindles.

built into a single head. Special multiple spindle heads have been designed which can be attached to the spindles of upright drilling machines. Multiple-spindle drilling machines are also called "way-type" drilling machines; the name being derived from "ways" on which the multiple drill heads slide. One to four ways can be built into one machine, each way containing a multiple drill head. The ways can be designed to cause the drill heads to slide at any angle from vertical to horizontal.

Gang drilling machines consist of two or more drilling machine spindles that are positioned vertically over a single table. Each spindle contains a different cutting tool and the work is moved from one spindle to the next. Gang drill presses are, in effect, upright drilling machines with two or more spindles. Figure 4-5 illustrates a gang drilling machine. Multiple spindle heads are mounted on two of the spindles and the work is held in a box jig which is fastened to the table. The work is moved from one spindle to another to complete all of the drilling operations.

Twist Drills

The basic metal-cutting tool for opening holes in metal is the twist drill. Its apparent simplicity and the ease with which it can be used tend to obscure the fact that it is a complicated cutting tool with a very complex cutting action. The twist drill must withstand high torque and thrust forces, as well as the temperatures resulting from the formation of the chip. The various elements of a twist drill are defined below and in Fig. 4-6:

Axis—The imaginary straight line forming the longitudinal center of the drill.

Body—That portion of the drill extending from the shank, or neck, to the outer corners of the cutting edge.

Flutes—The helical grooves cut, or formed, in the body of the drill to provide cutting lips, to permit removal of chips, and to allow cutting fluid to reach the cutting lips.

Land—The peripheral portion of the body between adjacent flutes.

Body Diameter Clearance—That portion of the land that has been cut away so that it will not rub against the walls of the hole.

Margin—The cylindrical portion of the land which is not cut away to provide clearance.

Web—The central portion of the body that joins the lands. The extreme ends of the web form the chisel edge. The thickness of the web is not uniform, but increases from the point toward the shank.

Point—The cutting end of the drill, made up of the ends of the lands and the web. In form it resembles a cone, but departs from a true cone to furnish relief behind the cutting lips in order that they can penetrate into the metal and form a chip.

Point Angle—The angle included between the cutting lips, projected upon a plane parallel to the drill axis, and parallel also to the two cutting lips.

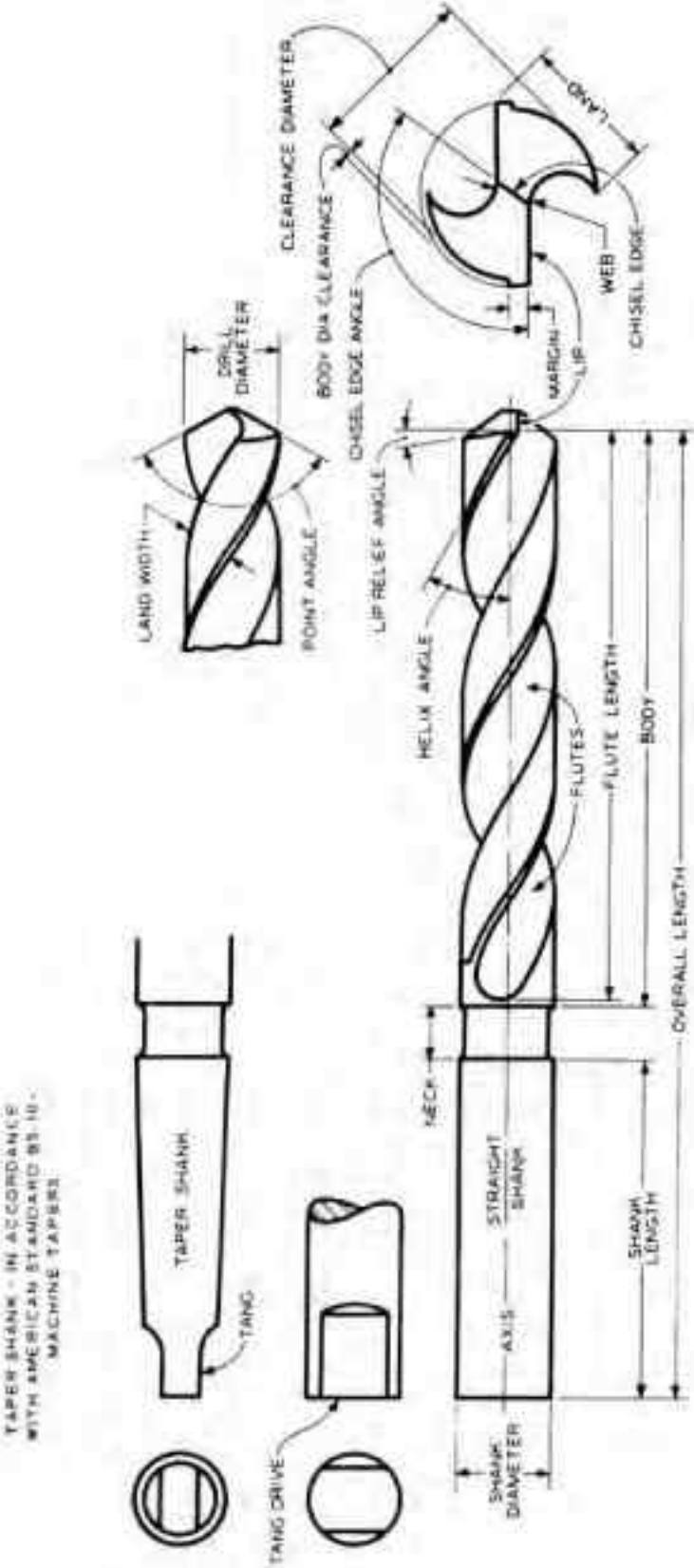


Fig. 4-6. Nomenclature of a twist drill.

Lips—The two cutting edges extending from the chisel edge to the periphery.

Relief—The result of the removal of tool material behind or adjacent to the cutting lip and leading edge of the land, to provide clearance; prevent heel drag; and to allow the cutting lips to penetrate into the work and form the chip.

Lip Relief Angle—The axial relief angle at the outer corner of the lip. It is measured by projection into a plane, tangent to the periphery, at the outer corner of the lip.

Chisel Edge—The edge at the end of the web that connects the cutting lips.

Chisel Edge Angle—The angle included between the chisel edge and the cutting lip, as viewed from the end of the drill.

Helix Angle—The angle made by the leading edge of the land with a plane containing the axis of the drill. This angle is also the rake angle of the drill.

Shank—That part of the drill by which it is held and driven.

Straight Shank—A shank having the form of a cylinder. Usually provided on drills that are to be held in a chuck.

Taper Shank—A shank having an American Standard (Morse) Self-Holding Taper.

Tang—The flattened end of a taper shank, intended to fit into a driving slot in a socket or a machine spindle.

Tang Drive—Two opposite, parallel driving flats on the extreme end of a straight shank.

Twist Drill Sizes

Twist drill sizes are given in fractions of an inch, number sizes, and letter sizes; metric drills are specified in terms of a millimeter. The sizes are specified by the American National Standards Institute (ANSI) and are shown in Table A-2 in the Appendix.

Holes produced by twist drills are usually somewhat oversize, as shown in Table 4-1. The accuracy of the hole is dependent on the following factors: 1. the actual size of the drill diameter; 2. the accuracy of the drill point; 3. the accuracy of the drill chuck and drill sleeve; 4. the accuracy and rigidity of the drilling machine spindle; 5. the rigidity of the drilling machine; and, 6. the rigidity of the workpiece and the setup. Sometimes slightly rounding the corners of the lips of the drill at the margins with a hone will improve the accuracy of the drilled hole. Since the tolerance of most drilled holes is usually quite liberal and the holes produced by twist drills are usually somewhat oversize, it is possible to drill a rather wide range of metric size holes with inch size twist drills. Table A-2 in the Appendix provides a guide for this purpose.

Table 4-1. Oversize Amount Normally Cut by a Drill
Under Normal Shop Conditions, in Inches

Drill Dia, Inch	Amount Oversize, Inch		
	Average Max	Mean	Average Min
$\frac{1}{16}$	.002	.0015	.001
$\frac{3}{16}$	.0045	.003	.001
$\frac{1}{4}$	.0065	.004	.0025
$\frac{1}{2}$	.008	.005	.003
$\frac{3}{4}$	.008	.005	.003
1	.009	.007	.004

Courtesy of the Metal Cutting Tool Institute

Twist Drill Geometry

The flutes of most twist drills have a standard helix angle and these drills can be used to drill almost all materials. For general purpose work, twist drills having a non-standard helix angle are not required; however, in cases where certain materials must frequently be drilled, non-standard helix angle drills may have an advantage. High helix angle twist drills are made and they are recommended for drilling low tensile strength materials such as aluminum, magnesium, die casting alloys, and some plastics. There are also low helix angle twist drills which are used to drill brass, soft bronze, and sheet metal. These materials are also drilled using straight fluted drills, having a zero degree helix angle, because they will not tend to pull or run ahead of the feed, and when breaking through the hole they will have less tendency to grab. Twist drills having a low helix angle combined with a thicker web are used when drilling some very hard and tough materials.

After a twist drill has been made, its geometry is fixed except at the point, where it may be ground to meet different requirements. The point geometry consists of the point angle, lip relief angle, chisel edge angle, the lip lengths, and the chisel edge length. Suggested lip relief angles are given in Table 4-2. The lip relief angle is dependent on the size of the drill and on the type of material being drilled. Lip relief angles are also specified in Fig. 4-7 for drill sizes that are approximately $\frac{1}{2}$ in. (13.0 mm) and larger. Since the size of the drill is important, the values in Table 4-2 should supersede those in Fig. 4-7 for sizes other than $\frac{1}{2}$ in. The importance of the chisel edge angle should not be overlooked. This angle gives an indication of the amount of relief behind the chisel edge and behind the portion of the lips near the chisel edge. The recommended values of the chisel edge angle in Fig. 4-7 will provide adequate relief in

Table 4-2. Suggested Lip Relief Angles at the Periphery

Drill Sizes	Drill Diameter Range		Suggested Lip Relief Angle at Periphery		
	inch	mm	For General Purpose Drills	Hard and Tough Materials	Soft and Free Machining Materials
No. 80 to No. 61	.0135 to .0390	0.35 to 1.00	24°	20°	26°
No. 60 to No. 41	.0400 to .0960	1.05 to 2.45	21°	18°	24°
No. 40 to No. 31	.0980 to .1200	2.50 to 3.05	18°	16°	22°
No. 30 to 1/4"	.1285 to .2500	3.25 to 6.35	16°	14°	20°
F to 11/32"	.2570 to .3433	6.55 to 8.75	14°	12°	18°
S to 1/2"	.3480 to .5000	8.85 to 12.70	12°	10°	16°
33/64" to 3/4"	.5156 to .7500	13.10 to 19.05	10°	8°	14°
45/64" and larger	.7656-	19.45-	8°	7°	12°

this region of the drill point. Failure to provide adequate relief will result in an increase in the thrust force required to feed the drill and cause the drill to wear more rapidly. Along with the other components of the drill point geometry, the size of the chisel edge angle should be watched when grinding a drill point.

Drill points recommended for drilling a variety of materials are shown in Fig. 4-7. All of these drill points must have two essential requirements, shown at A: 1. the lengths of the two lips "L" must be equal; and, 2. the angles "A" must be equal. Failure to meet these requirements will cause the lips to do an unequal amount of work, resulting in an excessively oversize hole, more rapid tool wear, and place a greater load on the drill.

There is a standard drill point geometry (see Fig. 4-7B) that will successfully drill most materials, unless it is exceptionally difficult to drill or when an advantage is to be obtained in production drilling. For most general purpose work, this drill point is satisfactory and can be recommended. When drilling tool steels and heat treated steels having a higher hardness, the point angle should be increased as shown in Fig. 4-7C. The point angle should be increased even more and the lip relief angle reduced when drilling tough materials that tend to harden when they are being worked, such as manganese steel. The drill point for such materials is shown at D. Sometimes a split point drill grind (views I and J) is even more effective for these materials, but this point is more difficult to grind. When grinding brass, soft bronze, and sheet metal there is a tendency for the drill to pull and to grab when breaking through the hole. This tendency is reduced by grinding a flat on the face of each lip, as shown in view E. Both flats must be made equal in size. The flats are small plane surfaces that reduce the effective rake angle on the cutting edges to zero degree. For drilling softer materials the point angle is decreased, which tends to reduce the wear resulting from the build-up of

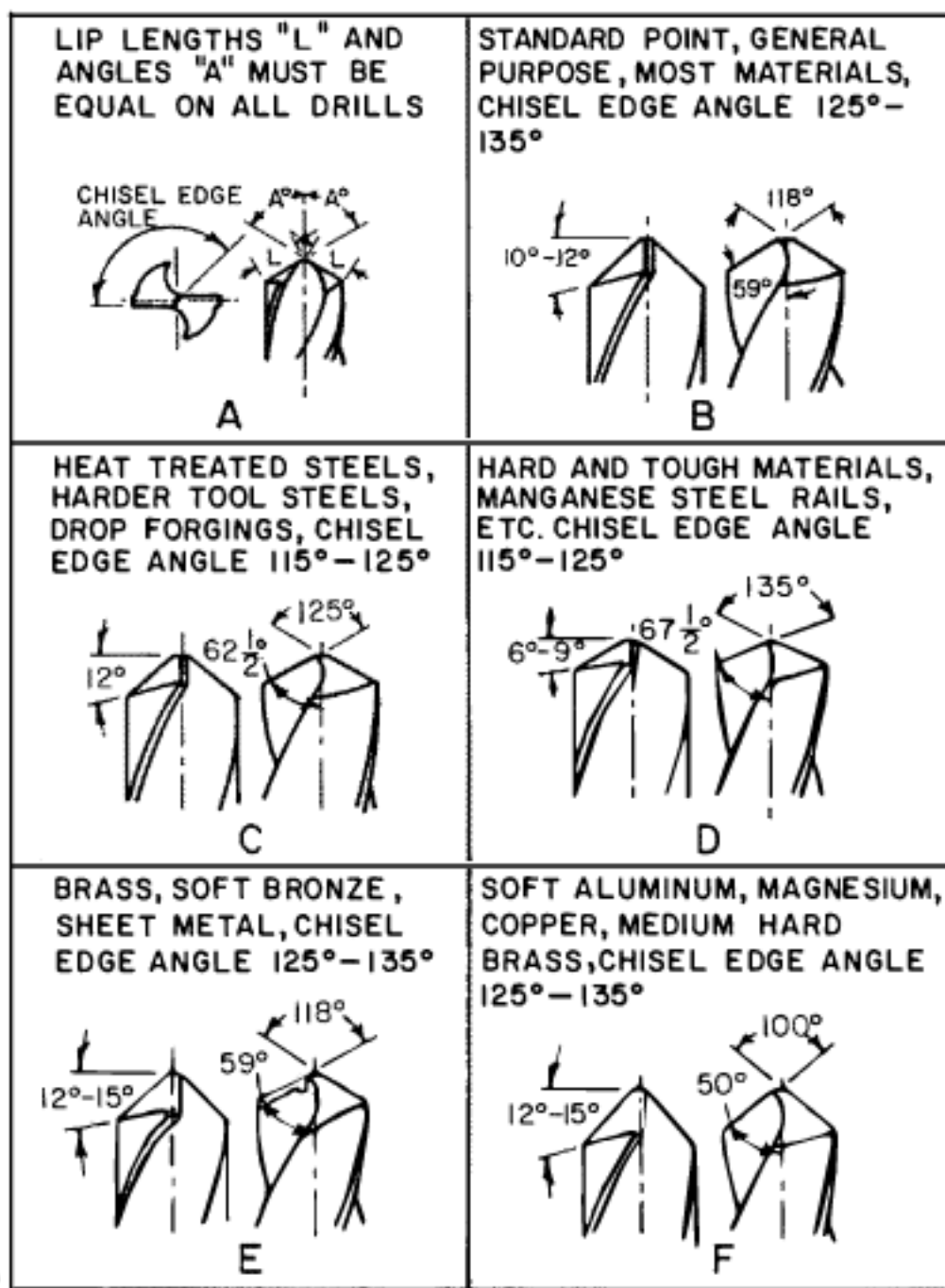


Fig. 4-7A. Suggested drill point geometry for drilling various materials.

heat in the corners of the point formed by the lips and the margins. As shown in views F, G, and H, a slight increase in the lip relief angle is also recommended for these materials. Surprisingly, it has been found that even soft and medium hard cast iron can be drilled more effectively with a more pointed drill point shape (View G).

The split point, views I and J, is a most effective drill point geometry

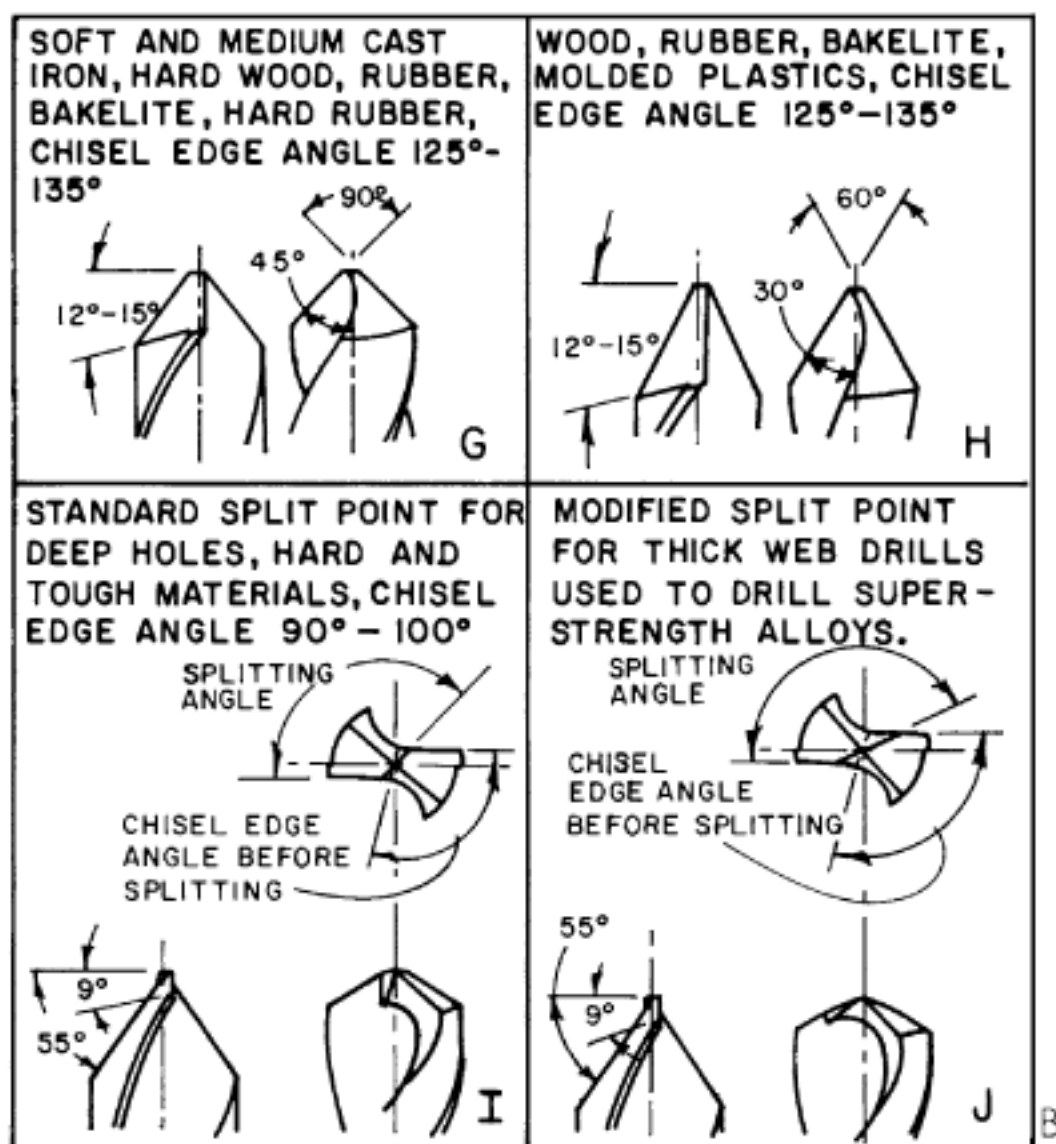


Fig. 4-7B. Suggested drill point geometry for drilling various materials.

for drilling deep holes, extremely hard and tough materials, and materials that are highly subject to work hardening. A photograph of this point is shown in Fig. 4-8. This drill point should be ground on a machine, such as a cutter and tool grinder or a point thinning machine, because it is somewhat difficult to grind. The chisel edge is replaced by two small secondary cutting edges. The secondary cutting edges reduce the extrusion of the metal (see Fig. 4-13) in this region to a minimum, providing instead a better cutting action and a better escape route for the chips produced in this region. This reduces the thrust force required to feed the drill by as much as one-half, thereby reducing the load on the drill and on the machine. The less severe cutting action of the split point

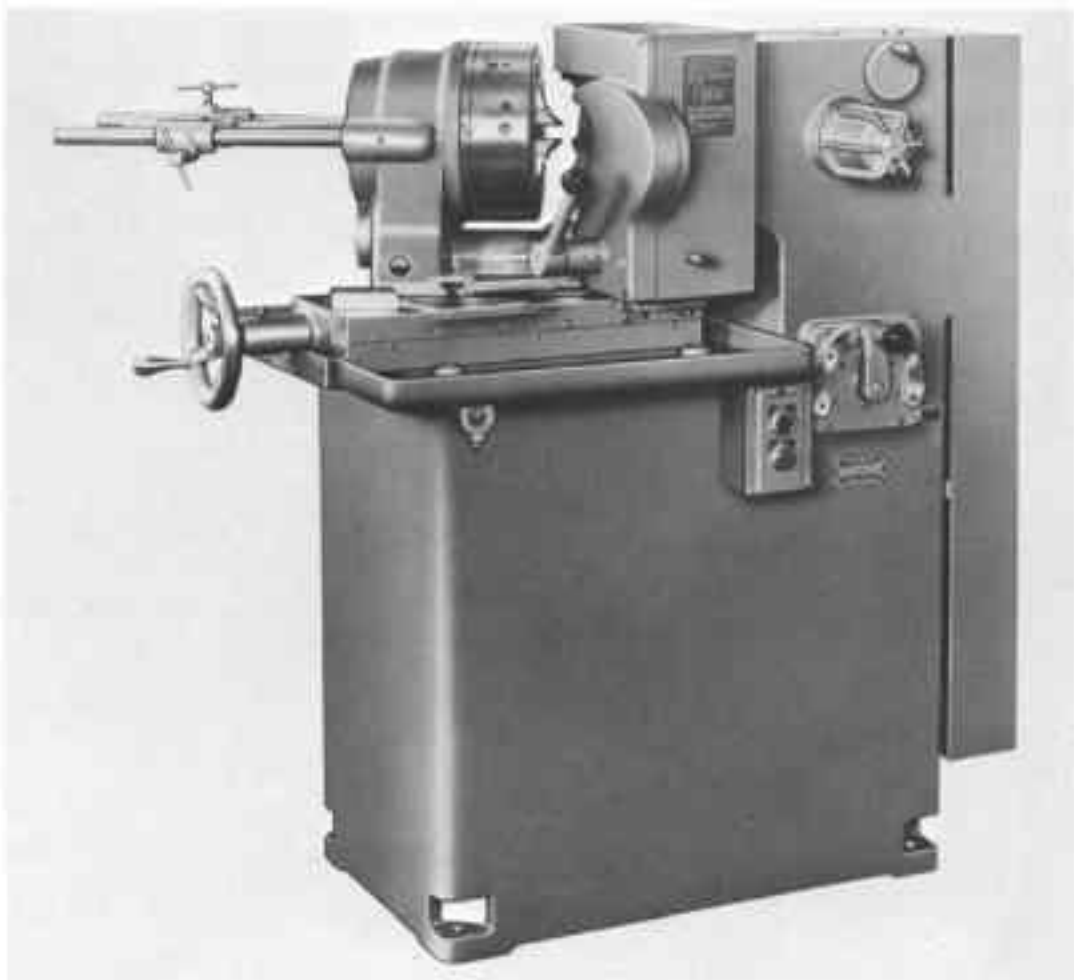


Courtesy of the National Twist Drill & Tool Div., Lear-Siegler Inc.

Fig. 4-8. Standard split-point drill grind. Web replaced by two secondary cutting edges.

will also reduce the degree of work hardening of the work material. A standard split point is shown in Fig. 4-8 and in view I, Fig. 4-7. This split point geometry is recommended for drilling the more conventional steels and for deep hole drilling. The secondary cutting edges at the drill point are formed by grinding a 55 degree bevel along the back edge of the land as shown. The bevel must also be oriented to form the correct splitting angle, which for the conventional split point drill should be approximately equal to the chisel edge angle, or approximately 130 to 135 degrees. At the center, the two cutting edges formed by the split should be separated by a small "chisel edge" that is about .005 to .015 inch (0.13 to 0.38 mm) wide, depending on the size of the drill. By increasing the splitting angle 5 to 15 degrees (150 degrees maximum) on standard twist drills that are ground to have a split point, the centering action, when starting to drill the hole, will be improved. This type of drill point may be used to drill ordinary materials on numerically controlled machines. Precipitation or age hardening stainless steels, high temperature alloys, and super-high-strength steels should be drilled with a thick web drill when possible, where the web is 50 to 200 percent thicker than the conventional web. A modified split point, shown in view J, Fig. 4-7, is recommended for drilling these materials with a heavy web drill. The splitting angle is increased to approximately 170 degrees to lengthen the secondary cutting edge, which should extend to approximately one-half of the length of the lip, or primary cutting edge. The modified split point should not be used on drills having a conventional web thickness.

Drill points are preferably ground on a machine designed for this purpose. The drill point grinding machine in Fig. 4-9 will grind a theoretically correct drill point. It can also grind a point having a small nib at the tip that will make the drill self-centering when starting. In many shops a drill point grinding machine is not available and drill points are ground by hand. A skilled machinist or toolmaker can grind a drill point by off-hand grinding. This skill is acquired by practice and is difficult to describe. Briefly, the drill must be positioned against the face of the grinding wheel at the correct angle and with one lip held horizontally. It is held and pivoted in the left hand near the point while it is rocked up and down by the right hand, which is holding the shank of the drill. At the same time it is rotated clockwise a small amount and moved slightly inward toward the grinding wheel. It is returned by making these moves in the opposite directions. Each lip is ground in this manner until it is sharp and the point has the correct geometry.



Courtesy of the Oliver Instrument Co., Inc.

Fig. 4-9. Oliver drill point grinding machine. This grinder will grind the lip relief angle increasing as the center of the drill is approached.

On most twist drills the web increases in thickness toward the shank end. When a drill is shortened by repeated grinding, the increase in the web thickness results in a lengthening of the chisel edge. This, in turn, causes the thrust force required to feed the drill to increase. For this reason, whenever the length of the chisel edge is increased much beyond its original length, the web should be thinned at the point of the drill. Even on new drills it is sometimes advisable to thin the web.

As shown in Fig. 4-10, in order to thin the web at the drill point, a small amount of metal is ground off from both flutes near the chisel edge. The same amount should be ground off from each flute. For best results, the drill should be clamped in position and ground on a cutter and tool grinding machine or on a special point thinning machine, using a thin grinding wheel that has a rounded face. The position of the twist drill relative to the grinding wheel is shown at A, Fig. 4-10. By exercising great care, web thinning, or point thinning, as it is also called, can be done with the drill held by hand. When necessary, it is better to thin the web at the point by the off-hand method than not at all.

There are two methods of thinning the web. The conventional method used on both large and small drills is to grind the lips and the side opposite the lips as shown at B, Fig. 4-10. The grind should not extend more than one-half the length of the lips, or less. On larger drills it is possible to grind only on the surface of the flutes opposite to the lips, as is shown at C, Fig. 4-10. This is the preferred method on larger drills.

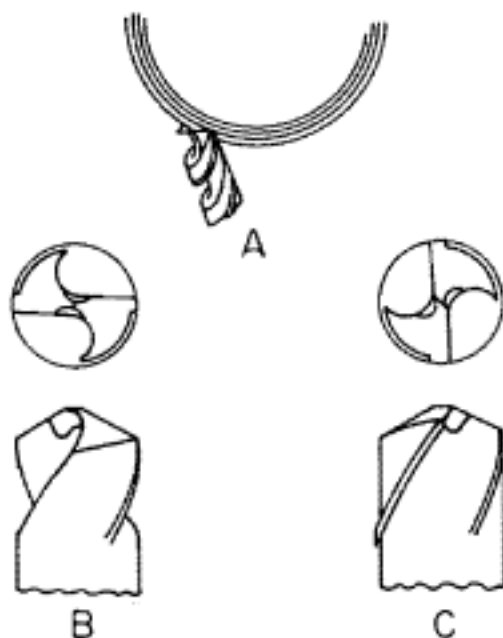


Fig. 4-10. A. Position of twist drill relative to narrow, round face grinding wheel for web thinning. B. Conventional grind for web thinning. C. Preferred web-thinning grind for larger drills.

Chip Formation in Drilling

When metals are subjected to the action of a cutting edge that is harder and stronger than the metal that is being cut, a chip is formed. As the cutting edge engages the workpiece, the material ahead of the cutting edge is compressed and escapes by flowing in a direction parallel to the face of the tool. As the cutting edge continues its penetration, the force on the tool and on the work increases until it is great enough to cause either rupture or shear within the grains of the metal, on a plane that is roughly perpendicular to the face of the tool. This plane is called the shear plane and is shown in Fig. 4-11. The shear plane is usually a somewhat curved surface rather than a perfectly flat surface. The chips may be discontinuous as at A in Fig. 4-11, or they may be continuous as shown at B. Brittle materials, such as cast iron, tend to form discontinuous chips while ductile materials, such as soft steel, tend to form continuous chips. Figure 4-12 is a photomicrograph of a ductile steel, showing the distortion of the grains along the shear plane in the forming of a continuous chip. The lips of the twist drill form a chip in exactly this manner.

The total cutting action of a twist drill, however, is more complex due to the action of the chisel edge, shown in Fig. 4-13. At the exact center of the chisel edge the only motion of the drill is such that the action is similar to a center punch that penetrates the metal by extruding it sideways. As the distance from the center increases, the rotation of the drill becomes effective and the chisel-edge wedge appears both to cut and extrude the metal. The very complex metal deformation can be seen by the deformation of the grains under the chisel edge in Fig. 4-13. This severe deformation causes the high thrust force required by twist drills. The severity of this deformation, and thereby the thrust force, can be reduced by thinning the web at the point and even more by the split point grind.

There are then two distinct chip formation actions in drilling. This

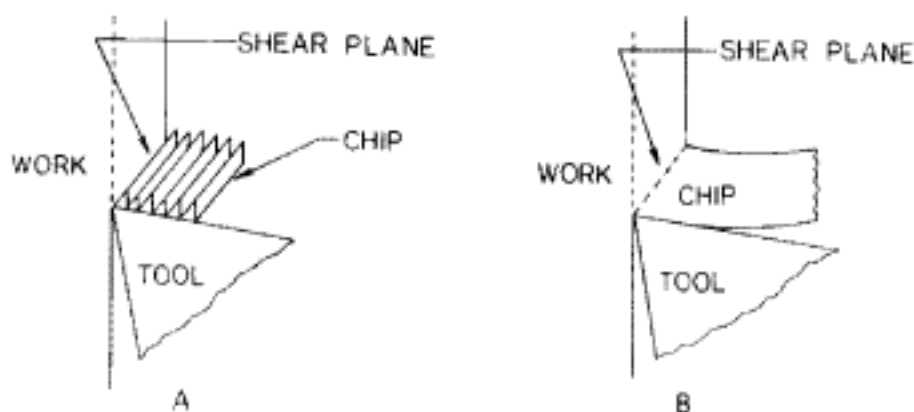
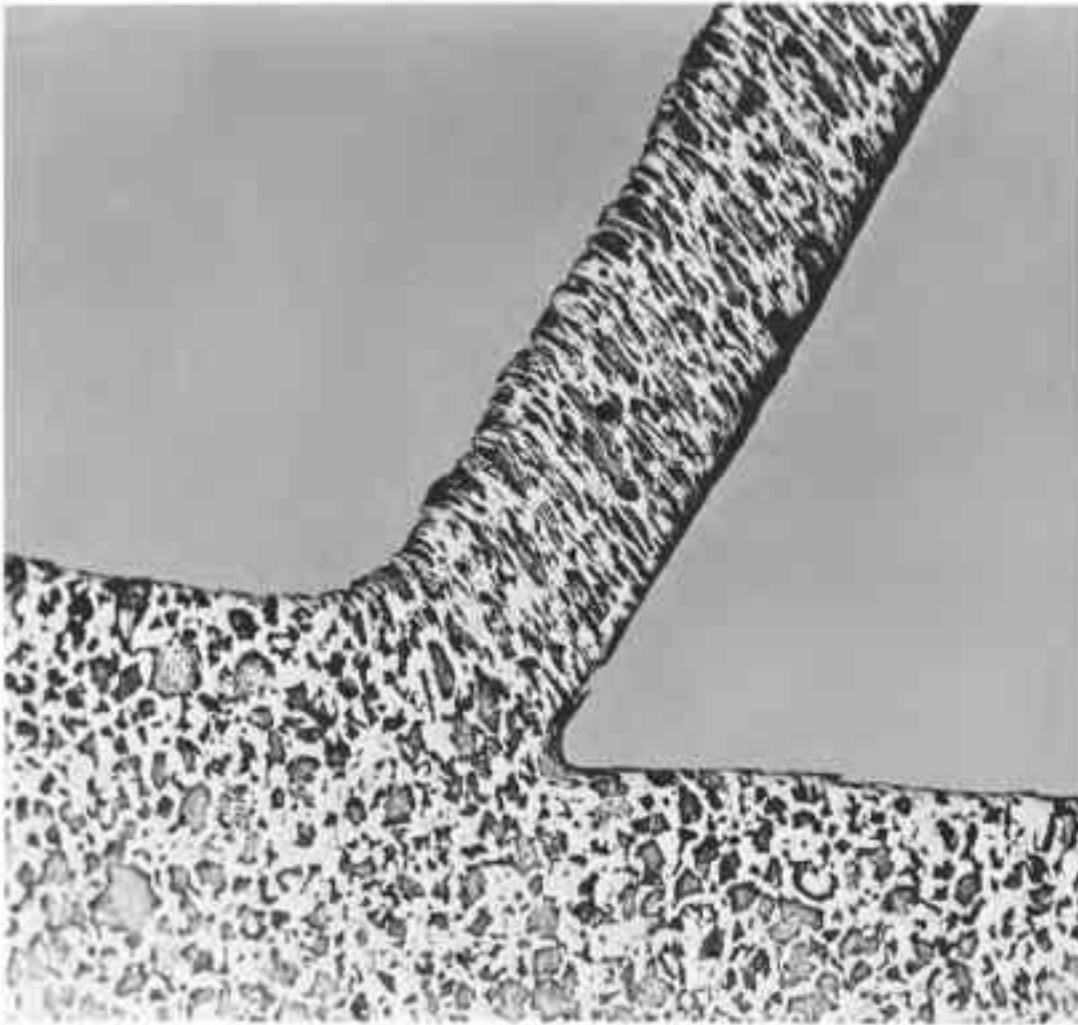


Fig. 4-11. Schematic drawing of formation of chip when cutting metal. A. Discontinuous chip. B. Continuous chip.



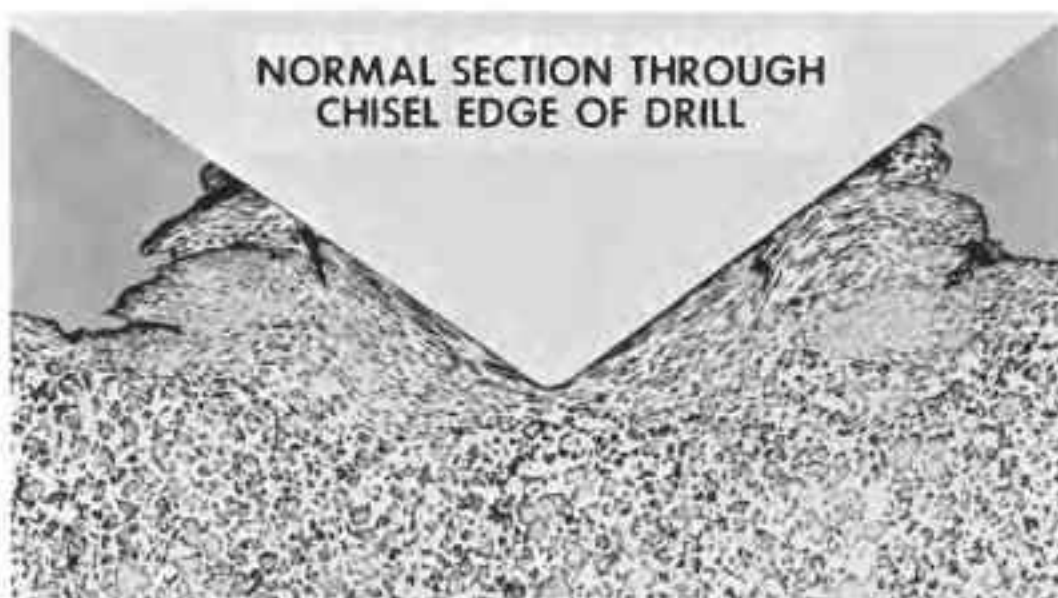
Courtesy of the National Twist Drill Company

Fig. 4-12. Photomicrograph of a metal chip formed by a cutting edge such as the lips of a twist drill.

can be readily seen in Fig. 4-14, which clearly shows the chip formed by the lip, as distinct from the chip formed by the chisel edge. In this respect, twist drills are unique among the metal-cutting tools.

Reamers

Figure 4-15 illustrates a chucking reamer and a hand reamer, and it defines their respective individual elements. Reamers are used to finish cut holes to a precise dimension and to provide a smooth surface finish on the walls of the hole. They are precision cutting tools that must always be handled with care to prevent chipping and other damage to the reamer teeth. *One cardinal principle in using a reamer is never to rotate it in the reverse direction.* Reamers should always be rotated in the forward, or cutting direction, at all times, regardless of whether they are entering or

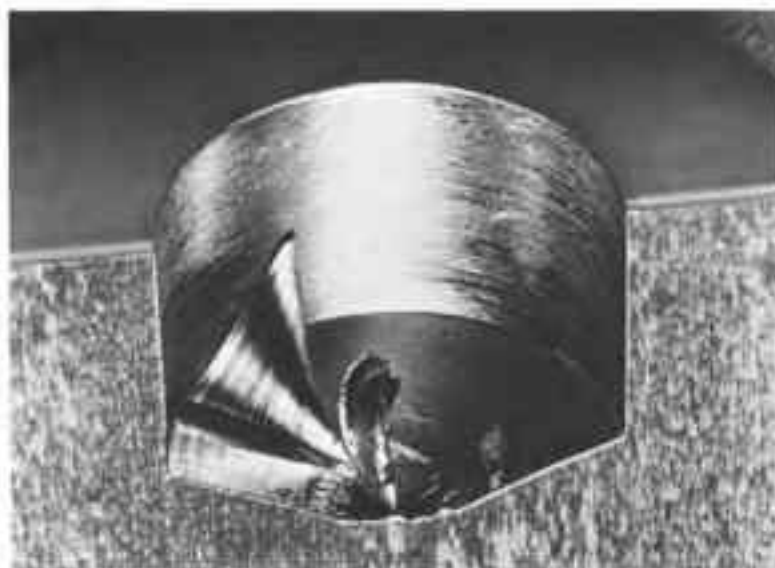


Courtesy of the National Twist Drill Company

Fig. 4-13. Photomicrograph through the axis of a drilled hole normal to the drill chisel-edge showing the complex cutting and extruding action by which the metal in this region is removed.

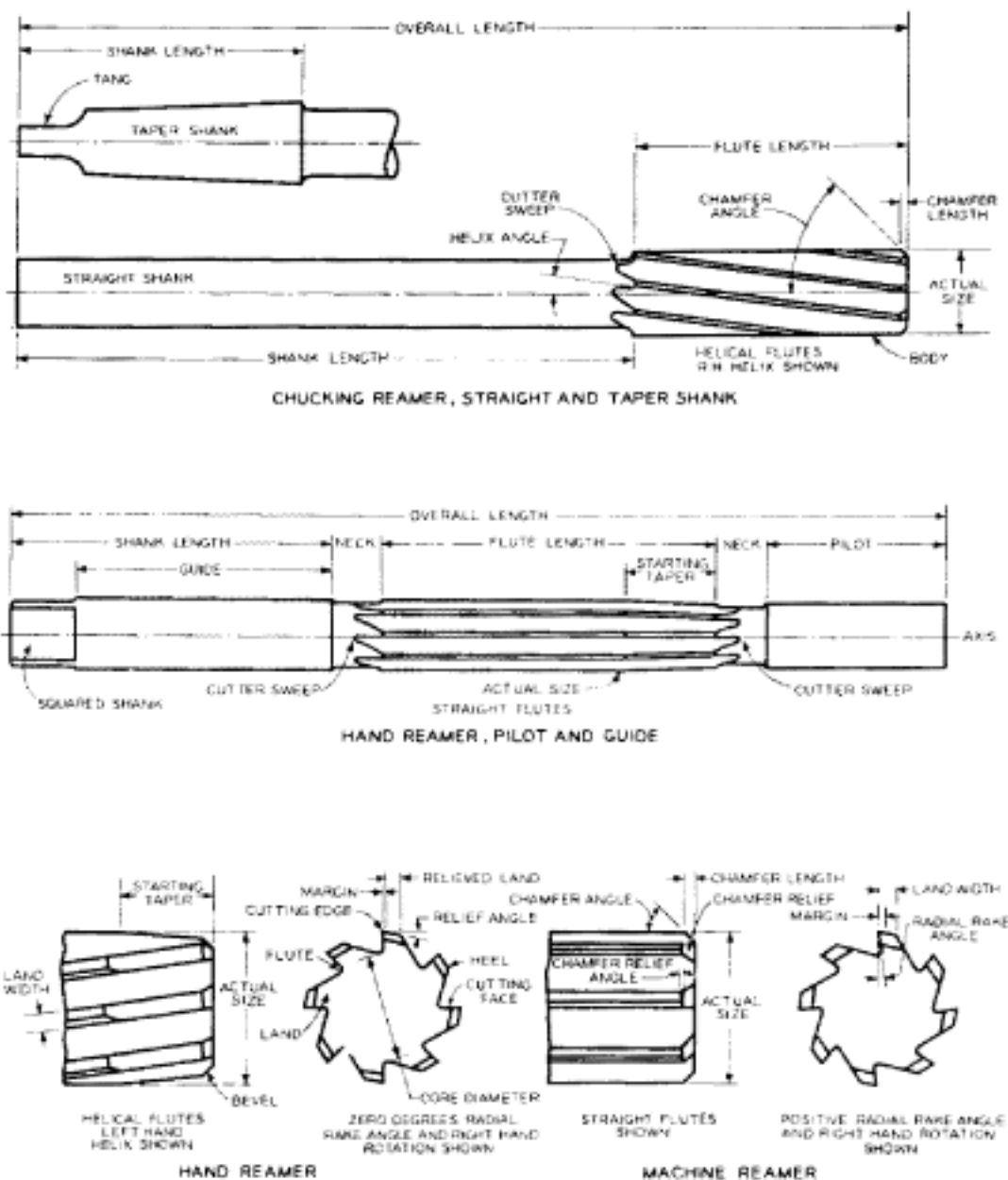
leaving the hole. Failure to do this will rapidly damage the margin and destroy the accuracy of the reamer. Reaming is a fast, efficient, and easily applied method of finishing holes.

There are two basic types of reamers: machine reamers and hand



Courtesy of the National Twist Drill Company

Fig. 4-14. Section of hole with drill removed to show separate chip formed by the lip and the chisel edge.



Courtesy of the Metal Cutting Tool Institute

Fig. 4-15. Reamer nomenclature.

reamers. Machine reamers may have a straight, or a taper, shank. They are used primarily in power-driven machine tools. Machine reamers have a chamfer ground on the ends of the lands. The chamfer angle is usually 45° , although other chamfer angles are sometimes used. The lands have a margin adjacent to the cutting edge, which is unrelieved. Relief is ground on the back of the chamfer, thereby making it into a sharp cutting edge. The chamfer edges do the cutting and remove the excess metal from the hole. The forces on the workpiece resulting from the cutting action of the reamer will cause the work to spring slightly away from the reamer, in the region of the chamfer edge. After the chamfer edge has passed a given portion of the hole, the work material will spring back and bear against

the margin of the lands of the reamer. The margin, being unrelieved, is a poor cutting edge. Primarily, it burnishes the sides of the hole to produce a good surface finish. It may also scrape a very small amount of metal out of the hole, thereby removing small irregularities. The margin also performs another important function in supporting and steadying the reamer as it passes through the hole. Sometimes, the margin is ground with a slight taper toward the chamfer edge. The length of this taper is from $\frac{1}{8}$ to $\frac{3}{16}$ inch and the angle, with respect to the axis of the reamer, is from 1° to 5° . This is sometimes called a lead. Actually, its function is to increase the burnishing action of the lands, thereby improving the surface finish of the hole.

Hand reamers are designed to be turned manually in reaming a hole. They are ground with a starting taper on the lands which is relieved to form a sharp cutting edge. (See Fig. 4-15.) A bevel edge is also ground on the end, however most of the cutting is done by the starting taper. Hand reamers are designed to remove less metal from a hole than machine reamers will. The margin of the hand reamer also burnishes the hole in the same manner as does the margin on the machine reamer.

The speed and feed through the hole of a hand reamer are controlled entirely by the workman. On machine reamers, the speed and feed are controlled by the machine setting. As a general rule, the cutting speed for reaming should be about one-half to two-thirds of the speed used for drilling the same material. Excessive cutting speeds will cause the reamer to chatter; consequently, the cutting speed must be kept low enough to prevent the occurrence of chatter. The feed for machine reaming is usually .0015 to .0040 inch per flute per revolution. Thus, the feed rate of a six-fluted reamer that is cutting at 100 rpm, using .004 inch per tooth feed, would be $6 \times .004$, or .024 inch per revolution. Expressed in different terms, the feed rate would be $100 \times .024$, or 2.4 inches per minute. When the feed used is too low the work may be glazed, the cutting edge will wear excessively, and, occasionally, chatter will occur. An excessively fast feed rate will reduce the accuracy of the hole and lower the quality of the surface finish.

It is very difficult to generalize on the amount of stock that should be removed by reaming as this is dependent upon the nature of the work material, the design of the reamer, the feed used, the finish required, and the depth of the hole. As a starting point for machine reaming, allow .008 to .012 inch on a $\frac{1}{4}$ -inch hole, .012 to .016 inch on a $\frac{1}{2}$ -inch hole, and .025 to .035 inch on holes up to $1\frac{1}{2}$ -inches in diameter. For hand reaming, the stock allowance is much smaller. The allowance ranges from .001 to .010 inch.

Two other types of reamers that are sometimes used are rose reamers and taper reamers. Rose reamers are used in a machine. They are similar to machine reamers in construction; however, they are designed to remove much more metal from a hole. They do not ream a hole as accurately or with as good a surface finish as do the standard machine reamers. Taper reamers are used to produce taper holes. Larger size reamers are often made as shell reamers. The shank, or arbor, of the

shell reamer is separate from the body of the reamer. The advantage of this design is that reamers of different size can be held on the same arbor and when a body is worn out the arbor does not require replacement.

Counterbores, Countersinks, and Spotfacers

Counterbores, countersinks, and spotfacers are frequently used in drill-press work in order to modify an existing hole. The counterbore, A, in Fig. 4-16, is used to enlarge the end of a hole by cutting a cylindrical surface that is concentric with the original hole. The concentricity is obtained by the pilot located on the end of the counterbore and which acts to guide the tool as it penetrates into the work. A countersink, B, in Fig. 4-16, is used to make a cone-shaped enlargement in the end of the hole. A spotfacing tool is shown at C. The purpose of the spotfacing operation

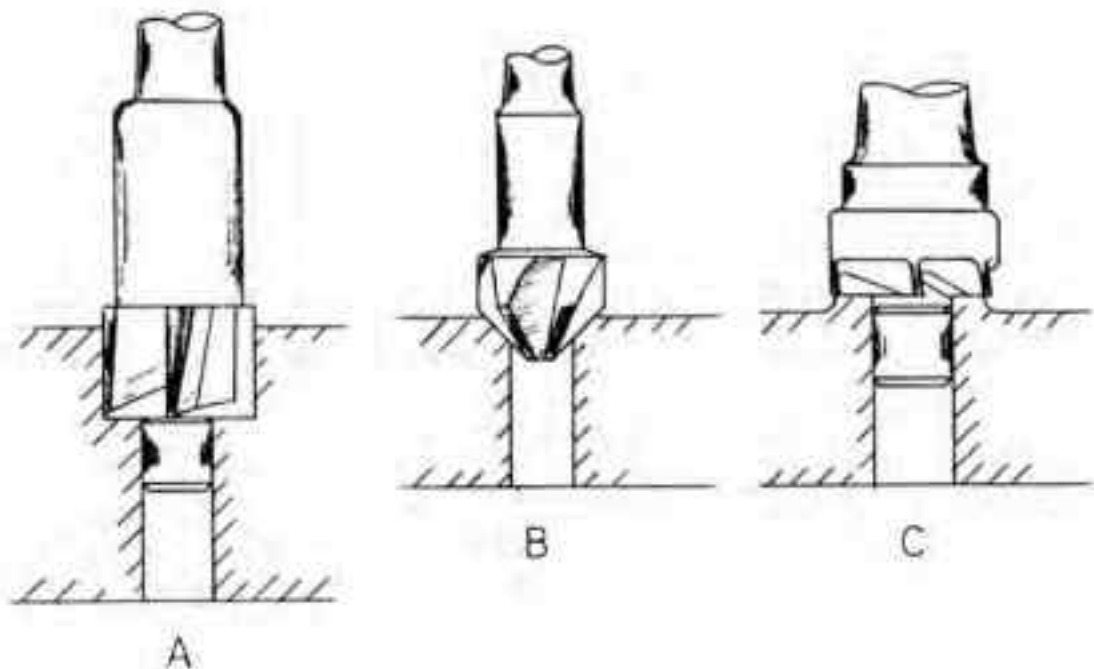
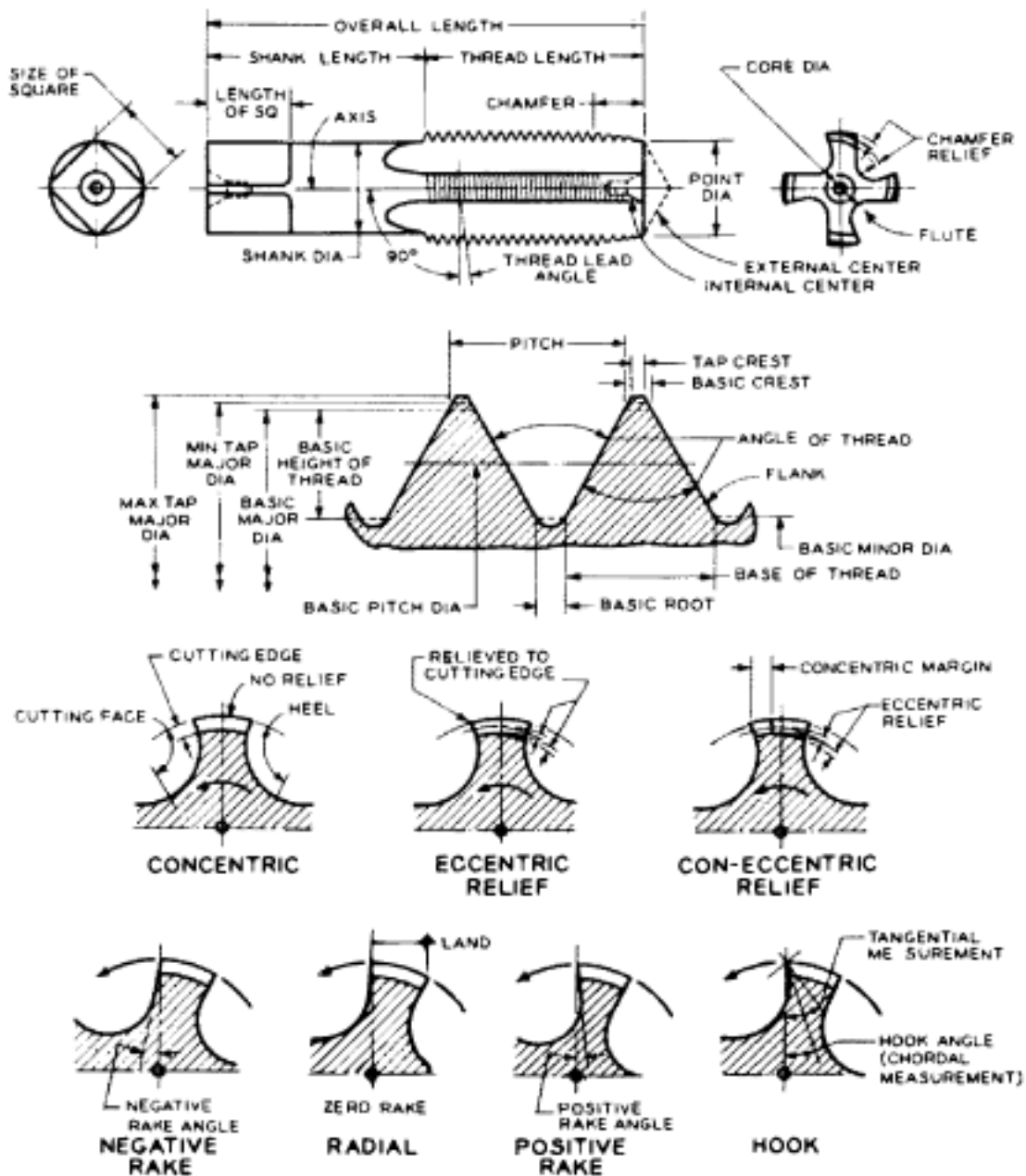


Fig. 4-16. A. Counterboring a hole. B. Countersinking a hole. C. Spot-facing a hole.

is to cut a smooth, flat surface which is perpendicular to the axis of the hole. Usually, this surface serves as a seat for the head of a cap screw or bolt; or as a seat for a nut.

Taps

The basic nomenclature of a tap is shown in Fig. 4-17. A tap is used to produce internal threads. It has thread-shaped teeth which cut a thread of similar form when it is screwed into a drilled hole. The first step in tapping an internal thread is to drill a hole with a tap drill. This is an ordinary drill that is appropriately sized so that it will leave just the right amount of metal in the hole for the tap to remove and finish.



Courtesy of the Metal Cutting Tool Institute

Fig. 4-17. Basic nomenclature of a tap.

the thread forms to required size. The tap is then screwed into the work-piece by hand, with the aid of a tap wrench, or by power, as applied by a machine tool such as a lathe or a drilling machine. The feed of the tap must always be equal to the lead of the thread being cut. The lead of a thread is the distance that it advances in one revolution. A tap will generally feed itself, once it has started, as it will tend to follow that portion of the thread that already has been cut in starting.

The size of the tap drill to be used to tap the hole is dependent upon the diameter of the thread, the size of the thread tooth (or pitch of the thread), the material of which the thread is made, the length of engage-

ment of the thread, and the available standard drill sizes. For ordinary manufacturing, the internal thread in a tapped hole need not be more than 75 or 80 per cent of the theoretical, standard thread depth. For some classes of work, not more than 50 per cent of the thread depth is required. The diameter of the tap drill should not be smaller than is necessary to give the required strength of thread, as every decrease in the diameter of the tap drill increases the power required for tapping, and, incidentally, the percentage of broken taps. Recommended tap drill sizes and hole sizes for tapping Unified Screw Threads can be found in various standard machine shop handbooks such as Machinery's Handbook.

Theoretical tap drill sizes for inch and metric threads can be calculated by the following formulas, in which the "Percent Full Thread" is expressed as a decimal; e.g., 75% is written .75 in the formula. The drill size nearest to the calculated hole size is used.

For American Unified Thread Form:

$$\text{Hole size} = \text{Basic major dia.} - \frac{1.08253 \times \text{percent full thread}}{\text{Number of threads per inch}} \quad (4-1)$$

For American National Thread form:

$$\text{Hole size} = \text{Basic major dia.} - \frac{1.299 \times \text{percent full thread}}{\text{Number of threads per inch}} \quad (4-2)$$

For ISO Metric Threads (all dimensions in millimeters):

$$\text{Hole size} = \text{Basic major dia.} - (1.08253 \times \text{pitch} \times \text{percent full thread}) \quad (4-3)$$

Example 4-1:

Find the tap drill diameter for a $\frac{1}{2}$ -13 Unified thread and for an M12-1.75 thread. The M12-1.75 thread designation means that it is metric (M), it has a diameter equal to 12 mm, and the pitch of the thread is 1.75 mm. In each case the tapped hole is to have a 60 percent full thread.

For the $\frac{1}{2}$ -13 Unified thread:

$$\text{Hole size} = .500 - \frac{1.08253 \times .60}{13} = .450 \text{ in.}$$

For the M12-1.75 metric thread:

$$\text{Hole size} = 12 - (1.08253 \times 1.75 \times .60) = 10.86 \text{ mm}$$

The nearest drill sizes are 29/64 in. and 11.0 mm, respectively.

Hand taps have a cylindrical shank and a driving square on the end to which a tap wrench can be attached. These taps are used for most machine tapping applications, as well as for hand tapping. They can be obtained as ground taps, which are finished by precision grinding, or as cut taps, which are finished by a cutting operation.

Hand taps are classified into the following types: taper, plug, and bottoming. Each type is distinguished by the amount of chamfer on the end. The angle formed between the chamfer (see Fig. 4-17) and the axis of the tap is called the chamfer angle. It determines the number of threads having their height reduced on the end of the tap. The chamfered threads are relieved behind the face and do all of the cutting to form the thread on the part. The number of flutes and the length of the chamfer determine the chip load, or the amount of material removed by each tooth.

Taper taps have a chamfer of $4\frac{1}{2}^\circ$ which reduces the height of approximately 8 to 10 threads. Plug taps have a chamfer angle of $10\frac{1}{2}^\circ$ with 3 to 5 threads reduced in height. Bottoming taps have a chamfer angle of 25° to 29° , which reduces approximately $1\frac{1}{2}$ threads. The most useful tap for general tapping applications is the plug tap. Plug taps can be used to tap through holes and also blind holes where a limited number of imperfect teeth are allowed. Bottoming taps are used to finish the bottom of blind holes where only a few imperfect teeth are permitted. Taper taps are used primarily for hand tapping thin workpieces, for tapping through holes in some abrasive materials, and for tapping the larger thread sizes and thread forms, where the chip thickness per tooth must be kept low. When a particularly hard, tough material is encountered, it is sometimes helpful to use a taper and a plug tap alternately. The taper tap starts easily but may stick after penetrating deeper; it is then removed and a plug tap is inserted to relieve the long cut made by the taper tap. For some very difficult jobs, a series of succeeding larger taps are used to tap the hole, such taps being made specially for each job. On the large majority of jobs, however, the plug tap can be used alone.

On a limited number of softer, ductile materials, threaded holes can be produced on a machine with a forming tap. This is usually associated with high volume production operations. Forming taps have no flutes of the conventional type. Instead of being round, the body of the tap is radially relieved to form a lobed construction, the number of lobes depending on the tap diameter. The nose end of the tap is tapered to allow it to enter the hole. Forming taps produce threads by plastic flow of the metal near the hole walls instead of by cutting. A chip is not formed. The thread form produced is not perfect, but usually there is no reduction in the strength of the thread.

A cutting fluid should always be used with all materials when tapping except on cast iron and plastics. When hand tapping a hole, the tap should be advanced approximately one turn and then rotated in the reverse direction until the chip is broken. By repeating this sequence until the hole is tapped, a smooth thread will be produced with minimum effort. Machine-driven taps are generally screwed into the hole while rotating in the forward direction and are reversed to remove the tap when the job

is finished. On very difficult jobs the tap is sometimes reversed when cutting into the hole in a manner similar to hand tapping.

Spade Drills

Spade drills, Fig. 4-18, are frequently used to drill large diameter holes. They are made in sizes ranging from one to five inches. A spade drill consists of a holder made from heat-treated steel and a high-speed steel, or cemented-carbide, tipped blade. The blade is seated in a slot in the end of the holder which is clamped together by a screw to secure the blade in place. Each holder can hold a range of different blade sizes. The shank of the holder may be straight or have a standard Morse taper. Some spade drill holders have coolant holes drilled lengthwise which supply a stream of coolant close to the cutting edges. Notches or chip breakers are ground on the cutting edges to reduce the width of the chip. These notches are unevenly spaced so that they do not occur at the same radial distances on the two cutting edges. Instead of a single chip, the notches cause several smaller chips to form on each cutting edge that are easier to break. A chip curling groove (chip curler) is ground on the face of each cutting edge which tends to curl and break the chips for easy disposal.

Several different types of blades are shown in Fig. 4-18 to show the versatility of this method of drilling. Besides drilling holes through solid metal, stepped holes can be drilled, and both counterboring and spot-

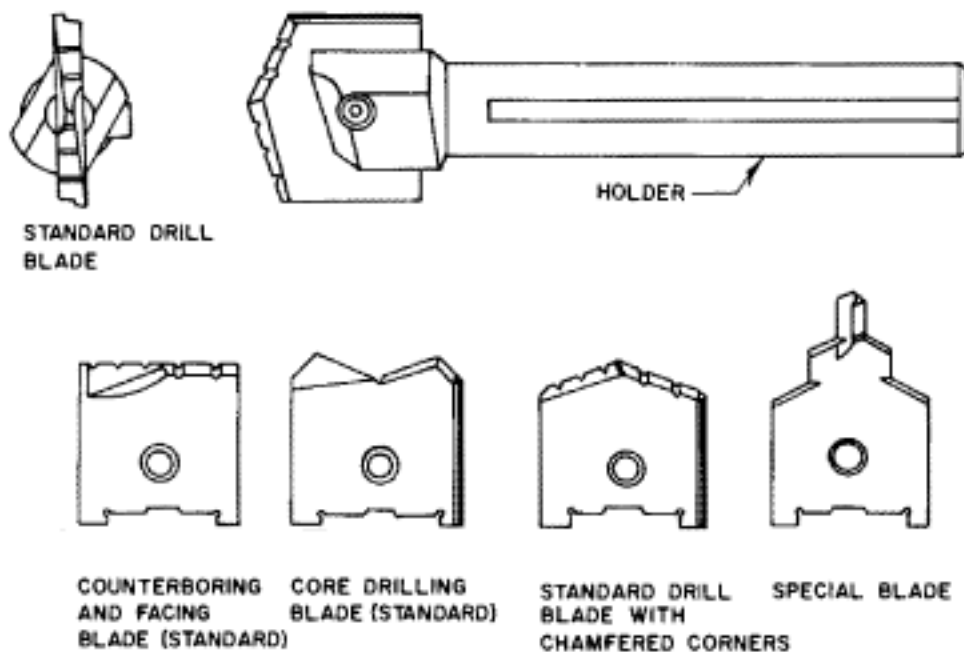


Fig. 4-18. Spade drill, including examples of standard and special blades.

facing can be done with spade drills. An important application of these drills is core drilling; i.e., enlarging an existing hole which may be a rough cored hole of a casting. The core drilling blade used for this operation has spurs ground on the end which bite into the metal and guide the drill, thereby drilling a hole that closely follows the path along which the drill is fed. Without the spurs, a core drill will be influenced by any eccentricity of the existing hole and may not drill a true hole. Core drilling operations may be performed on drilling machines having a power feed, on turret lathes, vertical turret lathes, on both horizontal and vertical boring mills, and on numerically controlled machines.

A power feed should be used when drilling with a spade drill. Only by applying extreme care and skill can spade drilling be accomplished with manual feed. The feed rate is a most important factor in successfully applying a spade drill and the constant feed rate supplied by a power feeding mechanism should be used whenever possible; for deep holes it should always be used. The feed rate should be adjusted so that the chips break into forms resembling a small "c" or number "9."

Spade drills are self starting if they are held in short stubby spade drill holders. Longer spade drills are started by first drilling a start hole or "cone" with a spade drill held in a stub holder. If the start drill is the same diameter as the second spade drill, the start drill should drill a hole that is to the full diameter of the spade drill. If the start drill is smaller than the second drill it should not drill to its full diameter, but should leave only a conical surface against which the second drill can start. Never use a center drill or drill a pilot hole with a twist drill to start a spade drill. When the spade drill contacts the sharp edge of the hole formed by these drills, it will tend to "hop around" which can cause the cutting edges of the spade drill to chip.

The size of the hole that can be drilled with a spade drill is limited only by the power and rigidity of the machine. When insufficient power is available to drill a large hole, the hole may be drilled in two passes using a smaller spade drill to drill the first hole. The power, torque, and thrust force required to feed the spade drill may be estimated by the procedures provided further on in this chapter.

Gun Drills

Gun drills are customarily used to drill deep holes where the length of the hole is five or six times its diameter. These are special drills and are usually used on special deep-hole drilling machines. They have a single flute and a single cutting edge which may be high-speed steel or a brazed-on cemented carbide tip. The cutting edge may be designed to cut to the center of the drill. Some gun drills have a cutting edge that does not cut to the center of the drill, thereby leaving a core of metal which can be easily removed if the hole is a through hole. This method of pro-

ducing, where metal is left in the center, is called *trepanning*. In either case the angles on the cutting edge are designed to make the drill produce a straight hole. The body of the gun drill is made from crimped tubing which allows a cutting fluid to be fed through the drill to an oil hole near the cutting edge. Gun drills must be started with the aid of a bushing. Once they are well started, however, they are self-supporting and will drill a straight hole. When using a gun drill in a lathe the need to use a guide bushing to start the drill, can be eliminated by first drilling and reaming a hole about one diameter deep in order to start the gun drill.

When a deep hole is required and a gun drill is not available, the hole can be drilled by brazing an extension onto the shank of a standard twist drill. The hole is first drilled about one diameter deep with a standard-length twist drill, and the remainder of the hole is drilled with the drill having the extended shank. This is a slow process because the drill must be backed out of the hole frequently, in order to remove the chips.

Estimating Drilling Thrust, Torque, and Power *

The thrust, torque, and power required to drill a hole depend on the following factors: 1. the work material, its microstructure and hardness; 2. the size of the drill; 3. the feed rate used when drilling; and, 4. the ratio expressed by the length of the chisel edge divided by the drill diameter, or c/d . In addition to these factors the power required to drill a hole is also dependent on the rotational speed (rpm) of the drill or work, whichever is rotating.

In the customary inch system of units the measure of power is the horsepower. Horsepower is not used in the SI metric system of units. In its place the kilowatt is used for both mechanical and electrical power. For all machining operations, including drilling, it is often desirable to have knowledge of the power used directly by the cutting tool in producing the chip, as well as of the power required by the driving motor to drive the machine tool. The machine tool driving motor must supply the power to drive the machine in addition to the power required by the cutting tool to cut the material. How efficiently the machine tool transmits the power from the driving motor to the cutting tool is measured by the machine tool efficiency factor, E . Typical values of this factor are provided in Table 4-8. Although the feeding force, called the *thrust force* in drilling, may be quite large, the power required to feed is usually very low because the feed velocity is usually very low; therefore, the power required for feeding can be disregarded.

The values in Table 4-3 through 4-6 are for sharp drills. All metal

* The formulas and tables in this section are based on research and experimental work conducted by Carl J. Oxford Jr., Vice President, National Twist Drill Division, Lear Siegler Inc., and by Dr. Milton C. Shaw of the Carnegie-Mellon Institute.

Table 4-3. Work Material Factor, K_d , for Drilling with a Sharp Drill

Work Material	Work Material Constant, K_d
AISI 1117 (Resulfurized free machining mild steel)	12,000
Steel, 200 HB	24,000
Steel, 300 HB	31,000
Steel, 400 HB	34,000
Cast Iron, 150 HB	14,000
Most Aluminum Alloys	7,000
Most Magnesium Alloys	4,000
Most Brasses	14,000
Leaded Brass	7,000
Austenitic Stainless Steel (Type 316)	24,000* for Torque 35,000* for Thrust
Titanium Alloy Ti6Al 4V 40R _c	18,000* for Torque 29,000* for Thrust
Rene 41 40R _c	40,000*† min.
Hastelloy-C	30,000* for Torque 37,000* for Thrust

* Values based upon a limited number of tests.

† Will increase with rapid wear.

Table 4-4. Feed Factors, F_f , for Drilling

Inch Units				SI Metric Units			
Feed, in./rev	F_f	Feed, in./rev	F_f	Feed, mm/rev	F_f	Feed, mm/rev	F_f
.0005	.0023	.012	.029	0.01	.025	0.30	.382
.001	.004	.013	.031	0.03	.060	0.35	.432
.002	.007	.015	.035	0.05	.091	0.40	.480
.003	.010	.018	.040	0.08	.133	0.45	.528
.004	.012	.020	.044	0.10	.158	0.50	.574
.005	.014	.022	.047	0.12	.183	0.55	.620
.006	.017	.025	.052	0.15	.219	0.65	.708
.007	.019	.030	.060	0.18	.254	0.75	.794
.008	.021	.035	.068	0.20	.276	0.90	.919
.009	.023	.040	.076	0.22	.298	1.00	1.000
.010	.025	.050	.091	0.25	.330	1.25	1.195

cutting tools wear as they are used and are expected to cut while having a certain amount of wear at the cutting edge. As the cutting edge becomes worn, more power is required to cut the material, the power required increasing with increased tool wear. A tool wear factor, provided in Table 4-7, is used in order to compensate for the wear on the cutting edge. Compensation factors for the chisel edge ratio, c/d , are given in Table 4-6. Since it is usually more convenient to accurately measure the web thickness at the drill point than to measure the length of the chisel edge, the approximate ratio of the web thickness at the drill point to the drill

Table 4-5. Drill Diameter Factors: F_T for Thrust; F_M for Torque

Inch Units						SI Metric Units					
Drill Diam., in.	F_T	F_M	Drill Diam., in.	F_T	F_M	Drill Diam., mm	F_T	F_M	Drill Diam., mm	F_T	F_M
.063	.110	.007	.875	.899	.786	1.60	1.46	2.33	22.00	11.86	260.8
.094	.151	.014	.938	.950	.891	2.40	2.02	4.84	24.00	12.71	305.1
.125	.189	.024	1.000	1.000	1.000	3.20	2.54	8.12	25.50	13.34	340.2
.156	.226	.035	1.063	1.050	1.116	4.00	3.03	12.12	27.00	13.97	377.1
.188	.263	.049	1.125	1.099	1.236	4.80	3.51	16.84	28.50	14.58	415.6
.219	.297	.065	1.250	1.195	1.494	5.60	3.97	22.22	32.00	16.00	512.0
.250	.330	.082	1.375	1.290	1.774	6.40	4.42	28.26	35.00	17.19	601.6
.281	.362	.102	1.500	1.383	2.075	7.20	4.85	34.93	38.00	18.36	697.6
.313	.395	.124	1.625	1.475	2.396	8.00	5.28	42.72	42.00	19.89	835.3
.344	.426	.146	1.750	1.565	2.738	8.80	5.66	50.13	45.00	21.02	945.8
.375	.456	.171	1.875	1.653	3.100	9.50	6.06	57.53	48.00	22.13	1062
.438	.517	.226	2.000	1.741	3.482	11.00	6.81	74.90	50.00	22.86	1143
.500	.574	.287	2.250	1.913	4.305	12.50	7.54	94.28	58.00	25.75	1493
.563	.632	.355	2.500	2.081	5.203	14.50	8.49	123.1	64.00	27.86	1783
.625	.687	.429	2.750	2.246	6.177	16.00	9.19	147.0	70.00	29.93	2095
.688	.741	.510	3.000	2.408	7.225	17.50	9.87	172.8	76.00	31.96	2429
.750	.794	.596	3.500	2.724	9.535	19.00	10.54	200.3	90.00	36.53	3293
.813	.847	.689	4.000	3.031	12.13	20.00	10.98	219.7	100.00	39.81	3981

Table 4-6. Chisel Edge Factors for Torque and Thrust

c/d	Approx. w/d	Torque Factor A	Thrust Factor B	Thrust Factor J	c/d	Approx. w/d	Torque Factor A	Thrust Factor B	Thrust Factor J
.03	.025	1.000	1.100	.001	.18	.155	1.085	1.355	.030
.05	.045	1.005	1.140	.003	.20	.175	1.105	1.380	.040
.08	.070	1.015	1.200	.006	.25	.220	1.155	1.445	.065
.10	.085	1.020	1.235	.010	.30	.260	1.235	1.500	.090
.13	.110	1.040	1.270	.017	.35	.300	1.310	1.575	.120
.15	.130	1.080	1.310	.022	.40	.350	1.395	1.620	.160

For drills of standard design, use $c/d = .18$.

For split point drills, use $c/d = .03$.

c/d = Length of Chisel Edge $\div$ Drill Diameter.

w/d = Web Thickness at Drill Point $\div$ Drill Diameter.

diameter, or w/d , is given for each c/d ratio in Table 4-6. The c/d ratio for all standard twist drills is .18, unless the drill has been ground so that the length is short or the web has been thinned; the c/d ratio for a split point drill is .03. Cutting fluids may reduce the thrust, torque, and power required to drill, however, no specific recommendation can be given since each cutting fluid has its own characteristics.

The formulas below are used to calculate the thrust, torque, and power for drilling. Since conditions may vary in different shops and machine tools are not all designed alike, the calculated results may not correspond precisely with the results obtained on the job; however, they do provide a

reasonable estimate which will be close enough in most practical situations.

For inch units only:

$$T = 2 K_d F_f F_T B W + K_d d^2 J W \quad (4-1)$$

$$M = K_d F_f F_M A W \quad (4-2)$$

$$P_c = \frac{M N}{63,025} \quad (4-3)$$

For SI metric units only:

$$T = 0.05 K_d F_f F_T B W + 0.007 K_d d^2 J W \quad (4-4)$$

$$M = \frac{K_d F_f F_M A W}{40,000} = 0.000025 K_d F_f F_M A W \quad (4-5)$$

$$P_c = \frac{M N}{9550} \quad (4-6)$$

For either inch or metric units:

$$P_m = \frac{P_c}{E} \quad (4-7)$$

where:

P_c = Power at the cutter; hp, or kW

P_m = Power at the motor; hp, or kW

M = Torque; in. lb, or N·m

T = Thrust; lb, or N

K_d = Work material factor

(See Table 4-3)

F_f = Feed factor

(See Table 4-4)

F_T = Thrust factor for drill diameter

(See Table 4-5)

F_M = Torque factor for drill diameter

(See Table 4-5)

A = Chisel edge factor for torque

(See Table 4-6)

B = Chisel edge factor for thrust

(See Table 4-6)

J = Chisel edge factor for thrust

(See Table 4-6)

W = Tool wear factor

(See Table 4-7)

N = Spindle speed; rpm

E = Machine tool efficiency factor

(See Table 4-8)

d = Drill diameter; in., or mm

c = Chisel edge length; in., or mm

(See Table 4-6)

w = Web thickness at drill point; in., or mm

(See Table 4-6)

Example 4-1:

A standard 1 inch diameter twist drill operating at 150 rpm and with a feed rate of .008 in./rev is used to drill 200 HB steel on a drill press having an efficiency of .80. Estimate the thrust, torque, power at the motor, and the power required to feed the drill through the work, P_f .

Table 4-7. Tool Wear Factors, W

Type of Drilling Operation	W
Drilling with a sharp drill	1.00
Normal drilling	1.30
Drilling hard-to-machine materials and drilling with a very dull drill	1.50

Table 4-8. Machine Tool Efficiency Factor, E

Type of Drive	E	Type of Drive	E
Direct Belt Drive	.90	Geared Head Drive	.70-.80
Back Gear Drive	.75		

$$\begin{array}{ll}
 K_d = 24,000 & \text{(From Table 4-3)} \\
 F_f = .021 & \text{(From Table 4-4)} \\
 F_T = 1.000 & \text{(From Table 4-5)} \\
 F_M = 1.000 & \text{(From Table 4-5)}
 \end{array}
 \qquad
 \begin{array}{ll}
 A = 1.085 & \text{(From Table 4-6)} \\
 B = 1.355 & \text{(From Table 4-6)} \\
 J = .030 & \text{(From Table 4-6)} \\
 W = 1.30 & \text{(From Table 4-7)}
 \end{array}$$

$$\begin{aligned}
 T &= 2 K_d F_f F_T B W + K_d d^2 J W \\
 &= 2 \times 24,000 \times .021 \times 1.000 \times 1.355 \times 1.30 + 24,000 \times 1.000^2 \times .030 \times 1.30 \\
 &= 2710 \text{ lbs} \quad (\text{rounded})
 \end{aligned}$$

$$\begin{aligned}
 M &= K_d F_f F_M A W \\
 &= 24,000 \times .021 \times 1.000 \times 1.085 \times 1.30 \\
 &= 710 \text{ lb in.} \quad (\text{rounded})
 \end{aligned}$$

$$\begin{aligned}
 P_c &= \frac{M N}{63,025} = \frac{710 \times 150}{63,025} \\
 &= 1.7 \text{ hp} \quad (\text{rounded})
 \end{aligned}$$

$$\begin{aligned}
 P_M &= \frac{P_c}{E} = \frac{1.7}{.80} \\
 &= 2.1 \text{ hp} \quad (\text{rounded})
 \end{aligned}$$

Let f equal the feed rate and P_f equal the power required to feed the drill.

$$f = .008 \times 150 = 1.2 \text{ in./min}$$

$$f = \frac{1.2}{12}$$

$$f = .1 \text{ ft/min}$$

$$\begin{aligned}
 P_f &= \frac{T f}{33,000} = \frac{2710 \times .1}{33,000} \\
 &= .008 \text{ hp}
 \end{aligned}$$

Example 4-2:

A 50 mm diameter spade drill which is 8 mm thick is to drill 150 HB gray cast iron using a feed rate of .20 mm/rev and a spindle speed of 200 rpm on a lathe having a machine tool efficiency factor of .80. The thickness of the spade drill at the point is 5.50 mm. Estimate the thrust and the power required to perform this operation.

$$K_d = 14,000 \quad (\text{From Table 4-3}) \quad A = 1.040 \quad (\text{From Table 4-6})$$

$$F_f = .276 \quad (\text{From Table 4-4}) \quad B = 1.270 \quad (\text{From Table 4-6})$$

$$F_r = 22.86 \quad (\text{From Table 4-5}) \quad J = .017 \quad (\text{From Table 4-6})$$

$$F_M = 1143 \quad (\text{From Table 4-5}) \quad W = 1.30 \quad (\text{From Table 4-7})$$

$$w/d = 5.5/50 = .110$$

$$T = 0.05 K_d F_f F_r B W + 0.007 K_d d^2 J W$$

$$= 0.05 \times 14,000 \times .276 \times 22.86 \times 1.270 \times 1.30 + 0.007 \times 14,000 \times 50^2 \times .017 \times 1.30$$

$$= 12,706 \text{ N} \quad (12,706 \times .2248 = 2856 \text{ lb})$$

$$M = \frac{K_d F_f F_M A W}{40,000} = \frac{14,000 \times .276 \times 1143 \times 1.040 \times 1.30}{40,000}$$

$$= 150 \text{ N} \cdot \text{m} \quad (\text{rounded})$$

$$P_c = \frac{M N}{9550} = \frac{150 \times 200}{9550}$$

$$= 3.14 \text{ kW} \quad (3.14 \times 1.341 = 4.2 \text{ hp})$$

$$P_m = \frac{P_c}{E} = \frac{3.14}{.80}$$

$$= 3.92 \text{ kW} \quad (3.92 \times 1.341 = 5.26 \text{ hp})$$

Drilling Machine Operation

Work on machine tools is accomplished in the following sequence:

1. Plan how to do the job.
2. Obtain all the tools and materials required to do the job.
3. Prepare the workpiece and the machine for the machining operation.
4. Start the job, proceed with thought and care, and get the job done.

Although the basic principles of machine tool operation are unvarying, the manner in which they are applied can vary from shop to shop. The reason for this is that the resources (men, tools, machine tools, and experience) of each individual shop are different. The skilled craftsman adapts his knowledge and skills to the resources that are available in the shop where he is employed. It is, therefore, not unusual for the same job to be done differently in various shops; however, regardless of the method used, the basic principles must be applied to each method. In this book the basic principles are emphasized. The student must, however, be prepared to adapt them to the circumstances with which he is confronted.

The object of all drilling machine work is to machine holes to a given size having an acceptable surface finish and located exactly where required on the workpiece. Drilling machines, or drill presses as they are also called, are primarily hole-producing machines unlike lathes or milling machines which machine a variety of shapes including holes.

A great many drilling jobs are preceded by a layout operation on the workpiece. Layout work was covered in Chapter 3 and will not be discussed in this chapter. Where a layout is required, it is assumed that this work has been done prior to the discussion in this chapter.

Drilling Speeds

For any machining operation, including drilling, the cutting speed is generally given in terms of feet per minute, which is abbreviated: fpm. It describes the rate of movement of a cutting tool relative to the workpiece. For example, a cutting speed of 100 fpm would describe a cutting tool traveling in a straight line at a uniform rate of motion, or speed, such that the tool would travel a distance of 100 feet in one minute. The effect would be the same if the tool was stationary and the workpiece

moved past the tool at this speed. In drilling, the tool moves, but it rotates instead of moving in a straight line. The surface on the lips adjacent to the margin is the part of the drill traveling at the highest speed when the drill is rotating. This surface travels a distance equal to the circumference of the drill, or $\pi \times$ drill diameter, every time that it makes one complete turn. Assume for a moment that the drill diameter is $\frac{1}{2}$ inch, its circumference would be $\frac{1}{2} \times 3.1416 = 1.57$ inches. Now, if this drill can cut at a cutting speed of 100 fpm, or $12 \times 100 = 1200$ inches per minute, the number of circumferences that will fit into this length is $\frac{1,200}{1.57}$ or 764 circumferences per minute. Since every circumference also represents one complete revolution of the drill, the $\frac{1}{2}$ -inch drill must turn 764 revolutions per minute if the outer edge of the lip is to move a distance of 1,200 inches per minute, or 100 feet per minute. To summarize this as a formula:

$$N = \frac{12 V}{\pi D} \quad (5-1)$$

- where: N = Spindle speed of the drill press, rpm
 (This is the speed at which the twist drill will operate.)
 V = Cutting speed in feet per minute (fpm), at which the drill can cut the work material.
 π = pi = 3.1416. The mathematical symbol for this number which is a constant for calculating the circumference of a circle.
 D = Diameter of the drill, in inches.

Some drilling machines have infinitely variable speeds within a range of speeds while others have discrete speeds which are in steps, such as are available in a geared head or on a step cone pulley. When a machine is equipped with a stepped range of speeds, it should be set at the speed that is the nearest to that calculated in Formula 5-1.

Example 5-1:

Two holes are to be drilled in a part made from AISI 4140 steel that has been quenched and tempered to have a hardness of 270 HB (Brinell hardness number). The diameter of one hole is $\frac{1}{8}$ inch and the diameter of the second hole is 1 inch. In Table 5-1 the cutting speed for this material is shown to be 60 fpm when the drill is made from high-speed steel.

A. Calculate the cutting speed for both drills.

1. For the $\frac{1}{8}$ inch drill:

$$N = \frac{12 V}{\pi D} = \frac{12 \times 60}{\pi \times 1/8} = 1833 \text{ rpm}$$

2. For the 1 inch drill:

$$N = \frac{12 \times 60}{\pi \times 1} = 229 \text{ rpm}$$

B. The drill press has the following speeds (rpm): 1500, 800, 500, 400, 250, and 150. What speeds should be selected to drill the holes?

1. For the $\frac{1}{8}$ inch drill—1500 rpm
2. For the 1 inch drill—250 rpm

The cutting speed in the metric system is given in terms of meters per minute (m/min) and the drill diameter is in millimeters (mm). Since the units are different, the cutting speed formula using SI units is different from the formula (5-1) in which customary inch units are used.

$$N = \frac{1000 V}{\pi D} \quad (5-2)$$

Where:

N = Revolutions per minute (rpm) at which the drill must operate.

V = Cutting speed, m/min

π = pi = 3.1416

D = Drill diameter, mm

Example 5-2:

An annealed die made from 01 oil hardening steel must be drilled with a 19.05 mm twist drill. Calculate the spindle speed of the drill press.

The cutting speed for 01 annealed oil hardening steel is 45 fpm (see Table 5-2), or $45 \times .3048 \approx 14$ m/min.

$$\begin{aligned} N &= \frac{1000 V}{\pi D} = \frac{1000 \times 14}{\pi \times 19.05} \\ &= 234 \text{ rpm} \end{aligned}$$

The recommended cutting speeds for drilling and reaming different materials with high-speed steel drills are given in Tables 5-1 through 5-5. The cutting speed is dependent on the material from which the drill or reamer is made, the material from which the workpiece is made, the hardness of the work material and its heat treatment. Another factor affecting the cutting speed is the length of time desired during which the tool will perform satisfactorily, called "tool life." Generally, a cutting tool operating at a lower cutting speed will last longer than when operating at a higher cutting speed. Twist drills are made from several types of high-speed steel; and less expensive drills, usually intended for home workshop use, are made from carbon tool steel. Carbon tool steel drills can only be operated at one-half of the cutting speeds listed in these tables.

Except when using high production and numerically controlled machines, drills less than approximately $\frac{1}{4}$ inch, or 6.4 mm, are usually fed

Table 5-1. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Plain Carbon and Alloy Steels

Material AISI and SAE Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm		
			Drilling	Reaming	
			HSS	HSS	Carbide
Free Machining Plain Carbon Steels (Resulphurized) 1212, 1213, 1214 1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211 1132, 1137, 1139, 1140, 1144, 1146, 1151	100-150	HR, A	120	80	400
	150-200	CD	125	80	350
	100-150	HR, A	110	75	375
	150-200	CD	120	80	350
	175-225	HR, A, N, CD	100	65	350
	275-325	Q and T	70	45	250
	325-375	Q and T	45	30	175
	375-425	Q and T	35	20	100
	100-150	HR, A, N, CD	130	85	400
	150-200	HR, A, N, CD	120	80	375
	200-250	N, CD	90	60	275
	100-125	HR, A, N, CD	100	65	300
Plain Carbon Steels, 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514 1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541 1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125-175	HR, A, N, CD	90	60	275
	175-225	HR, N, CD	70	45	200
	225-275	CD	60	40	175
	125-175	HR, A, N, CD	90	60	250
	175-225	HR, A, N, CD	75	50	200
	225-275	N, CD, Q and T	60	40	150
	275-325	Q and T	50	30	120
	325-375	Q and T	35	20	100
	375-425	Q and T	25	15	80
	125-175	HR, A, N, CD	85	55	250
	175-225	HR, A, N, CD	70	45	200
	225-275	N, CD, Q and T	50	30	140
	275-325	Q and T	40	25	110
	325-375	Q and T	30	20	90
	375-425	Q and T	15	10	70
	175-200	HR, A, N, CD	90	60	250
	200-250	HR, N, CD	80	50	225
	250-300	Q and T	55	30	200
	300-375	Q and T	40	25	150
	375-425	Q and T	30	15	100
	150-200	HR, A, N, CD	100	65	285
	200-250	HR, N, CD	90	60	250
	250-300	Q and T	65	40	200
	300-375	Q and T	45	30	150
	375-425	Q and T	30	15	110
Free Machining Alloy Steels (Resulphurized), 4140, 4150	125-175	HR, A, N, CD	85	55	250
	175-225	HR, N, CD	70	45	225
	225-275	CD, N, Q and T	55	35	200
	275-325	Q and T	50	30	150
	325-375	Q and T	35	25	125
	375-425	Q and T	25	15	90
Free Machining Alloy Steels (Leaded), 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 51L100, 86L20, 86L40	125-175	HR, A, N, CD	85	55	250
	175-225	HR, N, CD	70	45	225
	225-275	CD, N, Q and T	55	35	200
	275-325	Q and T	50	30	150
	325-375	Q and T	35	25	125
	375-425	Q and T	25	15	90
Alloy Steels, 4012, 4023, 4024, 4028, 4118, 4120, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5017, 5020, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8620, 8822, 94B17	125-175	HR, A, N, CD	85	55	250
	175-225	HR, N, CD	70	45	225
	225-275	CD, N, Q and T	55	35	200
	275-325	Q and T	50	30	150
	325-375	Q and T	35	25	125
	375-425	Q and T	25	15	90

* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; HB, Brinell hardness number.

Table 5-1 (cont). Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Plain Carbon and Alloy Steels

Material AISI and SAE Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm		
			Drilling	Reaming	
			HSS	HSS	Carbide
Alloy Steels, 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4160, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30	175-225 225-275 275-325 325-375 375-425	HR, A, N, CD N, CD, Q and T N, Q and T N, Q and T Q and T	75 60 45 30 20	50 40 30 15 15	200 175 150 100 80
Alloy Steels, E51100, E52100	175-225 225-275 275-325 325-375 375-425	HR, A, CD N, CD, Q and T N, Q and T N, Q and T Q and T	60 50 35 30 20	40 30 25 20 10	200 125 100 80 50
Ultra High Strength Steels (Not AISI), AMS6424, AMS6421 (98B37 Mod.), AMS6422 (98BV40), AMS6427, AMS6428, AMS6430, AMS6432, AMS6433, AMS6434, AMS6436, AMS6442, 300M, D6ac	220-300 300-350 350-400	A N N	50 35 20	30 20 10	180 125 90
Maraging Steels (Not AISI) 18% Nickel Grade 200 18% Nickel Grade 250 18% Nickel Grade 300 18% Nickel Grade 350	250-225	A	50	30	175
Nitriding Steels Nitalloy 125 Nitalloy 135 Nitalloy 135 (Mod.) Nitalloy 225 Nitalloy 230 Nitalloy N Nitalloy EZ Nitrex 1	200-250 250-300	A N, Q and T	60 35	40 20	175 125

* Abbreviations designate: A, annealed; Q and T, quenched and tempered; HB, Brinell hardness number.

through the workpiece using the manual controls on the drilling machine, because the operator can quickly develop a sense of feel and sight that will provide the correct feed rate and prevent drill breakage. Even drills up to one-half inch, or 13 mm, are often preferably fed "by hand." When a power feed is available on the machine, larger drills should be fed using the power feed, although on some occasions smaller drills are also fed by power, as mentioned before. The feed rate that should be used is dependent on the workpiece material (type, hardness, and heat treatment), the size of the drill, and the rigidity of the machine, the workpiece, and the setup. All of these factors must be considered in selecting the feed rate. As a

guide, Table 5-6 is given, which provides a range of feed rates recommended for different size twist drills.

The conditions influencing the cutting speed and feed may vary in different shops. Moreover, all of the factors may not be known; e.g., the hardness and heat treatment of the work material may not be known exactly. In such cases judgment should be used in applying the values given in the tables. These values do, however, provide a good starting point from which changes can be made as experience is gained.

Table 5-2. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Tool Steels

Material Tool Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm		
			Drilling	Reaming	
			HSS	HSS	Carbide
Water Hardening W1, W2, W5	150-200	A	85	55	200
Shock Resisting S1, S2, S5, S6, S7	175-225	A	50	35	175
Cold Work, Oil Hardening O1, O2, O6, O7	175-225	A	45	30	150
Cold Work, High Carbon High Chromium, D2, D3, D4, D5, D7	200-250	A	30	20	80
Cold Work, Air Hardening A2, A3, A8, A9, A10 A4, A6 A7	200-250	A	50	35	150
	200-250	A	45	30	125
	225-275	A	30	20	100
Hot Work, Chromium Type H10, H11, H12, H13, H14, H19	150-200	A	60	40	200
	200-250	A	50	30	150
	325-375	Q and T	30	20	100
Hot Work, Tungsten Type H21, H22, H23, H24, H25, H26	150-200	A	55	35	150
	200-250	A	40	25	125
Hot Work, Molybdenum Type H41, H42, H43	150-200	A	45	30	150
	200-250	A	35	20	125
Special Purpose, Low Alloy L2, L3, L6	150-200	A	60	40	200
Mold P2, P3, P4, P5, P6 P20, P21	100-150	A	75	50	225
	150-200	A	60	40	200
High-Speed Steel M1, M2, M6, M10, T1, T2, T6 M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8 T15, M3-2	200-250	A	45	30	150
	225-275	A	35	20	100
	225-275	A	25	15	80

* Abbreviations designate: A, annealed; Q and T, quenched and tempered.

Table 5-3. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Stainless Steels

Material Stainless Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm		
			Drilling	Reaming	
			HSS	HSS	Carbide
Free Machining Stainless Steels					
(Ferritic), 430F, 430F Se	135-185	A	90	60	250
(Austenitic), 203EZ, 303, 303 Se, 303 MA, 303 Pb, 303 Cu, 303 Plus X	135-185	A	85	55	225
	225-275	CD	70	45	200
(Martensitic), 416, 416 Se, 416 Plus X, 420F, 420F Se, 440F, 440F Se	135-185	A	90	60	250
	185-240	CD	70	45	200
	275-325	Q and T	40	25	150
	375-425	Q and T	20	10	80
Stainless Steels					
(Ferritic), 405, 409, 429, 430, 434, 436, 442, 446, 502	135-185	A	65	45	200
(Austenitic), 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	135-185	A	55	35	150
	225-275	CD	50	30	125
(Austenitic), 302B, 309, 309S, 310, 310S, 314, 316, 316L, 317, 330	135-185	A	50	30	150
(Martensitic), 403, 410, 420, 501	135-175	A	75	50	225
	175-225	A	65	45	200
	275-325	Q and T	40	25	125
	375-425	Q and T	25	15	80
(Martensitic), 414, 431 Greek Ascoloy	225-275	A	50	30	150
	275-325	Q and T	40	25	125
	375-425	Q and T	25	15	80
(Martensitic), 440A, 440B, 440C	225-275	A	45	30	125
	275-325	* Q and T	40	25	100
	375-425	Q and T	20	10	75
(Precipitation Hardening)					
15-5PH, 17-4PH, 17-7Ph, 17-14Cu Mo, AF-71, AFC-77, AM-350, AM-355, AM-362, Custom 455, HNM, PH13-8, PH14-8Mo, PH15-7Mo, Stainless W	150-200	A	50	30	150
	275-325	H	45	25	125
	325-375	H	35	20	75
	375-425	H	20	10	50

* Abbreviations designate: A, annealed; CD, cold drawn; Q and T, quench and tempered; H, precipitation hardened; and HB, Brinell hardness number.

Setting Up for Drilling

Before any metal cutting can be done on any type of machine tool, including drilling machines, the workpiece must be set up on the machine. The setup and the cutting operation must both be accurate to produce an accurate part. Setting up means aligning the part on the machine and then clamping it in place or holding it securely by other

Table 5-4. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Ferrous Cast Metals

Material Ferrous Cast Metals	Hardness, HB*	Material Condition*	Cutting Speed, fpm		
			Drilling	Reaming	
			HSS	HSS	Carbide
Gray Cast Iron					
ASTM Class 20	120-150	A	100	65	300
ASTM Class 25	160-200	AC	90	60	225
ASTM Class 30, 35, and 40	190-220	AC	80	55	180
ASTM Class 45 and 50	220-260	AC	60	40	125
ASTM Class 55 and 60	250-320	AC, HT	30	20	80
ASTM Type 1, 1b, 5 (Ni-Resist)	100-215	AC	50	30	150
ASTM Type 2, 3, 6 (Ni-Resist)	120-175	AC	40	25	140
ASTM Type 2b, 4 (Ni-Resist)	150-250	AC	30	20	125
Malleable Iron					
(Ferritic), 32510, 35018	110-160	MHT	110	75	325
(Pearlitic), 40010, 43010, 45006, 45008, 48005, 50005	160-200 200-240	MHT MHT	80 70	55 45	250 180
(Martensitic), 53004, 60003, 60004	200-255	MHT	55	35	160
(Martensitic), 70002, 70003	220-260	MHT	50	30	150
(Martensitic), 80002	240-280	MHT	45	30	100
(Martensitic), 90001	250-320	MHT	25	15	80
Nodular (Ductile) Iron					
(Ferritic), 60-40-18, 65-45-12	140-190	A	100	65	300
(Ferritic-Pearlitic), 80-55-06	190-225 225-260	AC AC	70 50	45 30	225 140
(Pearlitic-Martensitic), 100-70-03	240-300	HT	40	25	115
(Martensitic), 120-90-02	270-330 330-400	HT HT	25 10	15 5	65 30
Cast Steels					
(Low Carbon), 1010, 1020	100-150	AC, A, N	100	65	250
(Medium Carbon), 1030, 1040, 1050	125-175 175-225 225-300	AC, A, N AC, A, N AC, HT	90 70 55	60 45 35	240 220 200
(Low Carbon Alloy), 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150-200 200-250 250-300	AC, A, N AC, A, N AC, HT	75 65 50	50 40 30	220 200 150
(Medium Carbon Alloy), 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175-225 225-250 250-300 300-350 350-400	AC, A, N AC, A, N AC, HT AC, HT HT	70 60 45 30 20	45 35 30 20 10	200 180 150 140 100

* Abbreviations designate: A, annealed; AC, as cast; N, normalized; HT, heat treated; MHT, malleablizing heat treatment; and HB, Brinell hardness number.

Table 5-5. Recommended Cutting Speeds in Feet per Minute for Drilling and Reaming Light Metals and Copper Alloys

Material	Material Condition*	Cutting Speed, fpm		
		Drilling	Reaming	
		HSS	HSS	Carbide
Light Metals and Copper Alloys				
All Wrought Aluminum Alloys {	CD	400	300	800
	ST and A	350	275	750
All Aluminum Sand and Permanent Mold Casting Alloys *	AC	500	350	900
	ST and A	350	275	750
*Except Alloys 390.0 and 392.0 {	AC	125	100	250
	ST and A	45	40	200
All Wrought Magnesium Alloys	A, CD, ST and A	500	350	1000
All Cast Magnesium Alloys	A, AC, ST and A	450	375	1000
All Soft Brasses and Bronzes {	A	160	160	320
	CD	175	175	360
All Medium Brasses and Bronzes {	A	120	110	250
	CD	140	120	275
All Hard Brasses and Bronzes {	A	60	50	180
	CD	65	60	200

* Abbreviations designate: A, annealed; CD, cold drawn; AC, as cast; ST and A, solution treated and aged.

Table 5-6. Recommended Feeds in Inches per Revolution for High-Speed Steel Twist Drills

Drill Diameter, inch	Feed, in./rev	Drill Diameter, inch	Feed, in./rev
$\frac{1}{16}$ to $\frac{1}{8}$	.001-.003	Over $\frac{1}{2}$ to $\frac{3}{4}$	.005-.012
Over $\frac{1}{8}$ to $\frac{1}{4}$	.002-.006	Over $\frac{3}{4}$ to 1	.006-.015
Over $\frac{1}{4}$ to $\frac{1}{2}$	.004-.010	Over 1	.010-.025

means. In the case of drilling, the workpiece may or may not be clamped to the drill press table, depending on the size and shape of the workpiece and the size of the largest drill to be used. If the workpiece is massive it may simply be placed on the drill press table or base plate, as shown in Fig. 4-4. In such cases, the friction resulting from the mass is sufficient to resist any movement of the workpiece caused by the drill torque. Small and medium size workpieces may be held against the table by hand—but not in the hand—or they may be clamped to the table. The

decision on whether to hold the workpiece against the table by hand or whether to clamp it is dependent on the ability to safely resist the torque of the largest drill to be used. When making this decision, it is always best to err in the direction of safety; i.e., when in doubt, clamp the workpiece.

Large size drills (approximately $\frac{5}{8}$ in. or 15 mm and larger) exert a very high torque which can only be resisted safely by clamping the workpiece to the table, unless the mass of the workpiece is large enough to withstand the torque without movement. The most common method of clamping is with U-shaped strap clamps, such as shown in Fig. 5-3C. The principles of applying strap clamps will now be explained. These principles apply not only to the application of strap clamps on drilling machines, but also on other machine tools such as shapers, planers, milling machines, horizontal and vertical boring machines, and on lathe faceplates.

When clamping a workpiece, the first step is to determine where the strap clamps should be placed. They must always be positioned so that the surface or surfaces to be machined can be reached by the cutting tools without interference from the clamps. Suitable surfaces on the workpiece are then selected on the basis of the ability to hold the part

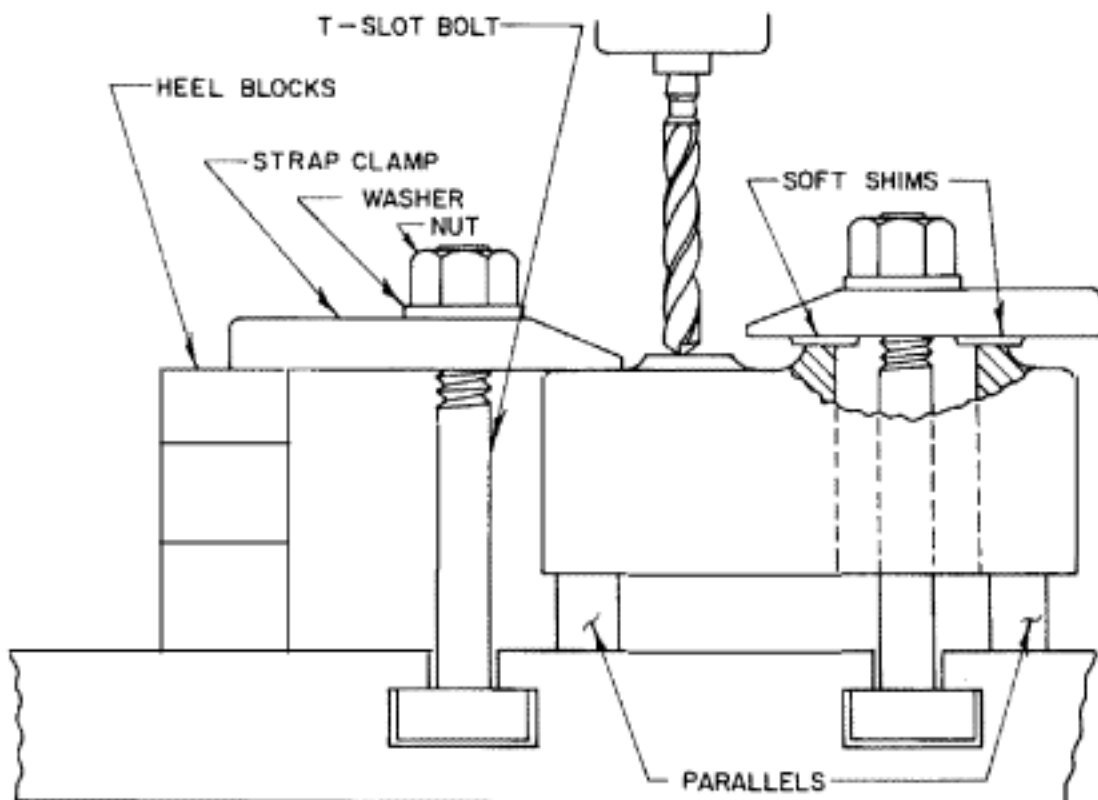


Fig. 5-1. Typical drilling machine setup using strap clamps and T-slot bolts.

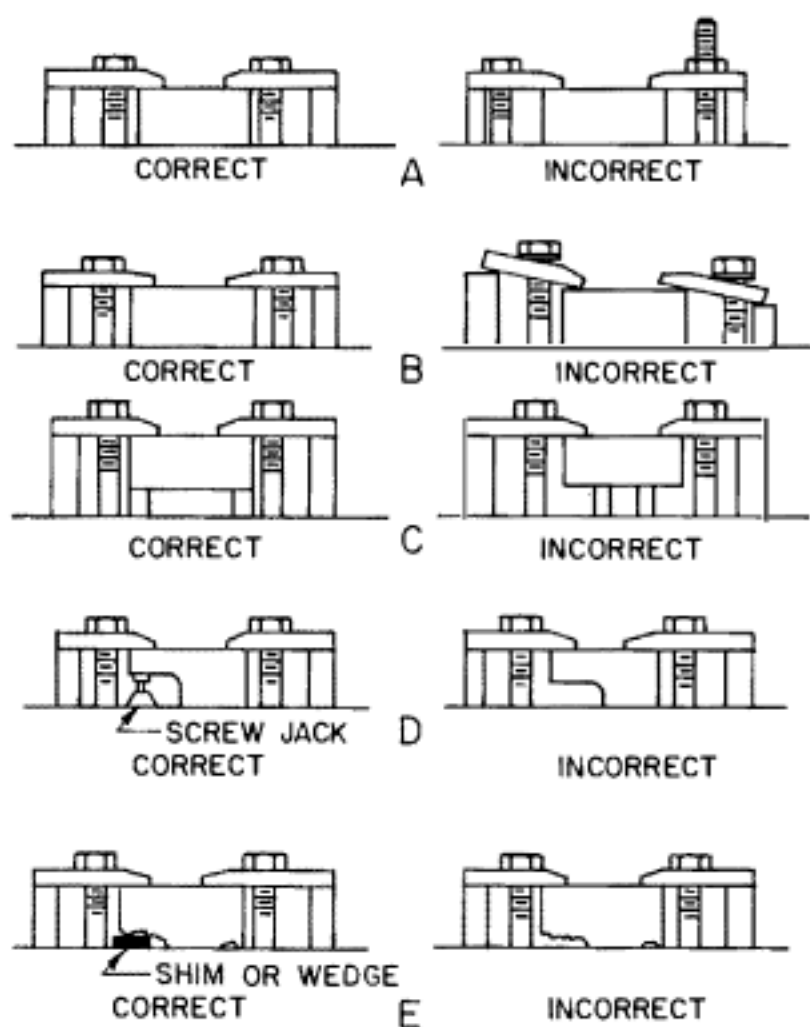
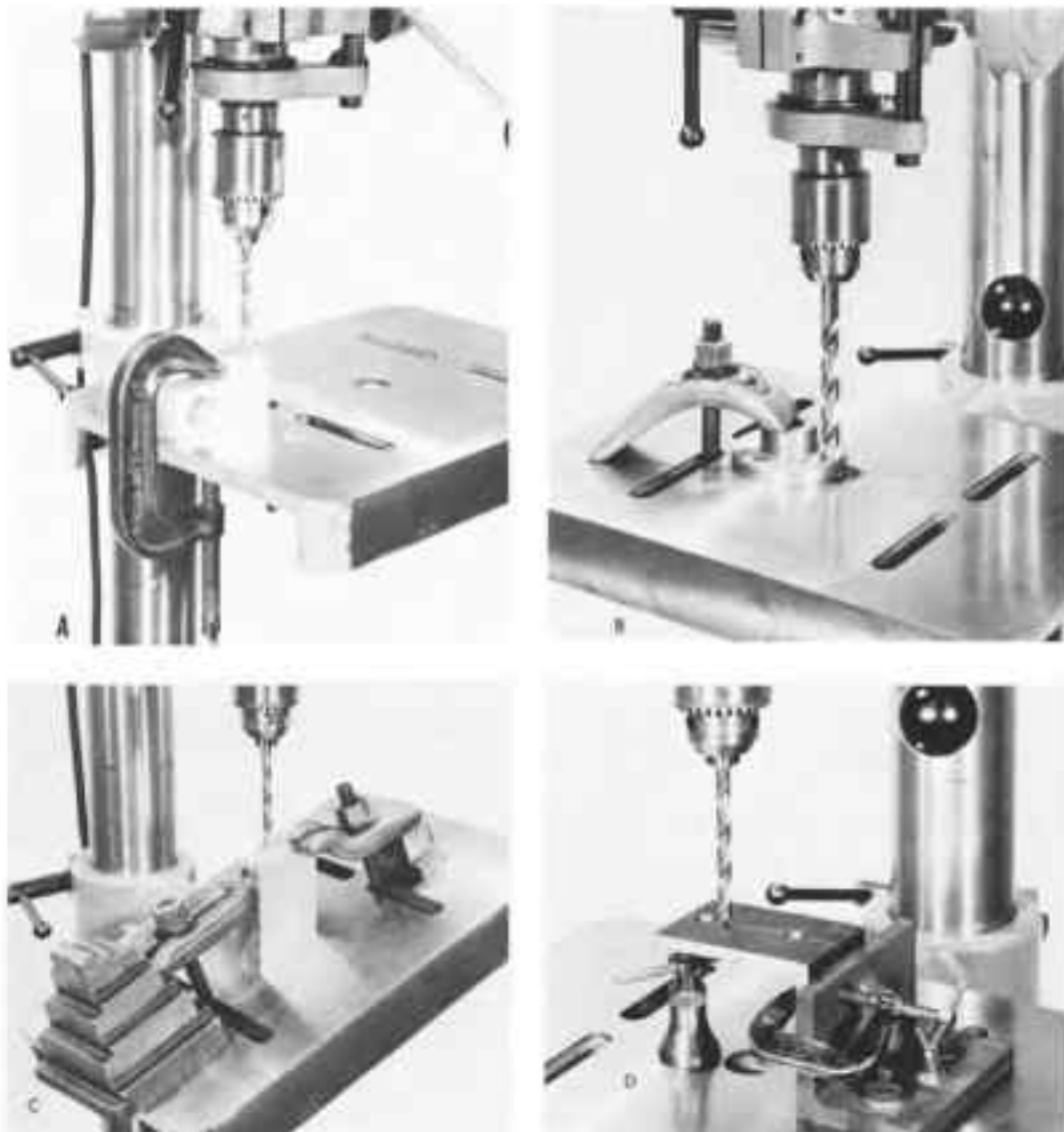


Fig. 5-2. Correct and incorrect applications of strap clamps.

securely and the accessibility of T-slots or other openings in the table in which the clamping bolts—called T-slot bolts—can be anchored. This may also determine where on the machine tool table the workpiece is to be positioned.

A typical setup for drilling is shown in Fig. 5-1. In this setup the workpiece is placed on parallels to provide clearance for the drill when it has passed through the hole. The drill press table must always be protected from damage by the drill. If the holes to be drilled are blind holes which do not pass through the workpiece, it would be better to clamp it directly to the table. The T-slot bolt is placed in a T-slot or other suitable opening in the drill press table and the strap clamp is placed over the bolt. The open end of the strap clamp is placed on the workpiece and the other end on heel blocks of suitable height so that the clamp is parallel to the

table. A washer and a nut are then placed on the bolt and tightened. In some instances, the T-slot bolt may be placed through an opening in the workpiece, as shown by the right-hand clamping arrangement in Fig. 5-1. In this case heel blocks may not be required. Finish machined surfaces on the workpiece must be protected from damage by placing soft metal shims between the strap clamp and the workpiece.



Courtesy of the Clausing Div. of the Atlas Press Co.

Fig. 5-3. A. Workpiece clamped to table with a C-clamp. B. Workpiece clamped with a universal adjustable type clamp. C. Drilling hole at an angle by tilting drill press table; workpiece clamped with strap clamps. D. Angle plate provides clamping or locating surface perpendicular to table.

Several correct and incorrect applications of strap clamps are illustrated in Fig. 5-2. As shown in view A, the T-slot bolt should be placed as close to the workpiece as possible. It should always be placed closer to the workpiece than to the heel blocks to assure that the greater proportion of the clamping force applied by the bolt and nut is applied to the workpiece and not to the heel block. Also shown in this view is the principle that the T-slot bolt should not be longer than necessary. As already mentioned and shown in view B, the height of the heel blocks should be such that the strap clamps are parallel with the machine tool table. Parallels, when used, should be placed directly below the strap clamps, as shown in view C, to prevent the clamping force from causing the workpiece to bend or spring out of shape. Otherwise, when the clamping force is released, the workpiece will spring back to its original shape causing the surfaces that were machined while the part was clamped to be inaccurate. Any overhang of the workpiece below a clamp must be supported for this reason. When the opening of the overhang is large, a screw jack may be used as a support against springing the part, as shown in view D. When the opening is narrow, shims and wedges are used, as shown in view E.

In most instances, when drilling with a medium size drill (approximately $\frac{3}{8}$ to $\frac{5}{8}$ in. or 9.5 to 16 mm), the workpiece should also be clamped to the table, as shown by the examples in Fig. 5-3. An ordinary C-clamp may be used, as in view A. A unique type strap clamp is shown in view B. This clamp is called a universal adjustable clamp and it does not require heel blocks. Conventional strap clamps are used in view C to hold the workpiece while drilling a hole at an angle by tilting the drill press table. The clamp at the left is a pin-type strap clamp; it has a pin at one end that is used to clamp inside a hole. Holes are sometimes drilled in a workpiece just for this purpose. The other clamp is an offset type which allows a shorter bolt to be used. Angle plates (view D, Fig. 5-3) provide a surface perpendicular to the machine tool table against which it is often convenient to clamp the workpiece. In view D the long overhang of the workpiece is supported against the thrust of the drill by a screw jack.

When the workpiece is long enough to provide sufficient leverage against the drill torque, medium size holes can be drilled by holding the workpiece against the table by hand, as in Fig. 5-4. In this case the drill torque should be resisted by a stop which is clamped to the table, as shown. There are advantages to not clamping the workpiece when drilling a hole. It is much faster and easier to locate the workpiece with respect to the drill when it is free to move about on the table. Then, when the drill is started, the workpiece will float and allow the drill to follow the center punch mark, thereby starting the hole in the desired location on the

workpiece. A technique that is sometimes used when the workpiece is clamped is to loosen the clamps slightly when starting the hole. Some clamping pressure should remain on the workpiece, which should be just free enough to allow the center punch mark to align the workpiece to the drill as it starts to drill the hole. Using a light manual or hand feed, the drill is allowed to drill to a depth of about one-half of the drill diameter. Then the drill is withdrawn and the work is clamped firmly to the table for drilling the remainder of the hole.

When using small drills (approximately $\frac{5}{16}$ in. or 8 mm and smaller), the workpiece does not usually need to be clamped, although it may have to be held in a drill press vise to provide the leverage required to safely overcome the drill torque. As an example, in view A, Fig. 5-5, the workpiece is large enough to be held against the table by hand when drilling the small hole. In view B a smaller workpiece is clamped in a vise which provides the necessary leverage to resist the torque of the small drill.

Drill press vises are a necessary accessory in drilling. They are used to hold workpieces when drilling large, medium, and small size holes. When drilling with large and medium size drills, the vise may be clamped to the table while holding the workpiece. A typical example is shown in Fig. 5-6A, where the vise is clamped to the table while a medium size hole is



Fig. 5-4. Using a bolt clamped to the table to resist the drill torque while holding the workpiece against the table by hand.

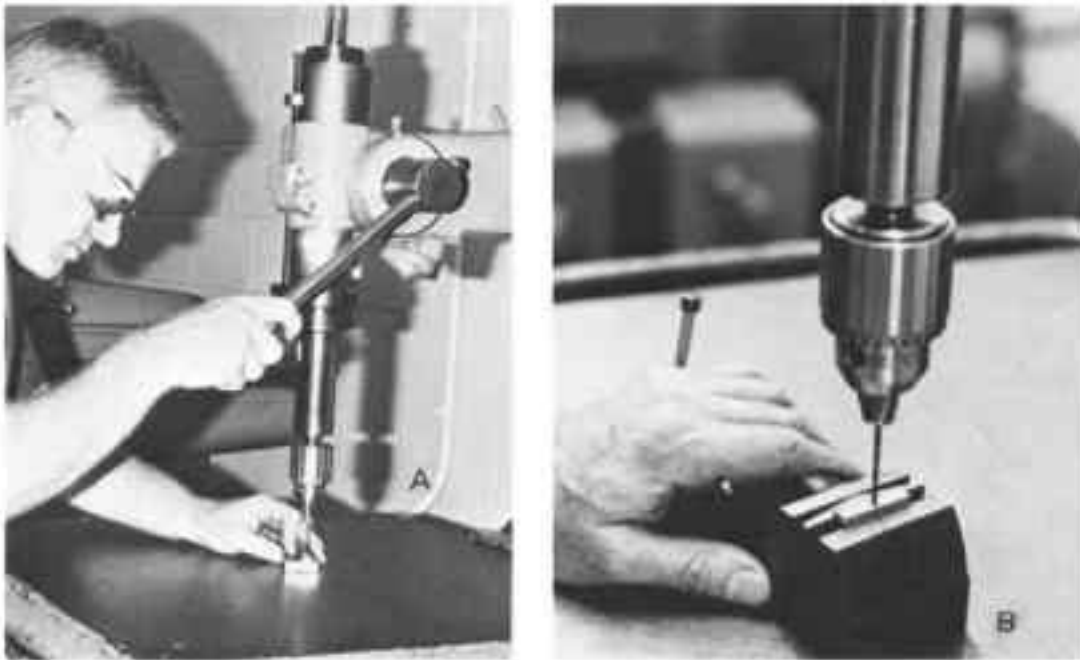


Fig. 5-5. Drilling with a small drill. A. Workpiece large enough to safely resist torque of small drill while held by hand. B. Smaller workpiece clamped in vise to resist torque of small drill.

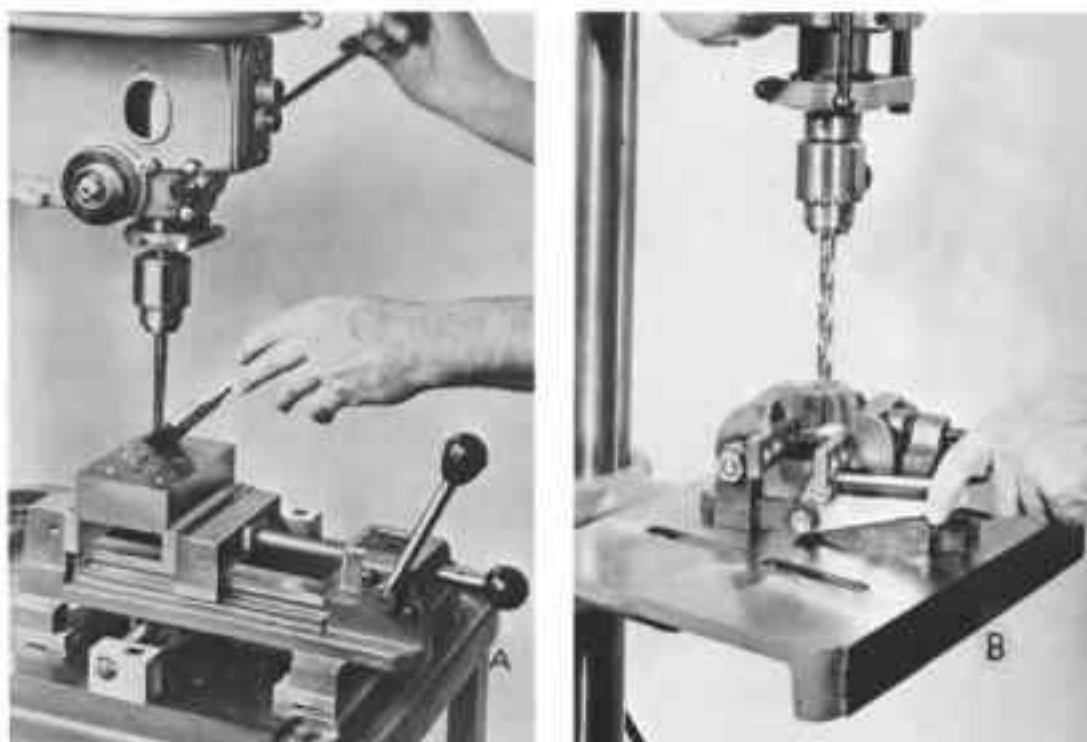
drilled. This vise has stepped jaws which enable the workpiece to be clamped above the seat of the vise to provide clearance for the drill as it passes through the hole; otherwise, it would be necessary to place the workpiece on parallels in the vise to provide the necessary clearance as shown in Fig. 5-6B. Some drill press vises have a V-groove machined on the solid jaw, which is used to locate and hold round workpieces, as shown in Fig. 5-7, A and B.

Occasionally, ingenuity must be used to set up the workpiece for drilling. As an example, in Fig. 5-8 an ornamental iron part is held on a wood board to drill the hole. The wood "fixture" is rounded on the end to conform to the shape of the workpiece.

The spindle of the radial drill press can be positioned in areas on the workpiece which are sometimes difficult, or impossible, to reach with an upright drill press. Such a setup is shown at A, Fig. 5-9, which illustrates the punch plate of a die block being drilled. In this setup the punch plate is placed on parallels that are positioned on the drill press table. The parallels will provide clearance for the drill when it penetrates the punch plate and prevent it from damaging the table top. It is not necessary to clamp the punch plate to the table as its weight is sufficient to resist the drill torque and prevent it from moving as the hole is being drilled. Smaller workpieces, as in B, Fig. 5-9, must be clamped to the table to prevent their movement when the holes are drilled. The hole being drilled is not a through hole; thus, it is not necessary to use parallels as at A.

Note the use of strap clamps in this setup. At A, Fig. 5-10, a round bar of steel is bolted against a V-notch (not visible) machined into the side of the table. The V-notch aligns the bar in a vertical position for drilling holes in the end, as shown. The table of the radial drill press may be set at an angle to drill an angular hole, shown at B, Fig. 5-10. The part is clamped with "homemade" strap clamps which are thick steel straps through which a hole has been drilled.

A job that is frequently encountered is that of drilling a hole through, and perpendicular to, the axis of a cylindrical part, as shown in Fig. 5-11. The workpiece is first laid out as described in Chapter 3. (See E, Fig. 3-4.) The part is placed in a V-block, or, if it is a long part, in a matched pair of V-blocks. It is positioned so that the drill will not cut into the V-block. Some V-blocks have a hole through the center for drill clearance. When two V-blocks are used the drill can be positioned between them (Fig. 5-12). The scribed layout line on the end face of the workpiece is aligned in a vertical position by the blade or the beam of a combination square (A and B, Fig. 5-11). The work is then clamped into position, as shown at C. The square is not used to align the drill with the workpiece. Very large cylindrical parts can sometimes be aligned in the manner shown at D. The blade of a combination set has a center head and a



A. Courtesy of Lazy Tools Inc.;

B. Courtesy of Clausing Div. of the Atlas Press Co.

Fig. 5-6. A. Drilling a medium-size hole with the workpiece clamped in vise and resting on stepped jaws of vise which is clamped to table by patented "Rapid Lox" base.

B. Workpiece clamped in vise while resting on parallels.

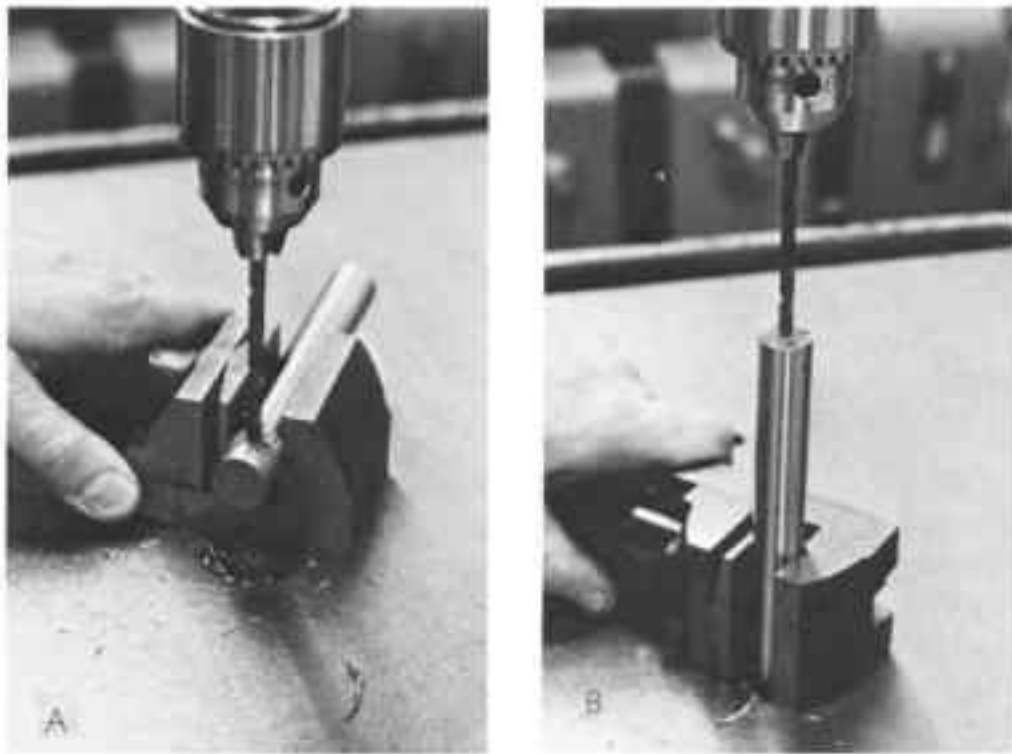


Fig. 5-7. V-groove in solid jaw of vise for locating and holding round workpieces. A. Workpiece held for drilling perpendicular to axis. B. Workpiece held for drilling parallel to or along axis.

square head fastened to it, as shown. The blade is positioned over the punch mark at the center of the hole. Sometimes, greater accuracy can be obtained by placing the blade opposite the layout line as shown at D. The workpiece is adjusted until the spirit level on the square head is exactly level. The part is then clamped into position.

Drilling the Hole

The operation of drilling the hole can start when all of the preparations have been completed; i.e., the work is set up, a sharp drill of the correct size is held in the spindle of the machine, and the machine has been set up for the correct cutting speed and feed. The first step is to align the drill with respect to the center punch mark at the center of the hole to be drilled. The drill point is then fed into the workpiece until it has penetrated approximately two-thirds of the drill's diameter. The drill is then backed away and the relation of the drilled spot to the scribed layout circle should be observed. At A, Fig. 5-13, the spot produced by the drill point is not concentric with the layout circle, indicating that if the hole were to continue to be drilled in this position it would not be in the desired location. To correct this situation, cut one or more shallow grooves on the heavy side, as shown at B, Fig. 5-13, with a gouge, or draw chisel (Fig. 5-14). The groove unbalances the cutting forces acting against

the lip of the drill causing it to shift in the direction of least resistance, which is toward the grooved side. The drill point is fed a little deeper into the work and the concentricity between the spot and the layout circle is again checked. If necessary, additional grooves are cut with a draw chisel. When the spot and the layout circle are concentric, as shown at C, Fig. 5-14, the hole can be drilled to the required depth. When the hole has been drilled, about one-half of each witness mark will remain to show that the hole is drilled exactly where the layout has indicated that it be made. It should be mentioned that it is usually not necessary to cut grooves in every hole as described. Frequently, the initial check of concentricity between the spot and the layout circle will show that they are concentric and the hole can be drilled to depth without additional positioning of the work.

An alternate method of starting a drilled hole, preferred by some, is to center drill a hole first (see Fig. 5-15). The center-drilled hole is then used as a guide for starting the twist drill. The center drill is short, stiff, and will not bend. Furthermore, it has a relatively small chisel edge when compared to larger size drills, thereby enabling the center drill to pene-



Courtesy of the Clamping Div. of the Atlas Press Co.

Fig. 5-8. Drilling an ornamental iron part using a wood "fixture," which is clamped to the table.

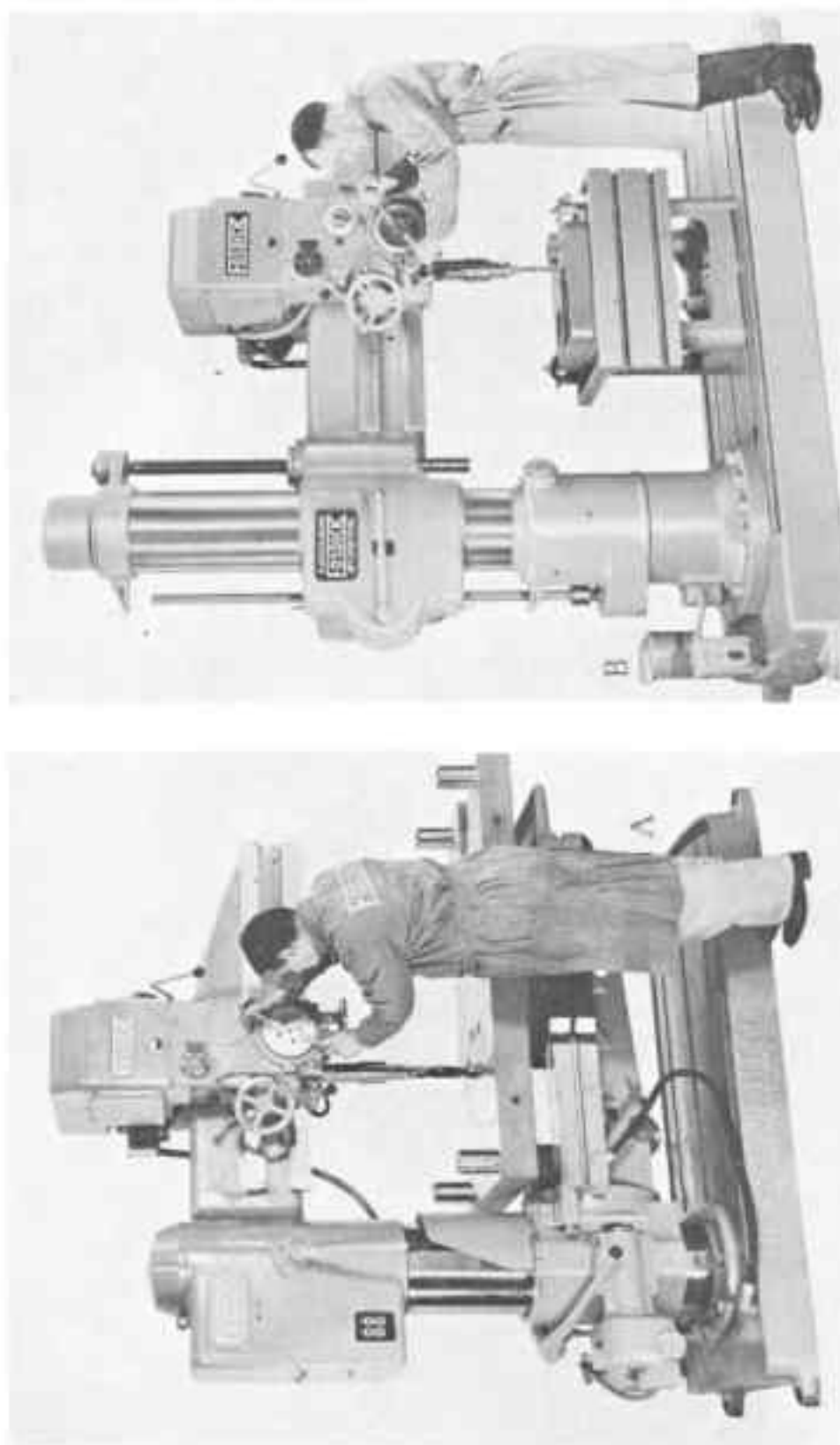


Fig. 5-9. Typical radial drilling machine setups. A. Weight of the die shoe is sufficient to resist drill torque. Work placed on parallel to perfect table from drill. B. Work clamped to table top when not drilling a through hole.

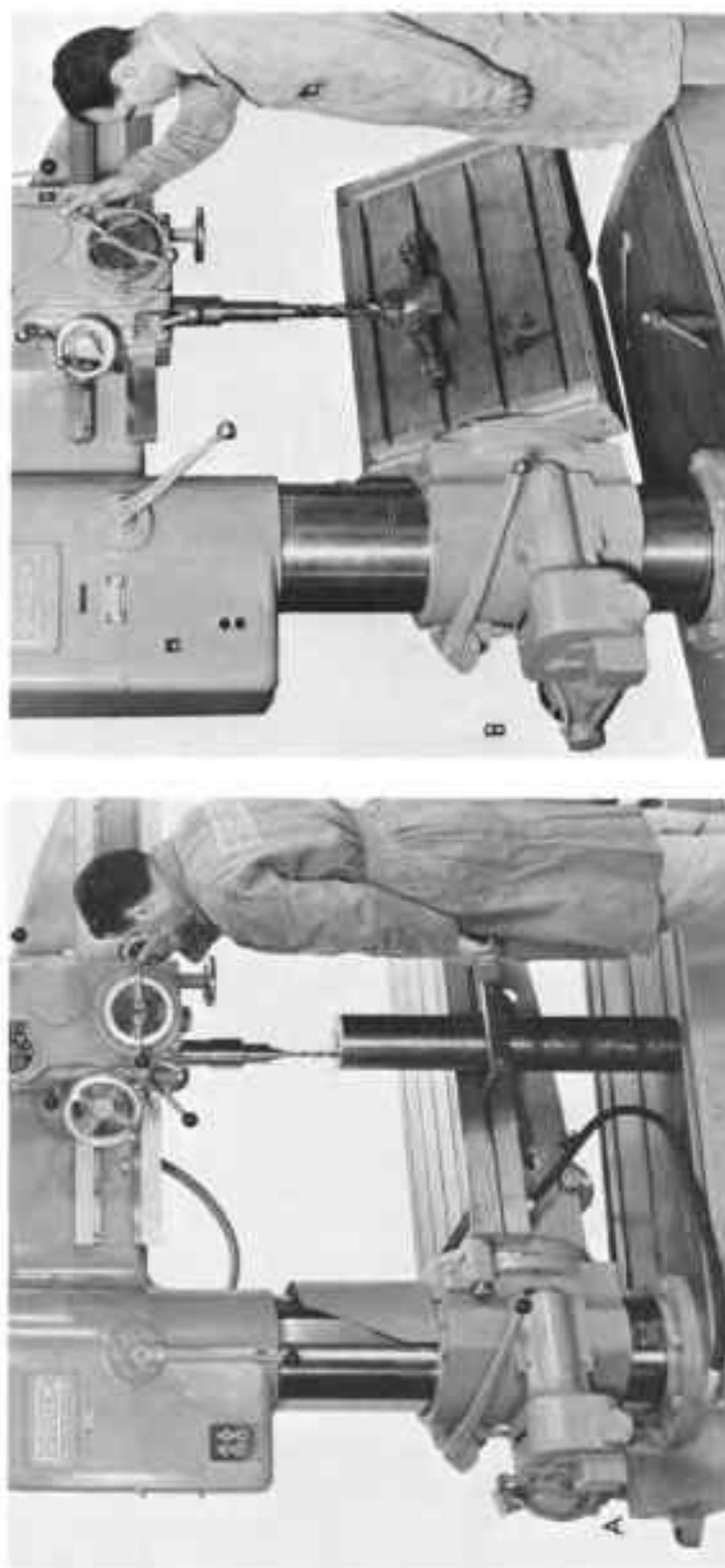
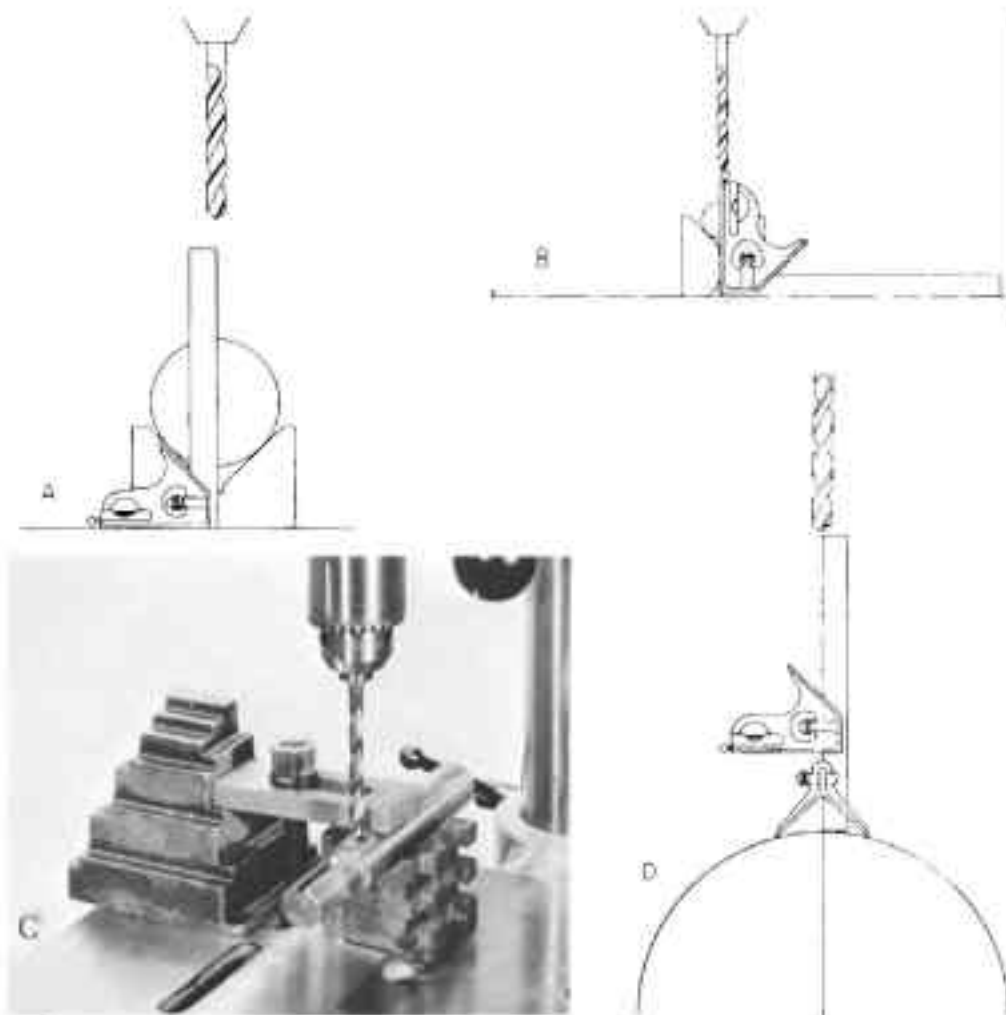


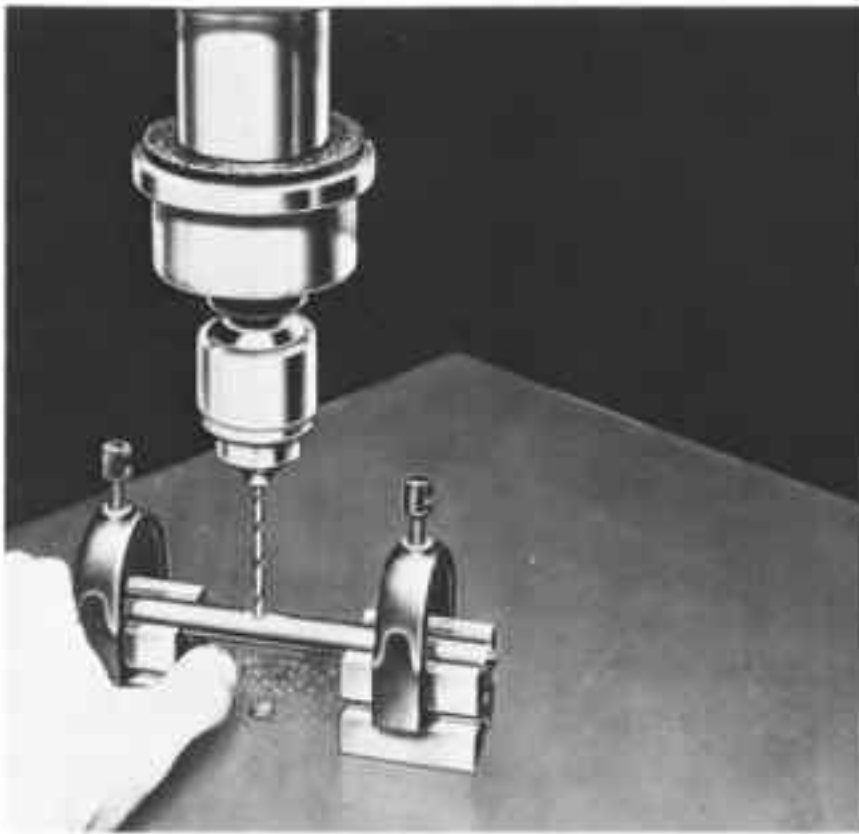
Fig. 5-10. Radial drilling machine setups. A. Cylindrical workpiece clamped in V-notch on side of table for drilling in end face. B. Table tilted at an angle for drilling.

trate the workpiece with greater ease. The coned surface formed by the center drill has an included angle of 60° which is steeper than the included angle of the drill point. Thus, the lips of the twist drill contact the workpiece first and the chisel edge does no work until the location of the hole is established. The point angle of the lips will cause the drill to follow the axis of the hole made by the center drill, and, since the chisel edge of the twist drill does not engage the work, the drill will penetrate into the work with ease. This method is especially recommended for starting holes on machines where the work is normally clamped to the table and where the spindle can be aligned exactly to the desired location on the workpiece. When this method is used on an upright or sensitive drilling machine, the workpiece should not be clamped when starting the hole with the center



Lower left view courtesy of the Clausing Division of the Atlas Press Company

Fig. 5-11. (Upper left) Using the blade of a combination square to align cylindrical part. (Upper right) Using the beam of a combination square to align a cylindrical part. (Lower left) Setup for clamping cylindrical part in V-block. (Lower right) Using a center head combined with a square head to align a large cylindrical part.



Courtesy of The L. S. Starrett Company

Fig. 5-12. Drilling hole perpendicular to axis of cylindrical part which is clamped in matched pair of V-blocks.

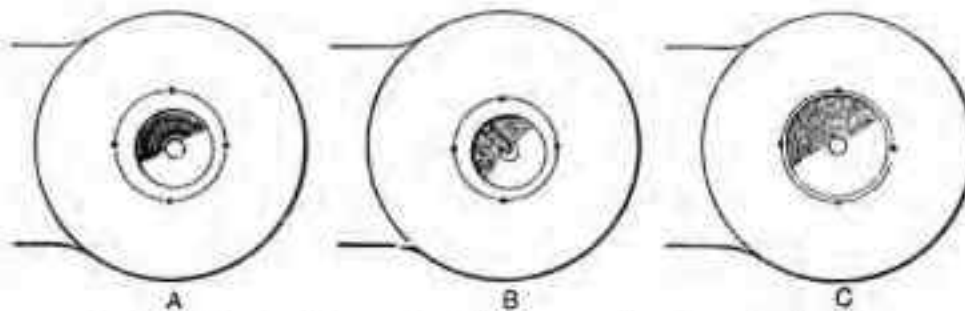


Fig. 5-13. Method of starting drill, concentric with scribed circle.

drill. On radial drilling machines the arm and the spindle head should not be locked. Allowing the workpiece or spindle to "float" lets the center drill find and align itself with the center punch mark as it starts drilling into the workpiece. After the workpiece and the machine spindle are aligned, and when the center hole has been drilled, the workpiece may be clamped, when necessary, to drill the hole to size. On radial drilling machines, the arm and the drill head are clamped.

On upright and sensitive drill presses, when the workpiece is too heavy to "float" on the table and align the center punch mark to the center drill, this method is not accurate and the previous method is recommended since any misalignment in starting the drill can be corrected with a draw chisel. In center drilling it is very difficult to correct any misalignment once the hole has been drilled. Another disadvantage is that it requires frequent tool changes.

The average thrust force required to feed a large-size drill is very high. A large proportion of this thrust is caused by the action of the chisel edge. The thrust required can be greatly reduced and the drill made to cut faster, by first using a smaller drill to make a "lead hole," as shown in Fig. 5-16.

Hole Location Transfer

In machine repair and toolroom work the location of a hole must frequently be transferred from one part to another. There are three methods that can be used: 1. spotting with a drill; 2. using transfer punches; and, 3. using transfer screws.

A typical example of the first method is shown in Fig. 5-17, where a .500 inch dowel pin hole in the hardened die must be transferred to the die shoe, which is not hardened. The illustrations below the assembled view show the sequence of operations. First the die must be very accurately located on the die shoe and then they are firmly clamped together by socket head screws. Using a $\frac{1}{2}$ inch twist drill, the die shoe is "spotted," as shown at B. The diameter of the spot should be less than

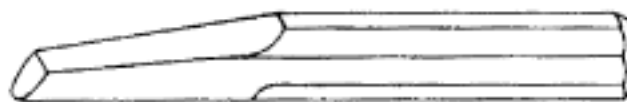


Fig. 5-14. Draw chisel or gouge.

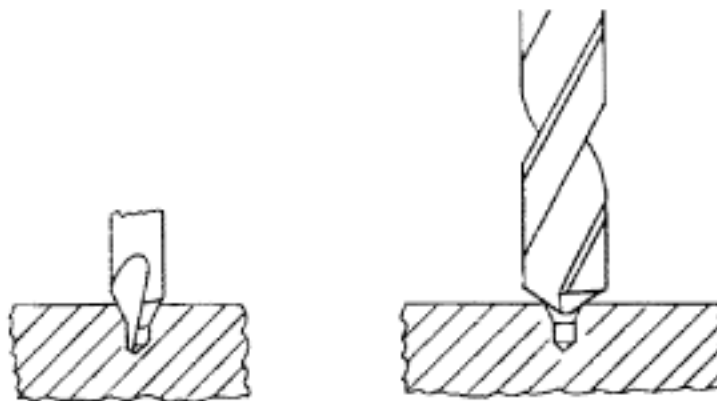


Fig. 5-15. Starting a drilled hole with a center drill.

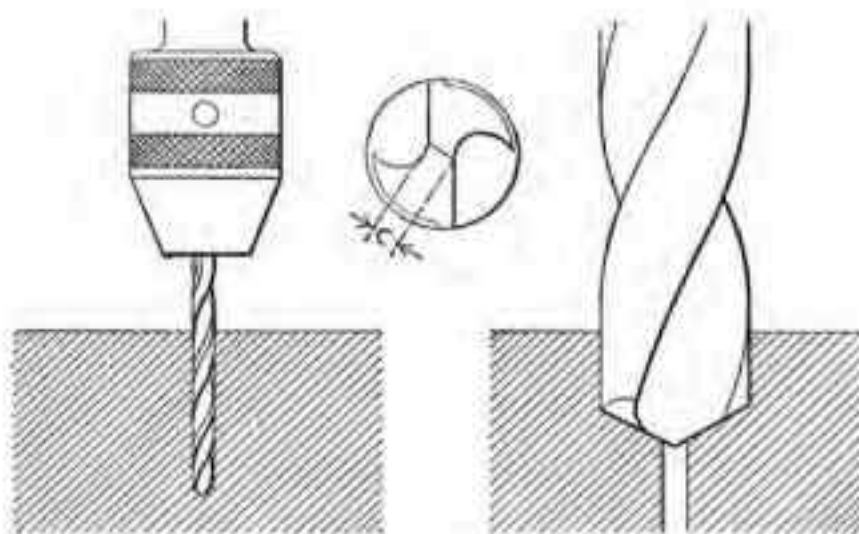


Fig. 5-16. A small lead hole is sometimes used to reduce the thrust required for a large drill.

the diameter of the drill—i.e., the spot should not be quite as deep as the point of the drill. Next the hole is drilled through the die shoe with a $\frac{3}{16}$ inch drill (see view C) and reamed with a $\frac{1}{2}$ inch hand reamer, as shown at D, to complete the job.

On other classes of work that do not require the same degree of accuracy the sequence may be changed slightly. For example, the hole may be drilled to size at B, omitting steps C and D. For tap drill holes, the parts may be disassembled at C before drilling the hole to the tap drill size, or the parts may be disassembled before drilling through if they cannot be clamped firmly together. If both parts are not hardened, a very precise alignment of the holes can be obtained by drilling both holes undersize and then reaming the holes together with a hand reamer or by machine reaming.

Transfer punches are cylindrical in shape to fit snugly into the hole in which they are to be used. One end is hardened and has a small sharp punch point that is ground concentric with the body. The other end must be soft—this is very important in order to prevent possible serious injury when this end is struck with a hammer. Transfer punches can be purchased commercially, or they can be made on the job from drill rod or other types of tool steel. The use of transfer punches is illustrated in Fig. 5-18. After the two parts are located and clamped together, the transfer punch is inserted in the hole and tapped lightly with a hammer. The parts are then disassembled and the transfer punch mark is very carefully enlarged slightly with a regular center punch. A start hole is then drilled into the workpiece using a $\frac{1}{32}$ inch or a 0.75 mm drill. Successively larger holes are then drilled into the workpiece to enlarge it to the required size and depth. This procedure will give the most precise results.

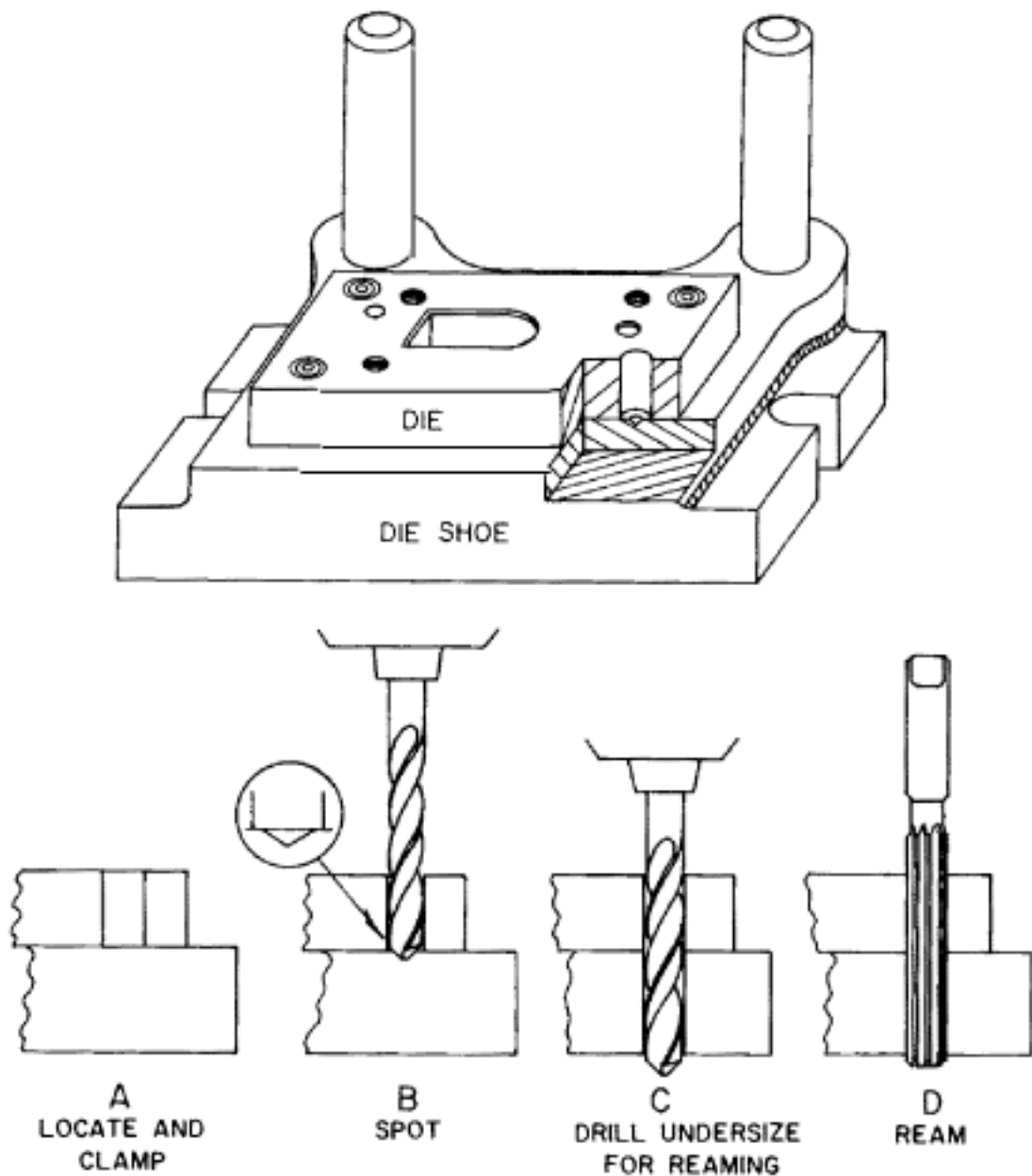


Fig. 5-17. Procedure for spotting a hole in a die shoe.

If less accuracy is satisfactory, a circle can be scribed with dividers, witness marks punched on the circle, and the hole drilled in the usual manner.

Transfer screws also have a sharp punch point on one end but their body is a screw thread designed to fit in a hole having a mating thread. They are used in a similar manner as transfer punches, except that it is usually necessary to tap one of the parts with a soft hammer instead of the transfer screw to form the punch mark.

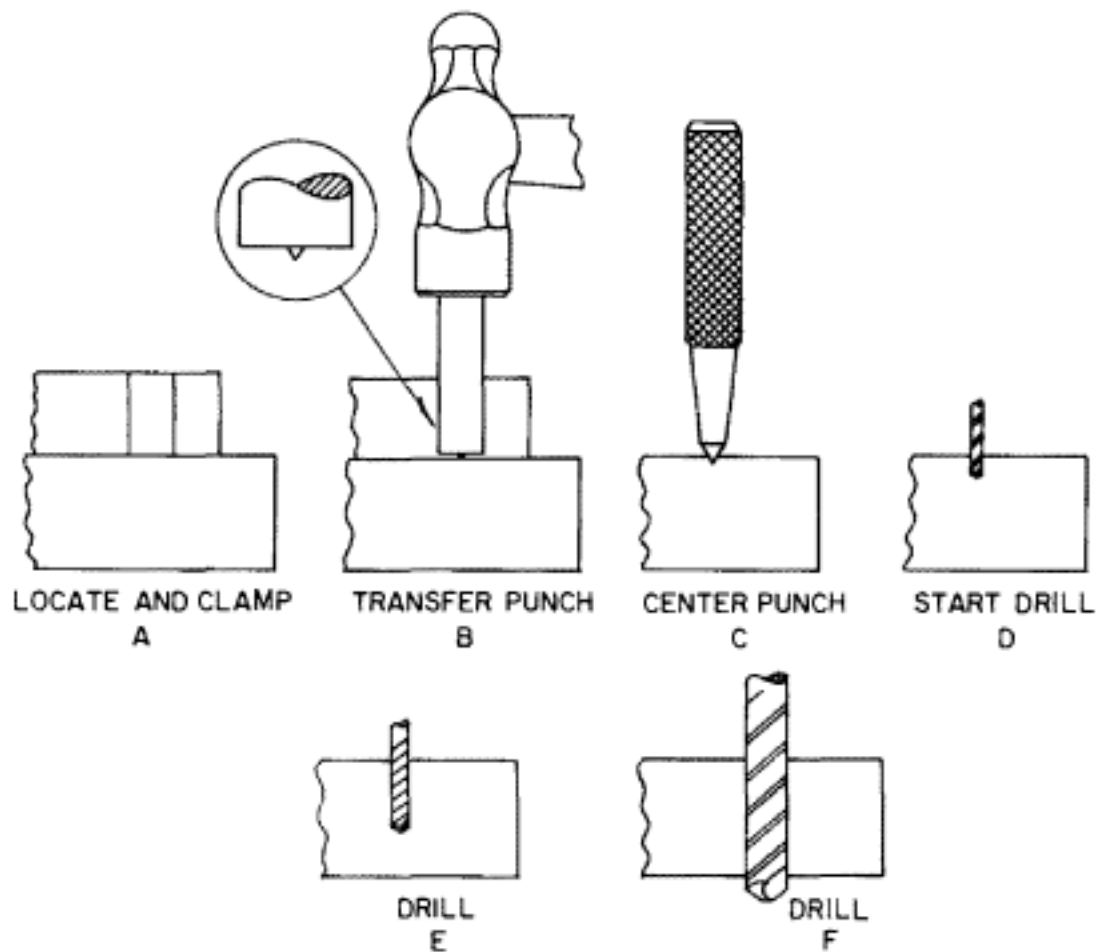


Fig. 5-18. Procedure for using a transfer punch.

Boring

In addition to the usual operations (drilling, reaming, counterboring etc.), drilling machines equipped with a power feed mechanism are sometimes used to perform boring operations. A single point tool is used in boring to enlarge an existing hole. Figure 15-3, in Chapter 15, illustrates a boring operation such as might be performed on a drilling machine, as well as on a jig boring machine. The single point boring tool is held in a boring bar that can be offset radially from its center of rotation by an offset boring head. This radial adjustment of the cutting tool enables it to be set to bore precise hole diameters.

Reaming

Reamers are used to finish drilled holes to size and to a smooth, surface finish. In order to produce accurate holes by reaming, the reamer must be sharp and correctly ground, the right amount of stock must be left in

the hole for reaming, and the reamer must be accurately aligned with the axis of the drilled hole. The feed of the reamer through the hole must not be too fast (.0015 to .004 inch per land), otherwise the accuracy and the surface finish of the hole will be impaired. Sometimes a reamer will chatter. This chatter may exist only at the start of the hole as the reamer aligns itself with the drilled hole. Chatter is objectionable because it impairs the surface finish of the hole and it can damage the surface finish of the reamer. Cemented-carbide tipped reamers, in particular, are quite likely to chip if chatter occurs even at the start of the hole.

To eliminate chatter the following steps should be tried: reduce the cutting speed; vary the feed; reduce the relief angle on the reamer; use a piloted reamer; use a reamer having helical flutes; increase the rigidity of the setup; and, as a last resort, use a reamer having irregular spacing of the flutes. The use of a helical fluted reamer is much preferred to one having irregular spacing. The helix angle of helical fluted reamers should not be greater than necessary, as this will tend to increase the force required to feed the reamer through the hole. One exception to this are taper reamers; these will cut better with a very steep spiral. Often, the difficulties associated with chatter can be eliminated by exercising great care in aligning the reamer with the hole, at the start of the operation. This should never be done carelessly.

A good supply of the correct cutting fluid should be used for the reaming operation except when reaming cast iron, which should be reamed dry, with only a jet of compressed air to act as a cooling medium. Use of the correct cutting fluids often materially improves the surface finish obtained in a reamed hole. Generally, for reaming steel, a sulfurized mineral oil will work best although good results have also been obtained using soluble oil. Soluble oil and lard oil compounds can be used on aluminum. Brass may be reamed dry, although the application of a soluble oil or a lard base, oil compound is often advisable. Soluble oil, or a sulfurized mineral oil, is recommended for stainless steels. Many proprietary cutting fluids can be purchased which will do an excellent job.

Tapping

When tapping is to be done in a drilling machine there must be provision for rapidly reversing the spindle so that the tap can be backed out of the hole after the thread is cut to the required depth. Most radial drilling machines are designed with a spindle reverse mechanism that is actuated by the spindle control lever. Upright and bench drilling machines are usually not equipped with such a mechanism for reversing the rotation of the spindle thus a special attachment for this operation is required. An automatic reverse tapping attachment is shown in Fig. 5-19. This attachment has a taper shank which is inserted in the spindle of the machine. The body of the chuck and the gage *B*, which may be used to control the depth to which holes are tapped, are both prevented from rotating with the machine spindle by rod *A*. When tapping a hole the spindle is lowered by hand and the tap is fed to the required depth manually. When the lower end of the stop rod *B*, comes into contact with the face of the

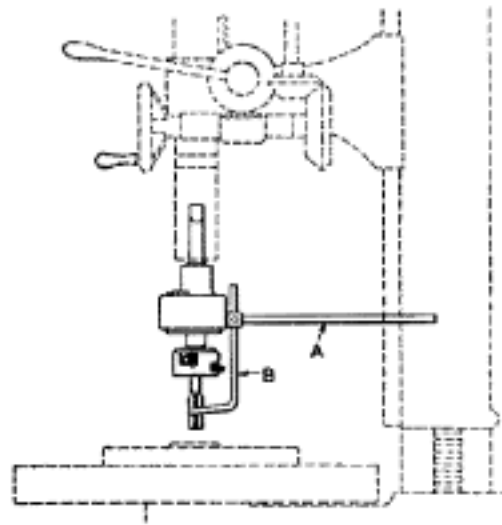


Fig. 5-19. Automatic tapping attachment applied to upright drilling machine.

work, the forward rotation of the tap is stopped. When the drill press spindle is raised, gears within the chuck body are engaged that cause the tap to back out of the hole rapidly. The tapping attachment may be equipped to have an adjustable friction drive that stops the rotation of the tap whenever the tapping torque exceeds a preset limit, thus safeguarding the taps against breakage.

There are many more variables that must be considered when selecting the cutting speed for tapping than for other machining operations; thus, it is not surprising that the best and most efficient cutting speeds for tapping cannot be tabulated with any assurance that the tap will then be operating at its highest efficiency. Among the factors that must be considered are:

1. Material to be tapped
2. Heat treatment and hardness of material to be tapped
3. Length of hole
4. Length of chamfer on top
5. Pitch of thread
6. Percentage of full thread to be cut
7. Cutting fluid to be used, and
8. Design of the drilling machine with respect to the sensitivity of the controls.

For example, speeds must be lowered as the length of the hole is increased because long holes accumulate chips, increase friction, and interfere with the flow of cutting fluids. Taps having long chamfers can be run faster than taps with short chamfers in short holes; however, in long holes the taps with the short chamfers can be run faster. Bottoming taps must be run slower than plug taps. A slower speed is required to tap full-depth thread than a 75-per cent-depth thread. When the tap diameter exceeds approximately $\frac{1}{2}$ inch, coarse thread taps should be run more

slowly than fine thread taps having the same diameter. Taper threaded taps, such as pipe taps, should be operated from one-half to three-quarters of the speed of a straight tap of comparable diameter. Automatic machines in which the movements required for tapping are machine controlled, can, in most cases, be operated at greater tapping speeds than manually operated machines. The machine operator must exercise good judgment in deciding the limits of his ability to control the machine in order to prevent tap breakage and other possible damage. A table of recommended tapping speeds is given in the appendix, but these may have to be modified by the factors discussed above.

When tapping, a cutting fluid should always be used with all materials except cast iron and plastics. Cast iron and plastics are machine tapped dry or with a jet of compressed air to cool the tap and help to remove the accumulation of chips. It is not practical to give specific recommendations for tapping other materials because of the wide variety of proprietary cutting fluids that can do an excellent job. The following recommendations are general and it should be understood that there are exceptions. Sulfur based oils with active sulfur are recommended for tapping plain carbon steels, alloy steels, malleable iron, and Monel metal. Tool steels, high speed steels, stainless steels, and alloy steels that have been heat treated to a higher hardness should be tapped using a chlorinated sulfur base oil. Mineral oil with a lard base is recommended for aluminum, brass, copper, manganese bronze, naval brass, phosphor bronze, and Tobin bronze. Aluminum and zinc die castings should be tapped with lard oil that has been diluted with 40 to 50 per cent kerosene. The proper cutting fluid used when tapping can increase tap life, improve surface finish, reduce friction, and result in better size control.

Drill Jigs and Fixtures

Drill jigs and fixtures are frequently used when a number of duplicate parts have to be drilled. The shop terms are understood to have the following meaning:

Drill fixtures are specially designed tools that are used to hold the workpiece in a fixed location. They may incorporate a method or locating device to establish a known relationship between locating, or reference, surfaces on the part and the spindle of the machine. Drill fixtures have come into prominence with the advent of numerically controlled drilling machines.

Drill jigs incorporate all of the features of drill fixtures and, in addition, they are constructed with guide bushings that guide the twist drill or the reamer in the required relationship with respect to the workpiece. Drill jigs are used on all types of drilling machines except numerically controlled machines.

Drill jigs provide for a fast and accurate method of locating the drill in the position on the workpiece where the hole is wanted, without requiring a layout to be made, and without the need to use the draw chisel or a center drill to start the hole. There are many different types of drill jigs, some of which are very simple, while others are quite complex. The basic functions of a drill jig are to locate the workpiece in the position



Courtesy of the Clamping Division of the Atlas Press Company

Fig. 5-20. Casting held in a drill jig for drilling operation. Note that the part is held to the table manually.

required for drilling, to clamp it in this position, and to guide the drill by means of hardened steel bushings. An example of the use of a drill jig is shown in Fig. 5-20. The part is located and clamped in the drill jig which is held to the table, manually. The jig can be positioned under the drill quickly and easily, for drilling several holes. Another drill jig is shown in Fig. 5-21. The workpiece is shown at A, and the jig is used to drill the $\frac{1}{4}$ -inch hole. The work is shown in position in the jig by the dashed and dotted line. Figure 5-22 illustrates the application of hardened-steel drill jig bushings. These bushings fit the drill closely and keep

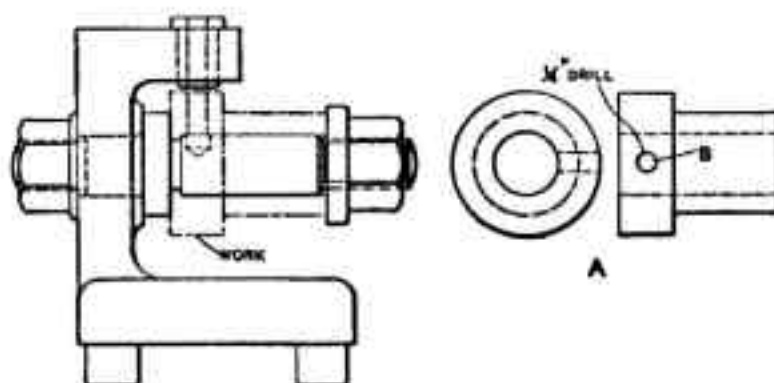


Fig. 5-21. Drill jig for drilling bushing.

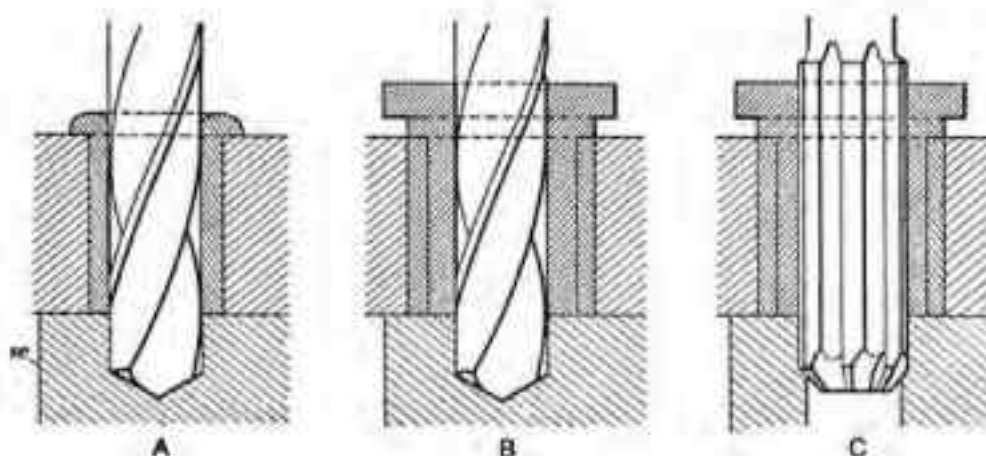


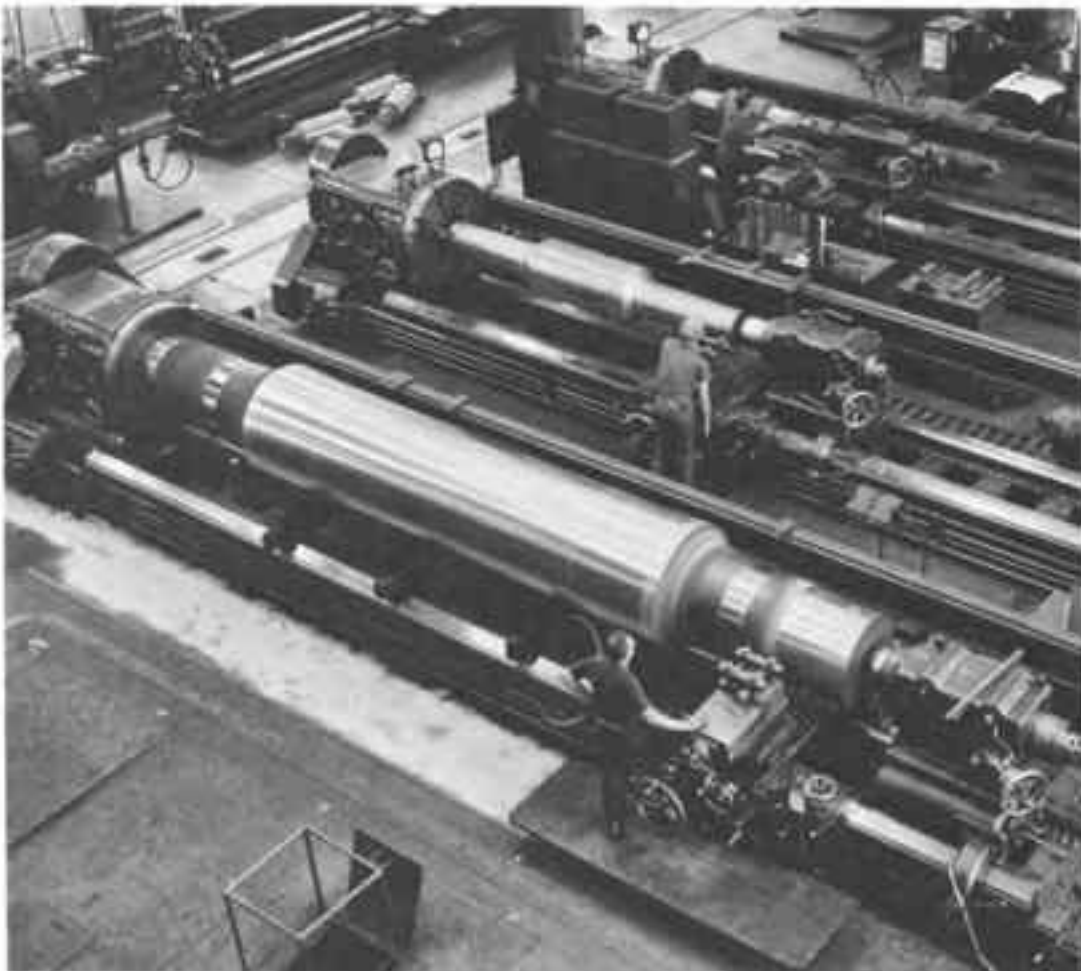
Fig. 5-22. Fixed and removable guide bushings for drill jig.

it in proper position. Some jigs have fixed guide bushings and others have removable guide bushings. A fixed bushing is shown by the sectional view at A, Fig. 5-22, which also indicates how the drill is guided while it is drilling the work, *w*. Jigs are equipped with removable bushings should drills of a different size need to be used or if drilled holes are to be finished by reaming. For example, if a hole is to be drilled and reamed, a removable bushing is used that fits the drill, as shown at B, and this is replaced by a bushing that fits the reamer, as shown at C.

Drill fixtures are similar to the drill jigs shown except that they do not have drill bushings. For this reason they are generally more open in construction than drill jigs. They are always fastened to the table of the machine so that their relationship to the spindle remains fixed. Drill jigs, on the other hand, may or may not be fastened to the table.

Engine Lathe Construction

The origin of the term "engine lathe" undoubtedly can be traced to the manner in which early metal-cutting lathes were powered. These early lathes were driven by a reciprocating steam engine through a system of pulleys, belts, and clutches. Usually, a single steam engine provided the power for several lathes as well as for the other machine tools in the machine shop. Later, a single electric motor replaced the steam engine



Courtesy of The R. K. LeBlond Machine Tool Company

Fig. 6-1. Large lathes at work.



Courtesy of the Atlas Press Company

Fig. 6-2. Modern 12" bench lathe.

to drive a group of machine tools. All modern engine lathes, as well as other machine tools, are driven by an individual electric motor. The individual motor drive eliminates the need for overhead shafting, reduces power losses, provides a more direct drive, and provides greater flexibility in the placement of the lathe on the machine shop floor.

Engine lathes differ from other lathes by their greater simplicity of construction and by their greater versatility in performing many different machining operations. Turning cylindrical surfaces, facing flat surfaces, cutting threads, drilling and boring holes, and cutting internal threads are some of the typical operations that are performed on engine lathes. Of equal importance in considering the versatility of the engine lathe is the ease with which the workpiece can be set up and held, and the ease with which the lathe can be manipulated in performing different operations. Although production lathes and numerically controlled lathes are used, to a large extent, where a variety of parts must be made, the engine lathe is still the basic machine tool in the machine shop. It is used extensively in jobbing shops and in tool and die shops. When operated with skill and with an understanding of its capabilities, it can produce parts to exacting standards of accuracy and finish. When the appropriate size of machine is used, the engine lathe can produce both large and small parts. A group of large engine lathes at work is shown in Fig. 6-1. As a contrast in size, a small bench lathe is shown in Fig. 6-2. A small high-precision toolroom lathe is shown in Fig. 6-3. The lathe shown in Fig. 6-4 is a sliding bed gap lathe which is also called a gap bed lathe. The bed of this lathe can be moved over the base to form an opening, as shown, which provides extra swing to allow the lathe to handle larger and odd shaped parts. Sliding the bed open also provides a greater bed length to allow it to handle longer parts. When the bed is closed, it functions as a



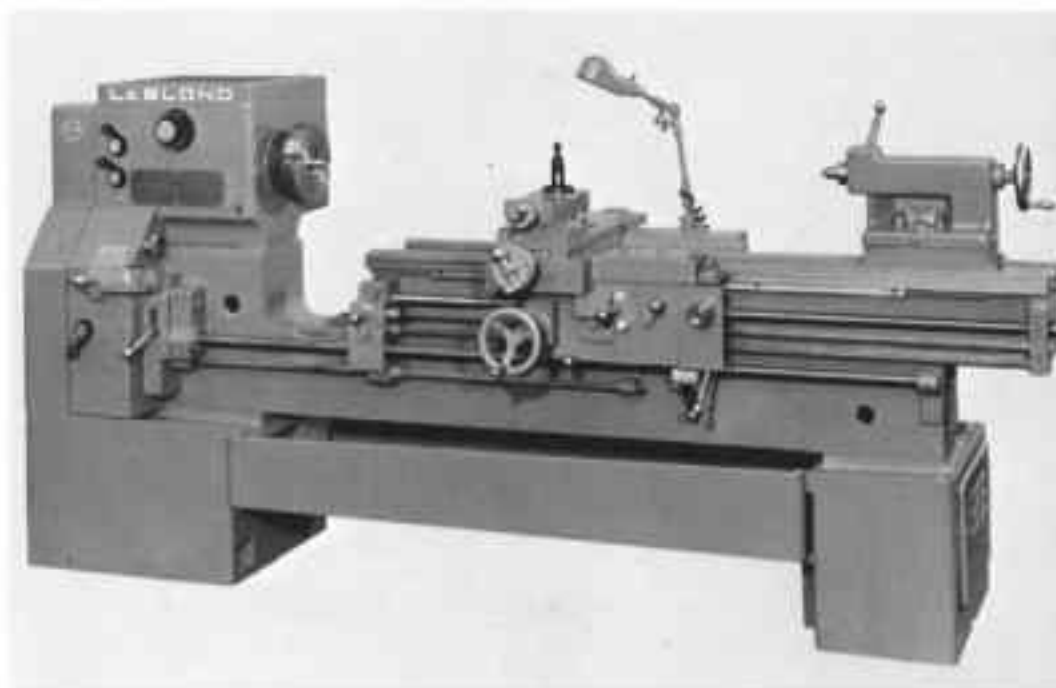
Courtesy of Hardinge Brothers Inc.

Fig. 6-3. Precision toolroom lathe.

regular engine lathe. A hollow spindle lathe is shown in Fig. 6-5. Parts can be turned on this lathe that are considerably longer than the normal center distance by positioning the workpiece through the hollow spindle. The workpiece can be held by chucks on both ends of the spindle. Hollow spindle lathes are used extensively in oil fields for turning and threading the ends of oil pipes.

Principal Features

The principal parts of an engine lathe are shown in Fig. 6-6. The bed of the lathe, Fig. 6-7, is a heavy rigid casting made from gray cast iron. Gray cast-iron is used as a material of construction for machine tool beds and frames because it has excellent vibration dampening qualities and it is a good bearing material for sliding bearings. The bed rests on two legs with the exception of small bench lathes which have the bed



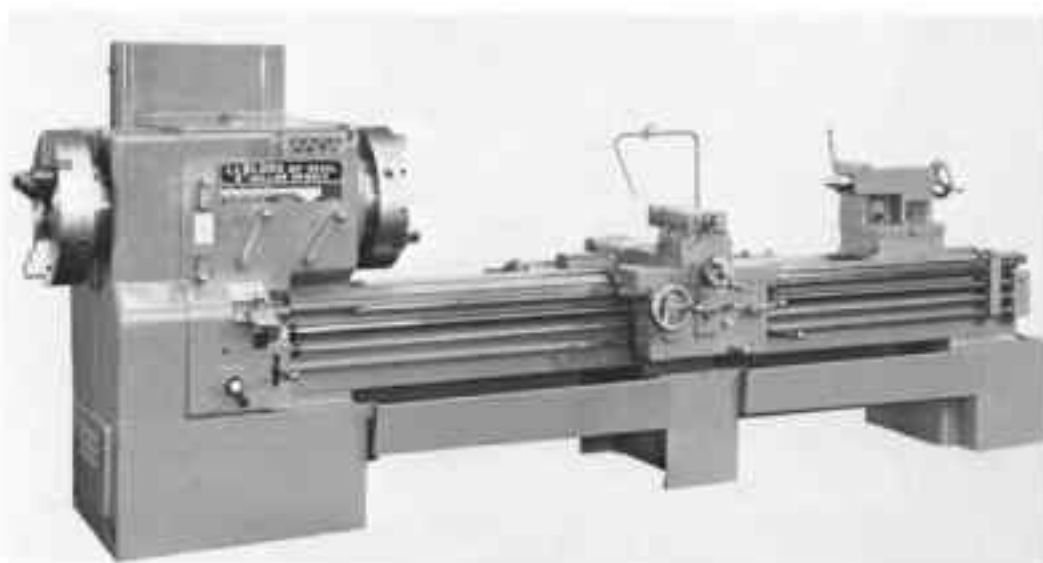
Courtesy of LeBlond Machine Tool Co.

Fig. 6-4. Sliding bedgap lathe.

attached directly to the bench. The top surfaces of the bed are accurately machined to provide a sliding surface for the carriage and to provide for the precise alignment of the headstock and the tailstock. This alignment is achieved by providing "ways" which are raised surfaces machined on the top of the bed. The ways are machined parallel to each other. One of the two ways is used to align the headstock and the tailstock. The other way is used to guide the movement of the carriage as it slides over the bed causing it to be parallel to the axis of the lathe. The direction of this carriage movement is called longitudinal. Some lathes, that shown in Fig. 6-2, for example, have flat beds that do not have ways. The surfaces on the sides of the bed that are perpendicular and adjacent to the top of the bed are carefully machined parallel to each other. These surfaces perform the function of the ways on flat bed lathes.

Methods of Drive

The lathe is driven by an electric motor that is usually located inside the cabinet leg at the headstock end. A multiple V-belt drive transmits the power from the motor to the headstock. The principal function of the headstock is to support and to align the headstock spindle. On many lathes it also contains a selective gear transmission that provides for different speeds of the lathe spindle. A typical geared headstock is shown in Fig. 6-8. The spindle speeds on geared head lathes are changed by placing the spindle speed change levers, located on the side of the headstock, in the position prescribed by the instruction plate located behind



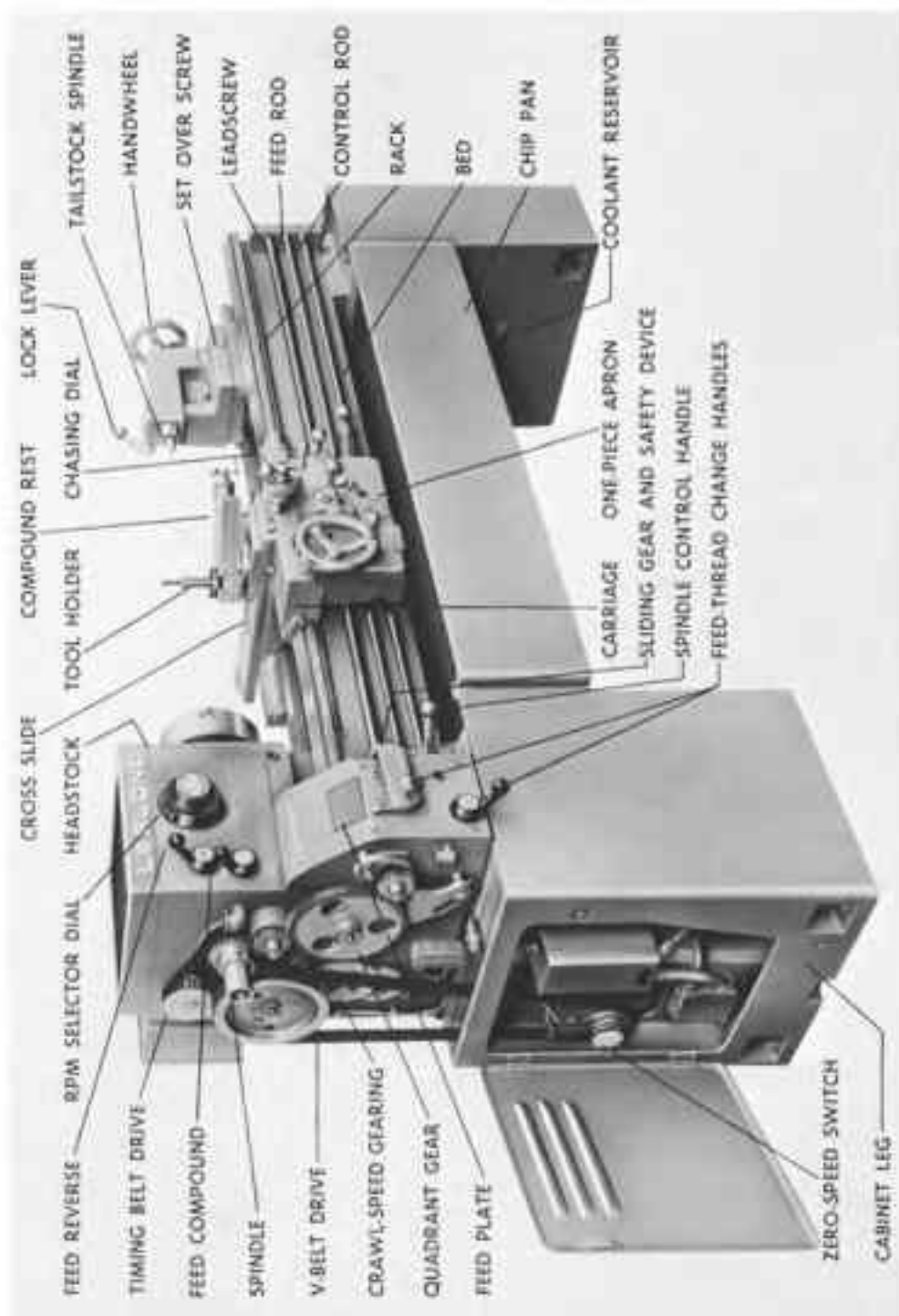
Courtesy of LeBlond Machine Tool Co.

Fig. 6-5. Hollow spindle lathe.

these levers. (See Fig. 6-9.) The speed change mechanism on many smaller lathes consists of stepped pulleys of different diameters as shown in Fig. 6-10. The belt can be on any of the four different pairs of pulleys that provide four different speeds to the lathe spindle. Four additional spindle speeds are obtained through a two-step pulley arrangement on the drive from the electric motor to the countershaft, which contains one of the stepped pulleys, or sheaves, that drives the spindle. Eight additional spindle speeds are obtained by engaging the back gears, thus providing the lathe with a total of 16 spindle speeds. The back gears consist of four gears located inside the headstock that reduce the speed of the stepped pulley in the headstock. When the back gears are engaged this pulley is disconnected from the headstock spindle; when the back gears are disengaged this pulley is connected to the headstock and drives it directly. Another type of headstock construction is shown in Fig. 6-11. A spindle-speed mechanism, located outside the headstock (not shown in Fig. 6-11), may be a stepped pulley arrangement, a gear transmission, or a variable-speed drive. The back-gear drive can be seen in this illustration. It provides the slow spindle speeds.

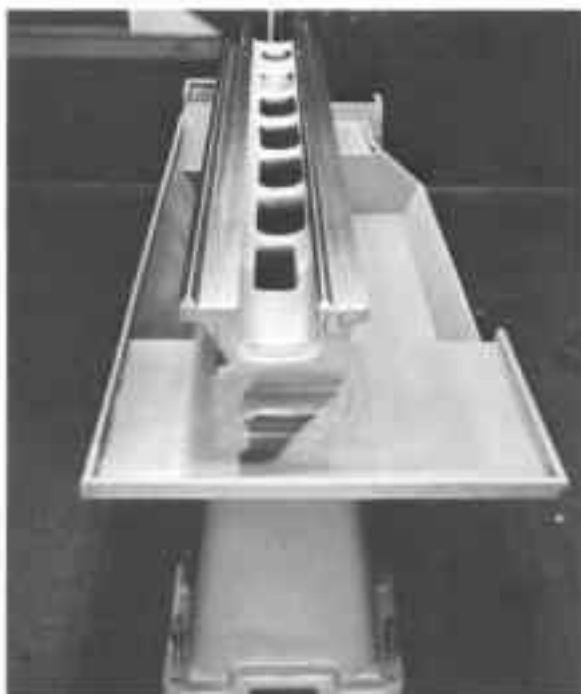
The Headstock Spindle

The lathe headstock spindle is mounted on machine-tool grade anti-friction bearings which can be seen in Fig. 6-11. These bearings are manufactured to closer dimensional tolerances than ordinary anti-friction bearings so that the runout, or eccentricity, of the spindle as it rotates is held to a minimum. The spindle is hollow to allow bar stock and lathe attachments to pass completely through, if desired. The spindle nose is very accurately machined to provide locating surfaces for lathe centers,



Courtesy of LeBlond Machine Tool Co.

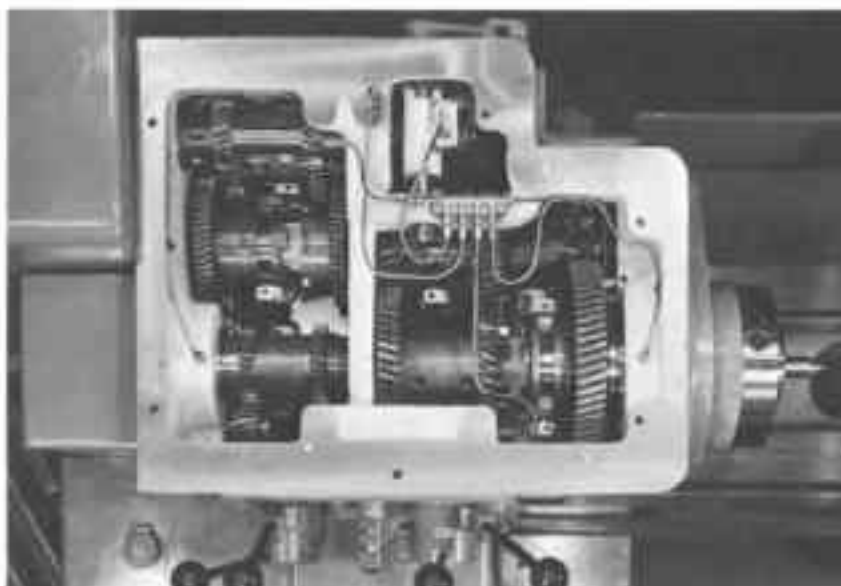
Fig. 6-6. The principal parts of an engine lathe.



Courtesy of The Monarch Machine Tool Company

Fig. 6-7. Lathe bed showing the ways of the lathe.

faceplates, and chucks. The inside of the spindle nose (Fig. 6-12) has a taper which is ground to a high degree of accuracy and surface finish. This taper provides a seat for an adapter which is used to reduce the size of the taper required by the headstock center. The headstock center



Courtesy of The Monarch Machine Tool Company

Fig. 6-8. Geared type headstock.



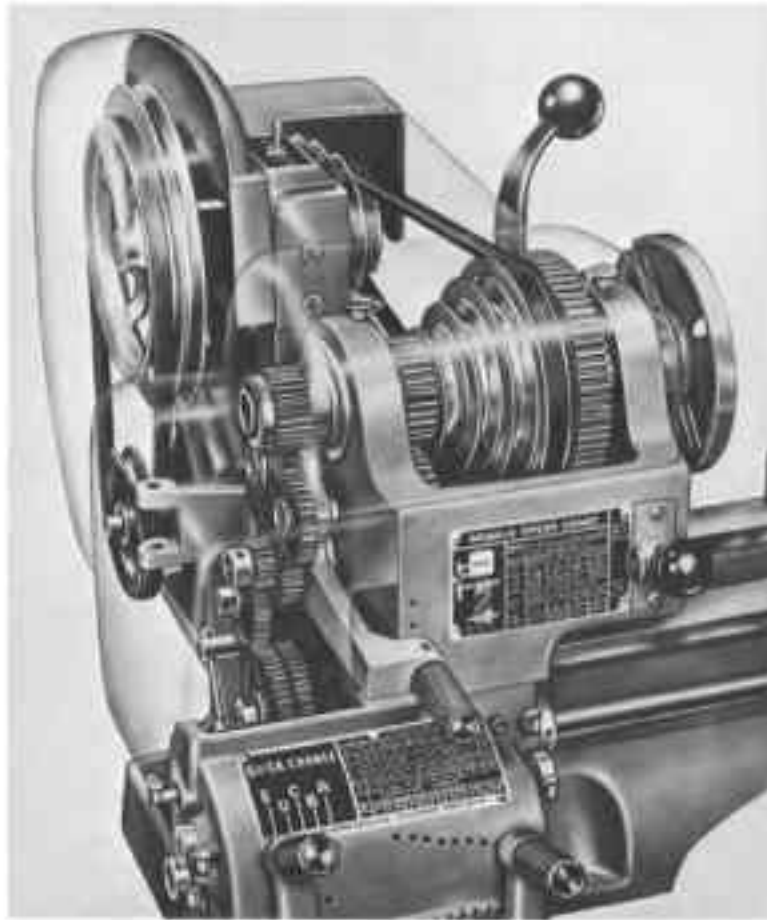
Courtesy of The Monarch Machine Tool Company

Fig. 6-9. Geared type headstock with quick-change gearbox.

is held by the taper in the adapter. The construction of the inside of the spindle nose for all engine lathes is the same, except for size. There are three different designs for the outside of the spindle noses, and these are shown in Fig. 6-13. At the left is the long, tapered type of spindle nose. The spindle nose mountings (chucks and faceplates) are located by the taper and are driven by the key. The threaded ring screws onto a matching thread on the chuck and faceplate, and it holds them securely on the taper. The center view (Fig. 6-13) shows a cam-lock type of spindle nose. It has a short, tapered surface which locates the spindle mountings. The chucks and faceplates have a cam stud (see Fig. 6-12) which fits into each of the holes located above the taper. The cam is turned by a chuck key, to hold the spindle mounting firmly, on the short taper. The cam studs also act as drivers for the mounting. A threaded spindle nose is shown in the view to the right, in Fig. 6-13. The spindle nose mounting is held in place and is driven by the threads. The shoulder behind the threads is used to help locate the mounting on the spindle nose. The spindle nose should be kept free of nicks and scratches, and it should be carefully cleaned before a chuck or a face is mounted on it. Also, the corresponding surfaces of the chuck or faceplate must be clean and kept free from any damage.

The Quick-Change Gearbox

The quick-change gearbox is located at the left side of the lathe below the headstock (see Figs. 6-9 and 13-11 in Chapter 13). It is connected to the spindle by a gear train as shown in Fig. 6-14. The gears in this train are called pick-off gears because they can be interchanged rapidly. Some of these gears are used to cut threads that cannot be cut with the standard pick-off gears and the settings of the quick-change gearbox.

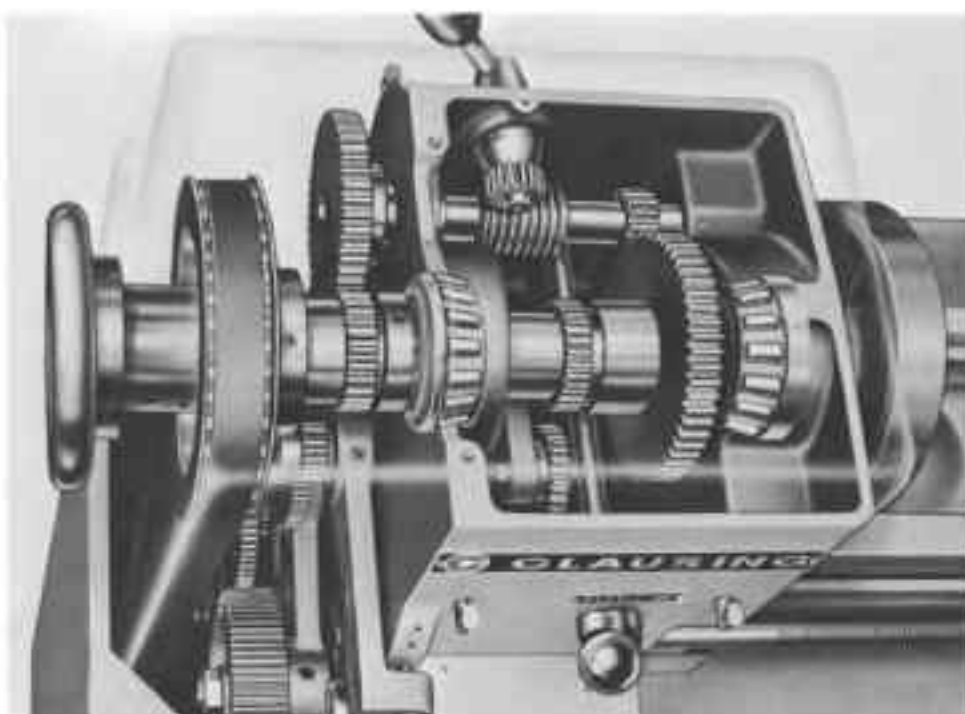


Courtesy of the Atlas Press Company

Fig. 6-10. Stepped pulley type headstock.

The function of the quick-change gearbox is to provide the lathe with the capability of cutting different screw threads and operating at different feed rates. Invented by the American, Frederick B. Miles, and later improved by another American, Wendell P. Norton, it was for many years called the "*Norton Quick-Change Gearbox*." It consists of a tumbler gear which can be positioned to engage any one of a cluster of different size gears to provide a variety of speed ratios. The tumbler gear is engaged by a lever located in front of the gearbox. Often, to provide additional speed ratios, a second set of gears is used and still another, third set can be used to reverse the direction of rotation of the leadscrew and the feed rod. The required number of threads per inch—or thread pitch—and the required feed rate are obtained by positioning the handles according to instructions provided on the quick-change gearbox.

The output shafts of the feed-thread gearbox are the feed rod and the lead screw. The function of the feed rod is to actuate the automatic-power feed mechanism located in the lathe apron. The lead screw is used to feed the carriage in the longitudinal (parallel to the lathe axis) direction when cutting threads. It should be used only to cut threads in order not to impair its accuracy as the result of wear and the imposition of



Courtesy of the Clausing Division of the Atlas Press Company

Fig. 6-11. Modern headstock showing back gears and antifriction spindle bearings.

heavy feeding forces. Some lathes (Fig. 6-6) have a control rod located below the lead screw and feed rod which is used to start and stop the headstock spindle. This rod is actuated by the spindle control handle.

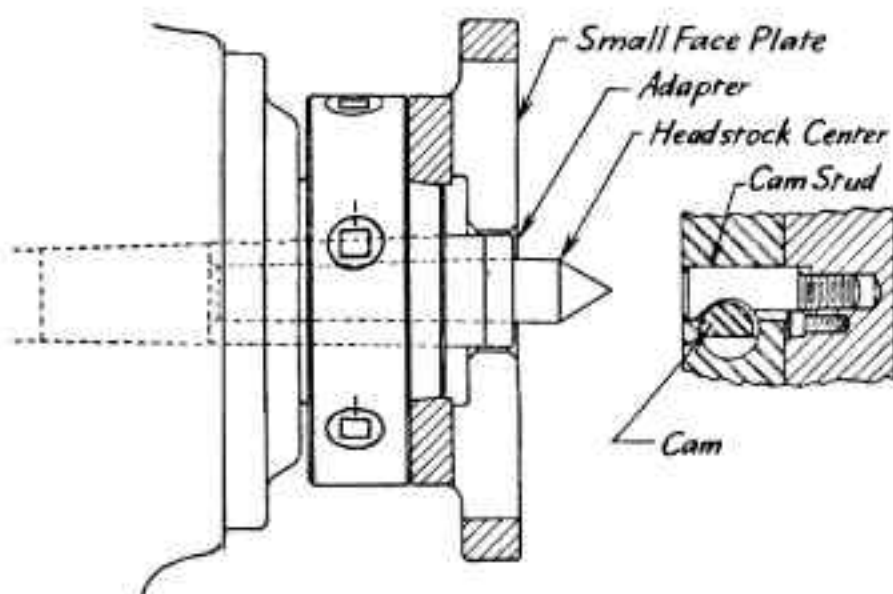


Fig. 6-12. Cam-lock type spindle nose showing adapter and headstock center in place.

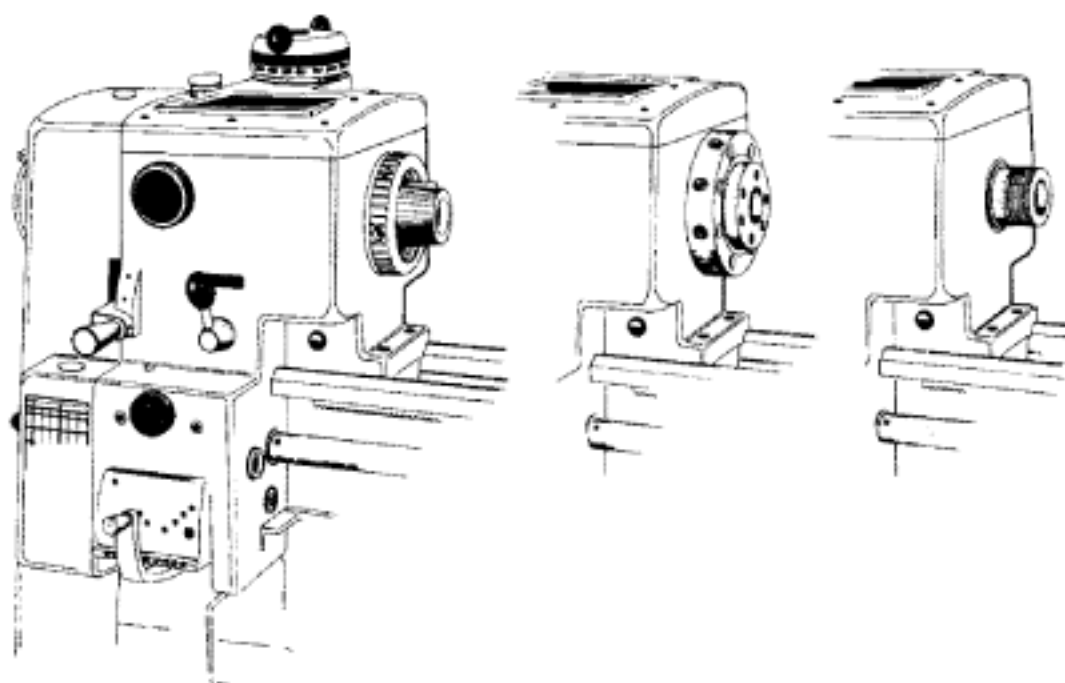


Fig. 6-13. (Left) Long taper type spindle nose. (Center) Cam-lock type spindle nose. (Right) Threaded spindle nose.

The Carriage

The carriage, Fig. 6-15, consists of the saddle, cross slide, compound rest, and apron. The saddle, Fig. 6-16, is an H-shaped casting that slides on the top of the lathe bed. It enables the cutting tool to be moved parallel or perpendicular to the axis of the lathe. The saddle rides on one of the ways on the lathe bed which guides the longitudinal movement of the carriage parallel to the lathe axis.

The Cross Slide

The cross slide moves over the dovetail slide on top of the saddle. The movement of the cross slide is perpendicular to the lathe axis. This movement is actuated by a feed screw, shown in Fig. 6-17. The feed screw can be turned manually by means of the cross-feed handle. The automatic cross feed is actuated by a gear in the apron.

Mounted behind the cross-feed handle is a micrometer cross-feed dial that has accurate graduations cut on its circumference which enables the lathe operator to determine the distance that the cross slide is moved. On many lathes each graduation corresponds to a movement of .0005 inch of the carriage which causes .001 inch to be removed from the diameter of the workpiece that is being turned, because a cut is taken from both sides of the workpiece as it revolves. Cross-feed dials graduated in this manner are said to be "direct reading" dials. For example, if .050 inch is to be removed from the diameter of the workpiece the cross-feed dial is moved 50 graduations. On some lathes the graduations correspond

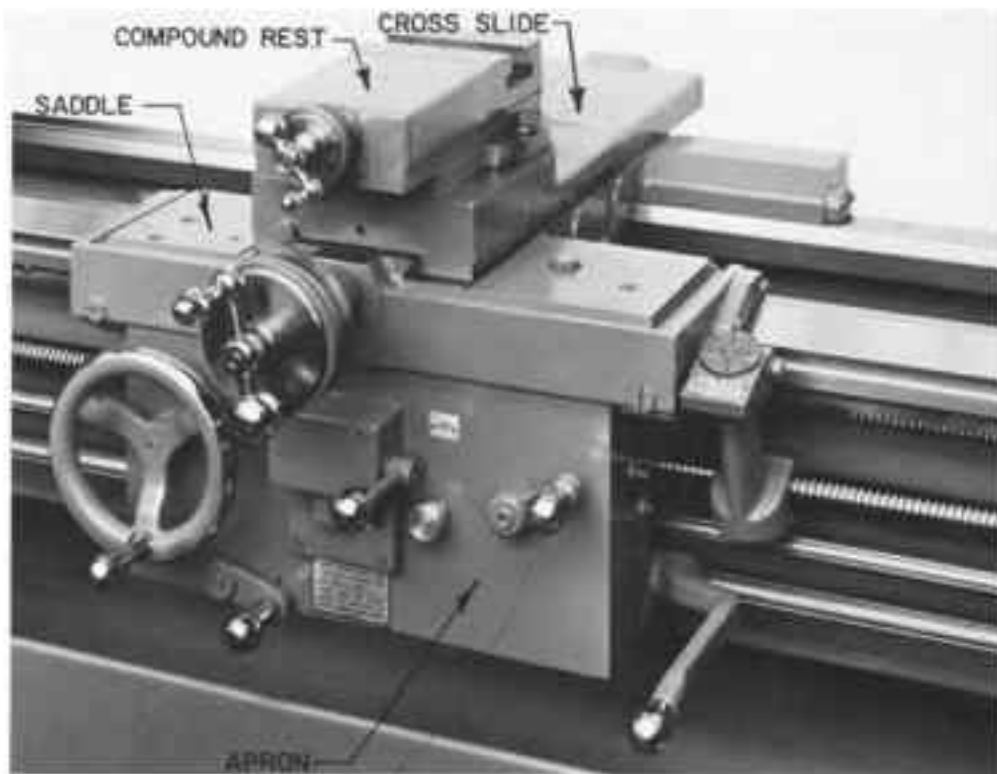


Courtesy of LeBlond Machine Tool Co.

Fig. 6-14. Front end of engine lathe showing gear train from headstock spindle to quick-change gearbox. The pick-off gears in the train can be quickly removed and replaced by other gears.

to a movement of .001 inch, thereby removing .002 inch from the diameter of the workpiece. In this case the amount to be removed from the diameter of the workpiece must be divided by two in order to determine the number of graduations through which the cross-feed dial must be moved. For example, if .050 inch is to be removed from the diameter of the workpiece, the cross-feed dial is moved 25 graduations.

Clearance must be provided between the cross-feed screw and the cross-feed nut (Fig. 6-17) in order to allow the screw to rotate without binding. This clearance is intentional although it results in a certain amount of lost motion between the nut and the screw. Thus, when the cross-feed screw is turned in one direction and then in the opposite direction, it will not move the carriage until the lost motion is taken up. In order to accommodate this lost motion, the cutting tool must be positioned by always moving the cross slide toward the workpiece. If the



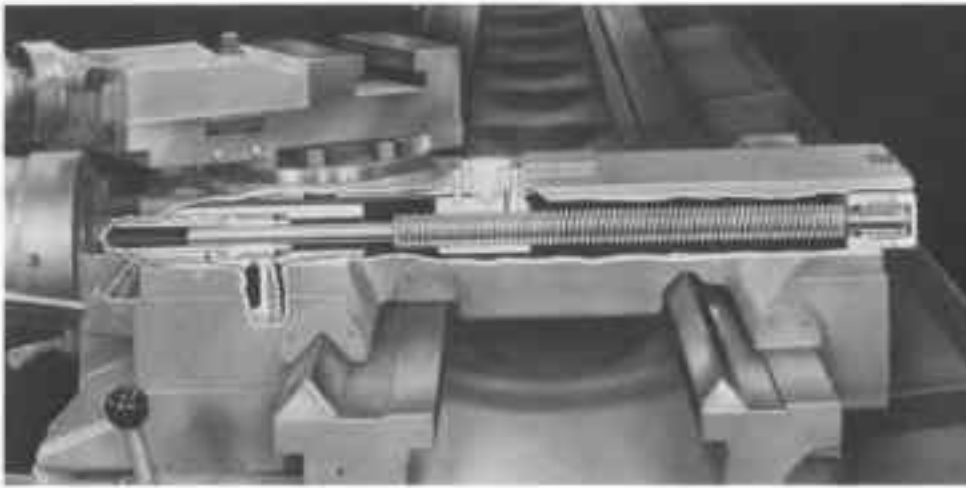
Courtesy of LeBlond Machine Tool Co.

Fig. 6-15. Principal parts of the lathe carriage.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-16. Lathe saddle.



Courtesy of The Monarch Machine Tool Company

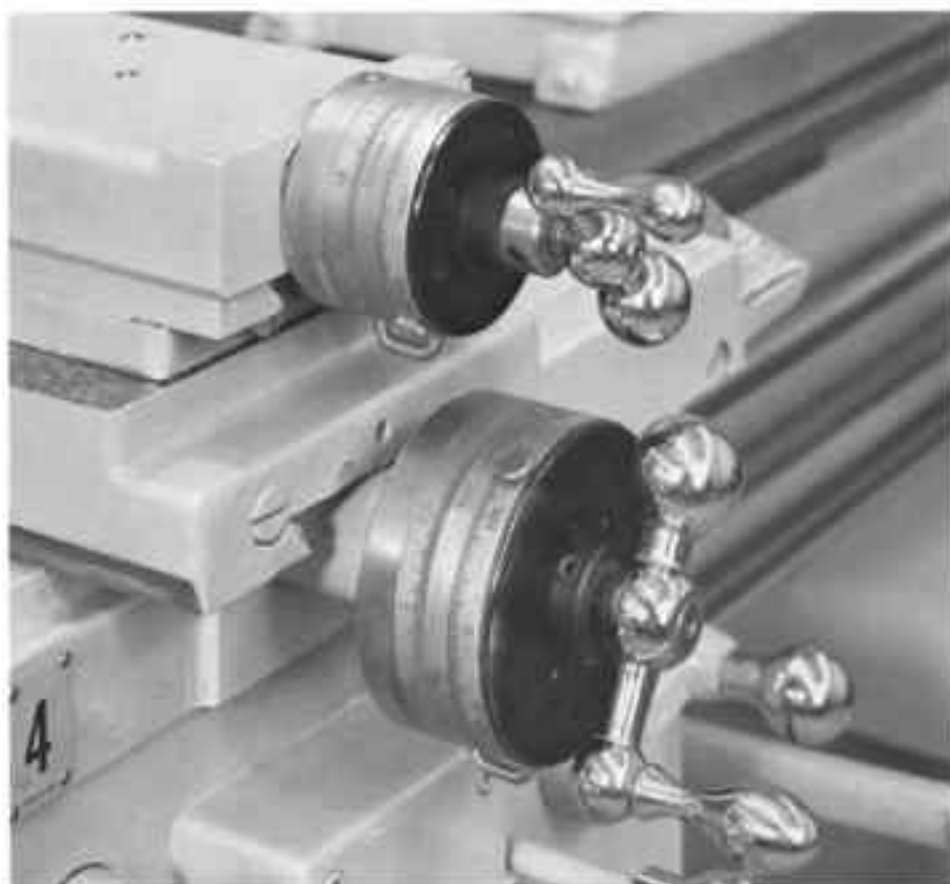
Fig. 6-17. Cross feed screw,

cutting tool must be positioned an exact distance in a direction away from the workpiece axis, the cross slide should be first moved back beyond the desired tool position and then moved forward toward the workpiece until the desired position is reached as determined by the cross-feed dial.

Many lathes are equipped with dual reading micrometer dials on the cross feed and on the compound rest screws as shown in Fig. 6-18. Dual reading dials have two scales, one of which is graduated in inches and the other in millimeters. The dials shown in Fig. 6-18 have one scale reading in .001-inch increments while the other reads 0.02 millimeter per division. Both scales are direct reading. The scales are located on separate dials which do not rotate at the same speed; one dial drives the other through an internal gear train located inside the housing. If the feed screw has an inch thread, the inch dial will rotate at the same speed as the feed screw; otherwise it is the metric dial that will rotate at the same speed as the metric feed screw.

The Compound Rest

The compound rest is clamped to the cross slide and its base can turn through an angle of 360 degrees when the clamp nuts are loosened. The base has graduations enabling it to be clamped at any angle with respect to the lathe axis. The compound-rest slide is moved over the base by a feed screw that is similar in design to the cross-feed screw. The compound-rest feed can only be actuated manually, by turning the compound-rest feed handle. A micrometer dial permits making accurate movements of the compound-rest slide. The compound-rest slide has a large T slot which is used to clamp a tool post or tool block. The cutting tool, or the cutting tool holder, is clamped in the tool post, or tool block. The function of the compound rest is to provide a movement of the cutting tool at any desired angle, with respect to the axis of the work.



Courtesy of LeBlond Machine Tool Co.

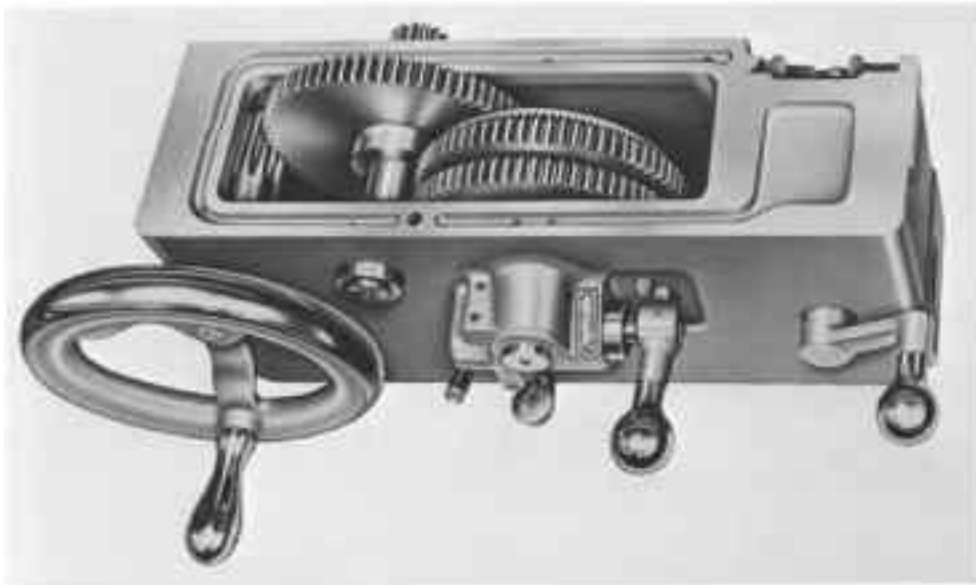
Fig. 6-18. Dual inch-metric micrometer dials on cross feed and compound-rest screws.

The Apron

The apron of the lathe, Fig. 6-19, is fastened to the underside of the saddle. It is essentially a housing that contains the gears and clutches required to actuate the automatic power, longitudinal and cross-feeds. It also supports a large handwheel that actuates the manual longitudinal feed and a lever, shown at the far right, called the "split-nut" lever, which actuates a split nut. The split nut engages the lead screw and drives the carriage when cutting threads. It is disengaged when the thread has been cut to the desired length in order to stop the motion of the carriage. The split-nut lever should never be used to feed the carriage to take any cut other than cutting screw threads. The lever in the center of the apron is used to selectively engage either the cross feed or the longitudinal feed. The longitudinal feed is actuated by a small pinion at the rear of the apron that engages a rack (see Fig. 6-15) which is located on the side of the bed above the lead screw.

The Tailstock

The tailstock is used to support long workpieces and also to hold drills, reamers, and other cutting tools. A drawing of a tailstock showing



Courtesy of LeBlond Machine Tool Co.

Fig. 6-19. Lathe apron.

all of its parts is illustrated in Fig. 6-20. It consists of a base that rests on the bed of the lathe and a body that is clamped to the base. The base, which is aligned with the axis of the lathe by one of the ways of the lathe, has an accurately machined guide that fits the body. The body and the base will slide over the lathe bed until it is in the desired position, then it is clamped to the bed by tightening the clamp bolt nut. The tailstock spindle is prevented from turning, by a key. When the tailstock handwheel is turned, the screw causes the tailstock spindle to slide in or out of the body. The spindle may be clamped by turning the binder handle. The inside of the spindle nose has a taper which is used to hold the tailstock center, drill chucks, or cutting tools such as drills or reamers. This taper must be absolutely clean before a center or other tool is inserted in it. It serves, primarily, to locate; its ability to resist torsion is very limited. It should not be used to hold large drills because the cutting torque of the drill may cause the taper to slip and possibly score the surface. When this occurs, the taper will no longer locate the center or the tool accurately, and it will no longer be able to hold. The repair of a tailstock spindle is a very expensive procedure. Scored tailstock spindle tapers cannot always be repaired, thus requiring the replacement of the entire spindle. The centers, drill chucks, and cutting tools are removed from the tailstock spindle by backing the spindle into the body until the object that is held in the spindle bumps against the plug end of the screw which forces it out. The body of the tailstock can be moved a limited distance perpendicular to the axis of the lathe by unclamping the clamp bolt nut and turning two adjusting screws located on each side of the base. This movement is used to accurately align the headstock and the tailstock spindles. It can also be used to turn tapers. An

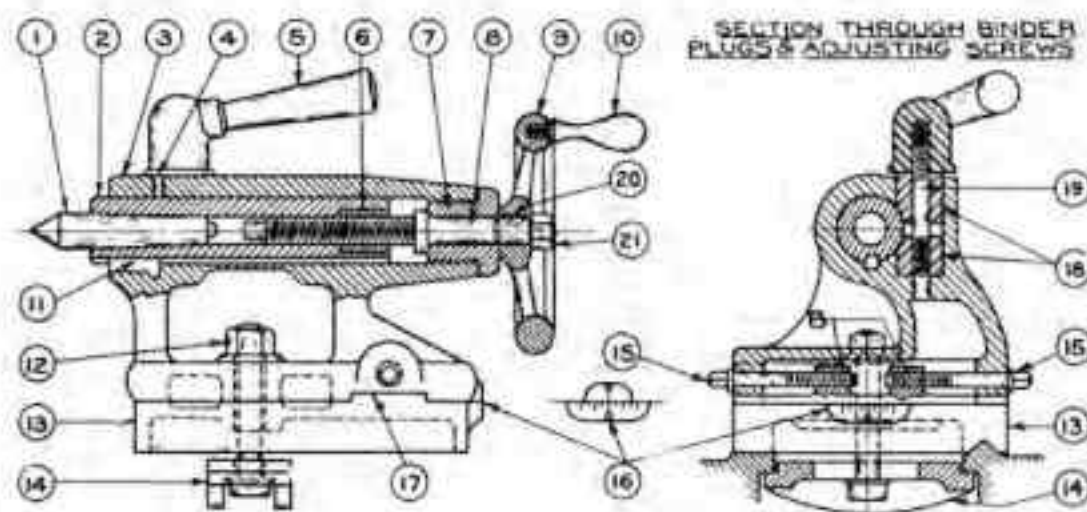


Fig. 6-20. Details of engine lathe tailstock: 1. Tailstock center; 2. Spindle; 3. Body of tailstock (top); 4. Oil hole; 5. Binder handle; 6. Spindle nut; 7. Screw bushing; 8. Screw; 9. Handwheel; 10. Machine handle; 11. Key for spindle; 12. Clamp bolt; 13. Base; 14. Clamp; 15. Adjusting screws; 16. Alignment scale; 17. Guide; 18. Binder plugs; 19. Binder stud; 20. Woodruff key; and 21. Handwheel nut.

alignment scale on the back of the tailstock gives a visual indication of the amount of offset or indicates when the spindle is properly aligned. This scale should not, however, be depended upon for great accuracy.

Engine Lathe Attachments

Many attachments have been developed to increase the usefulness and the accuracy of the engine lathe. Some of these attachments will be discussed in greater detail in subsequent chapters. A group of frequently used engine lathe attachments is shown in Fig. 6-21. A collet chuck and a group of collets are shown at the top of this illustration. A three-jaw universal chuck is shown in the lower left side. Separate jaws for holding smaller outside diameters, and for holding on inside diameters, are shown. The jaws of this chuck are actuated by the chuck key. They move in and out together and hold the round workpiece in a central location without the need for further adjustment. A four-jaw independent chuck is shown in the lower right-hand view. Each jaw is adjusted independently of the other jaws by the chuck key which is the T-shaped key adjacent to the chuck. The jaws can be reversed in the chuck body in order to grip onto inside or outside surfaces. A drill chuck is shown between the three- and four-jaw chucks. The drill chuck is used to hold straight-shank twist drills, reamers, and other cutting tools. It is usually held in the tailstock spindle when used in a lathe. A large faceplate is shown in Fig. 6-22. Workpieces are bolted directly to faceplates to perform various turning and facing operations. Lathe fixtures are also bolted to faceplates. A lathe fixture is a special device to hold and to locate workpieces, in a lathe, that cannot be held by other methods.



Courtesy of The Monarch Machine Tool Company

Fig. 6-21. Engine lathe attachments.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-22. Large faceplate for holding workpieces that cannot conveniently be held by other methods.

A micrometer carriage stop is shown in Fig. 6-23. It is used to obtain accurate positions of the carriage and to move it exact distances, longitudinally. A length rod is extended from the headstock. It can be accu-



Courtesy of The Monarch Machine Tool Company

Fig. 6-23. A micrometer carriage stop.

ately moved in one-inch increments. A micrometer head at the end of the length rod is used to make fine adjustments of less than one inch. The dial test indicator, which is mounted on the carriage, is used to position the carriage so that it will always be at the same distance from the end of the length rod.

A large steady rest is shown in Fig. 6-24. Steady rests are used to support long and slender parts. The steady rest also permits an unobstructed access to the ends of parts so that facing and boring operations can be performed on these ends. The steady rest is composed of a frame containing three jaws. These jaws form a three-point bearing surface and they can be adjusted in or out, radially, by turning the adjusting screws. The frame is hinged on one side allowing the upper half to swing open for inserting or removing the work. The base of the frame has V-grooves in it that fit the ways of the lathe bed. When the steady rest is in use, it is clamped to the bed and the jaws are set against the work thus supporting, or steadying it, during the operation.

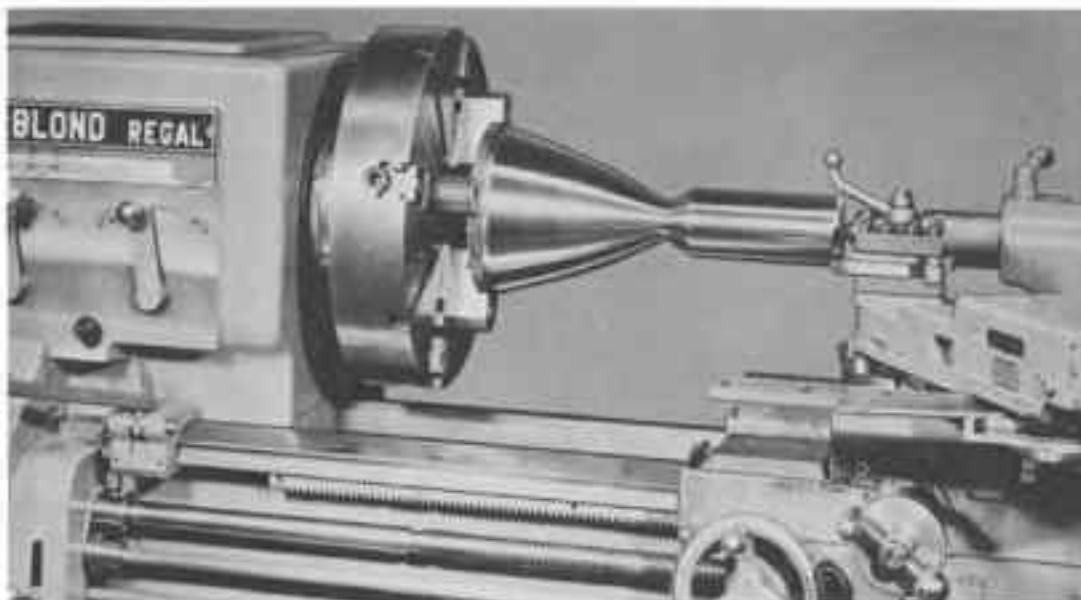
Tracer attachments can be used to turn and to bore irregular contours, as shown in Fig. 6-25. They can also be used to cut stepped shafts with shoulders and to turn tapers. Internal tapers, internal shoulders, and counterbores can be readily cut with this attachment. The tracer attachment is mounted on the carriage in place of the compound rest, as can be seen in Fig. 6-26. A template with the profile of the part to be turned is mounted on the carriage but is prevented from moving with the carriage. The stylus of the tracer attachment follows the profile of the template which causes the cutting tool to turn this profile on the part in a manner to be described.

The regular cross slide is removed from the lathe and replaced by one having a top slide (similar to the compound rest) mounted on it. This slide is actuated by a hydraulic cylinder which has its movement controlled by a stylus. The nose of the stylus must have the same shape as



Courtesy of LeBlond Machine Tool Co.

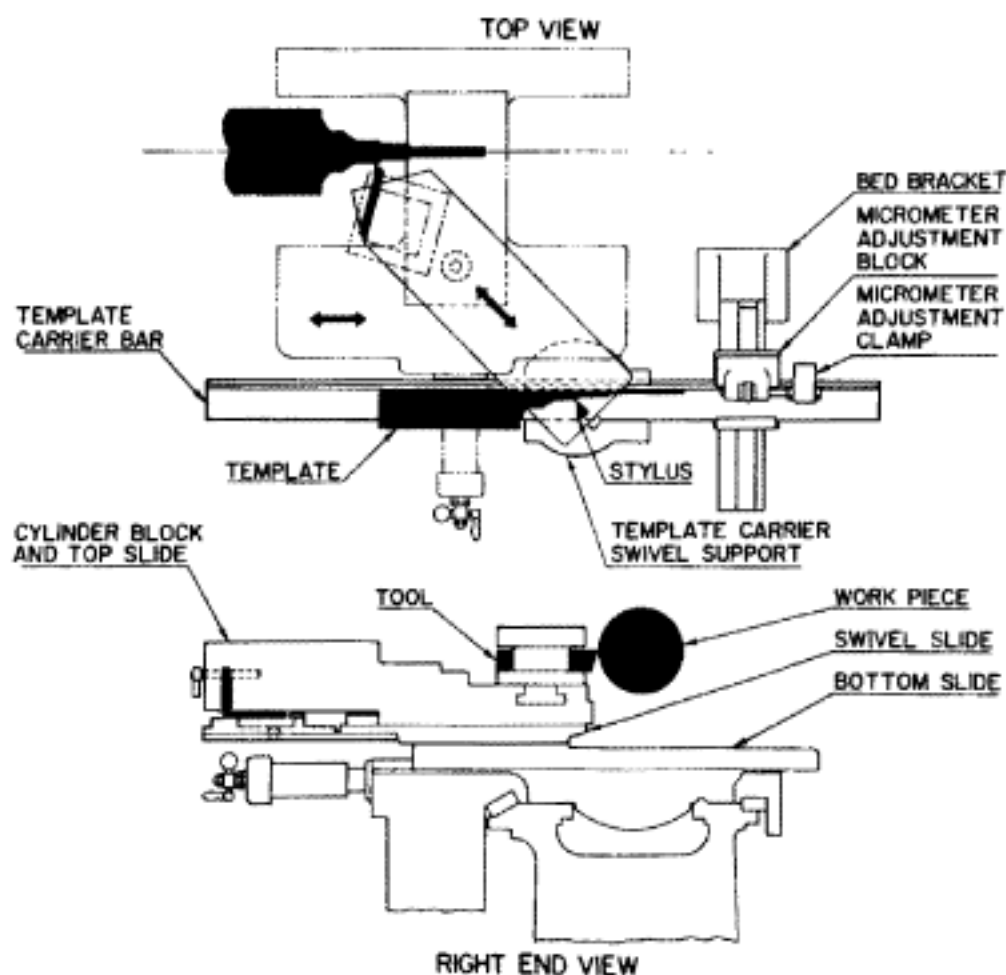
Fig. 6-24. Boring a large workpiece in a lathe with the end held by a steady rest.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-25. Part turned on engine lathe equipped with tracer attachment.

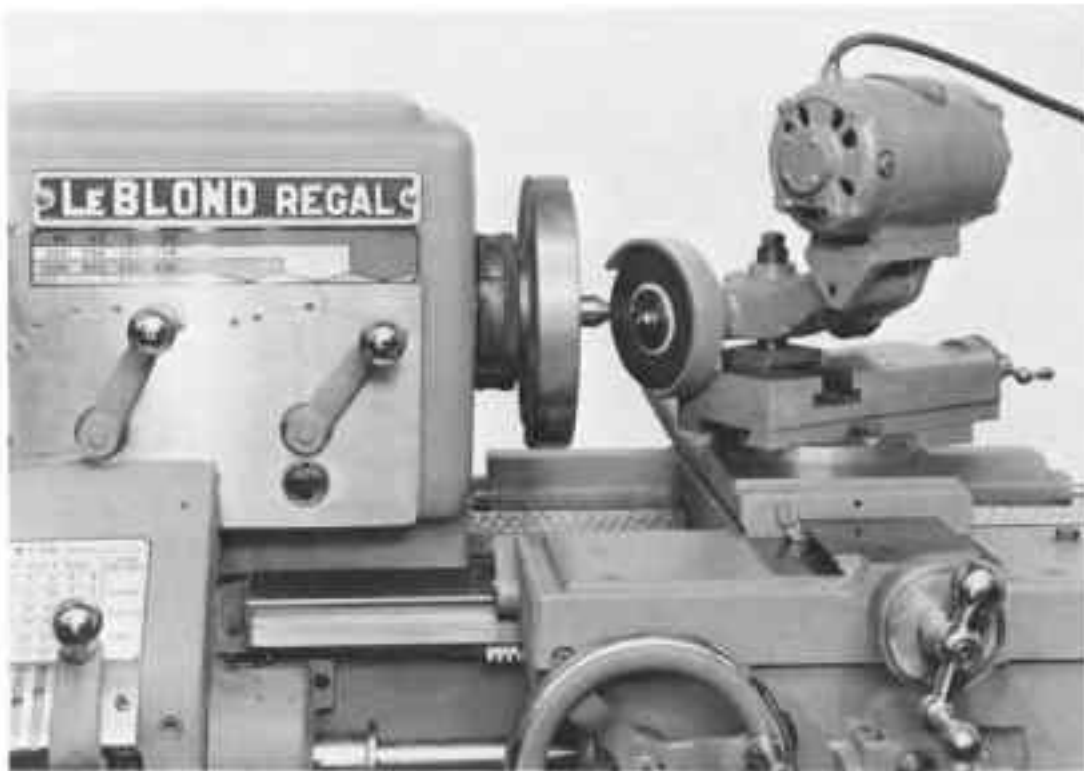
the nose of the cutting tool. The stylus follows the outline of the template which is mounted on the cross slide and fastened to the bed of the lathe. The position of the template can be adjusted by the micrometer adjustment clamp.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-26. Principle of operation of tracer control attachment.

In operation, the first diameter is cut to the required size by taking a trial cut, using the cross slide to set the tool to the required depth of cut; then the stylus is brought into contact with the template and the automatic power, longitudinal feed is engaged. (The trial cut can usually be eliminated on subsequent parts.) The stylus will then pilot the cutting tool causing it to reproduce the template outline on the workpiece in the following manner: the longitudinal feed moves the carriage toward the tailstock at a steady rate; when the stylus detects a change in the diameter of the part, or a shoulder, the top slide will move in or out at an angle, with respect to the axis of the workpiece. This angular movement of the top slide may be considered as being made up of two component motions: one parallel to the lathe axis and one perpendicular to it. For example, when a square shoulder is encountered, the top slide will move at an angle away from the work axis. The rate of movement of that portion of the angular motion of the top slide that is parallel to the lathe axis will be equal and opposite to the longitudinal feed rate. Since these opposite movements are equal and parallel to the lathe axis the tool will, in effect, remain stationary with respect to this direction of



Courtesy of LeBlond Machine Tool Co.

Fig. 6-27. Tool post attachment grinding a lathe center.

movement. However, it will be carried away from the axis of the work by that portion of the top slide motion that is moving away from the work axis. Thus, the tool is caused to feed out just as if the cross feed were being used. Contours are cut by making that portion of the angular movement that is parallel to the lathe axis, operate at a slower or faster rate than the longitudinal feed rate, while the tool is moving toward or away from the work axis.

A tool post grinding attachment is shown grinding a hardened lathe center in Fig. 6-27. Workpieces which have been heat treated to have a high hardness can only be machined by grinding. While most cylindrical and internal grinding is done on machines designed for that purpose, there are occasions when external and internal cylindrical and tapered surfaces are ground on an engine lathe.

Single-Point Cutting Tools and Their Performance

Single-point cutting tools are used on engine lathes, shapers, and planers. Boring tools, which are also single-point cutting tools, are used on a large variety of power machine tools to locate and enlarge holes. The performance of these tools has a direct relation to the quality of the work done and the rate at which it is produced. Treatment of this subject will include discussions of: cutting speeds, single-point cutting tool nomenclature, tool shapes, chip formation, chip control, and tool wear. An understanding of these topics will form a necessary foundation for the successful application of these tools in the shop. While the machinist and the toolmaker can only use those machine tools available to them in their respective shops, they will frequently have the responsibility of selecting the best cutting tool and cutting-tool shape available; and, they must know the correct cutting speed and feed with which these tools are to be used.

Hardness and Hardness Testing

Three factors have a very decisive influence on the cutting speed that can be used:

1. The cutting tool material.
2. The material that is to be cut.
3. The metallurgical condition of the material when it is being cut.

One indication of the metallurgical condition of a metal is its hardness; technically this is usually measured by determining its resistance to penetration. The most common hardness-testing machines used in the United States are the Brinell hardness tester and the Rockwell hardness tester. The diameter of the impression made by a steel ball is measured and this measurement is converted to a Brinell hardness number, abbreviated HB, which defines its hardness. Harder metals have a higher HB. There are several Rockwell hardness scales. One which is frequently encountered in measuring the hardness of hardened ferrous metals is the Rockwell C Scale, that is abbreviated HRC. It, too, has the higher numbers indicating higher hardnesses. In general, the Brinell hardness reading is given for softer ferrous metals and the Rockwell reading is given

for hardened ferrous metals, although there is a range in which they overlap.

The strength of a metal increases or decreases roughly in proportion to its hardness. It is not surprising then to discover that harder metals must be cut more slowly than softer metals. The hardness of any piece of soft metal can be increased by *cold drawing* or *cold rolling*. Annealing and normalizing are heat treating processes used to decrease the hardness of steel and cast iron although normalizing may be used to increase the hardness of a piece of steel that has been previously annealed. Most steels, when heated above a specific temperature and then quenched and tempered, will have their hardness increased. *Solution treating and aging* is another heat treatment process which is used to increase the hardness of some metals, particularly aluminum. The details of these processes are a subject in the study of metallurgy; however, the machinist and toolmaker must be aware that the condition of the metals that they are cutting is affected by these processes and that any particular type of metal can have different hardnesses. In fact, it is quite common to have a variation in hardness in a *single* piece of metal. It should now be clear that a cutting speed cannot be given for a specific type of metal without also specifying the hardness. The cutting speeds in Tables 7-1 through 7-5 are, therefore, listed not only for a given metal but also for the different hardnesses at which the metal is commonly cut.

Steels used in the construction of machinery are commonly designated by AISI (American Iron and Steel Institute) numbers. Tool steels are specified by a letter to identify the type, which is followed by a number, such as shown in Table 7-3. Other metals have similar methods of specification. These specifications are in common use in industry and they are used in this book.

Cutting Speed

The cutting speed is given in terms of feet per minute (fpm) in the customary inch system of units. Sometimes this is called surface feet per minute (sfpm). In the metric system of units the cutting speed is specified in meters per minute (m/min). Tables 7-1 through 7-5 give the recommended cutting speeds for cutting many materials in terms of feet per minute. These cutting speeds must be considered to be a starting point which may have to be modified as determined by actual experience on the job. When only a few parts are to be machined these speeds may be used without change, but when machining many parts that are alike and when maximum production must be obtained, a modification may be desirable. Many factors affect the cutting speed which may combine to warrant an increase or sometimes a decrease in the cutting speed. These factors will now be considered.

Tool Material. The type of material used to make the cutting tool is of prime importance with respect to the cutting speed used to cut any material. Cutting tool materials are treated in greater detail elsewhere

Table 7-1. Recommended Cutting Speeds in Feet per Minute for Turning Plain Carbon and Alloy Steels

Material (AISI and SAE Steels)	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Free Machining Plain Carbon Steels (Resulphurized), 1212, 1213, 1215	100-150 150-200	HR, A CD	150 160	600 625
1108, 1109, 1115, 1117, 1118, 1120 1126, 1111	100-150 150-200	HR, A CD	130 120	500 525
1132, 1137, 1139, 1140, 1144, 1146, 1151	175-225 275-325 325-375 375-425	HR, A, N, CD Q and T Q and T Q and T	120 75 50 40	400 300 225 200
Free Machining Plain Carbon Steels (Leaded), 11L17, 11L18, 12L13, 12L14	100-150 150-200 200-250	HR, A, N, CD HR, A, N, CD N, CD	140 145 110	550 560 400
Plain Carbon Steels, 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	100-125 125-175 175-225 225-275	HR, A, N, CD HR, A, N, CD HR, N, CD CD	120 110 90 70	450 400 350 300
1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	125-175 175-225 225-275 275-325 325-375 375-425	HR, A, N, CD HR, A, N, CD N, CD, Q and T Q and T Q and T Q and T	100 85 70 60 40 30	375 325 225 200 160 140
1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125-175 175-225 225-275 275-325 325-375 375-425	HR, A, N, CD HR, A, N, CD N, CD, Q and T Q and T Q and T Q and T	100 80 65 50 35 30	370 320 220 180 150 130
Free Machining Alloy Steels (Resulphurized), 4140, 4150	175-200 200-250 250-300 300-375 375-425	HR, A, N, CD HR, N, CD Q and T Q and T Q and T	110 90 65 50 40	400 350 300 225 165
Free Machining Alloy Steels (Leaded), 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40 *	150-200 200-250 250-300 300-375 375-425	HR, A, N, CD HR, N, CD Q and T Q and T Q and T	120 100 75 55 50	430 380 275 220 200
Alloy Steels, 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125-175 175-225 225-275 275-325 325-375 375-425	HR, A, N, CD HR, N, CD CD, N, Q and T Q and T Q and T Q and T	100 90 70 60 50 35	400 350 300 250 200 175

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.

* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; and HB, Brinell hardness number.

Table 7-1 (Concluded). Recommended Cutting Speeds in Feet per Minute for Turning Plain Carbon and Alloy Steels

Material AISI and SAE Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Alloy Steels, 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30	175-225 225-275 275-325 325-375 375-425	HR, A, N, CD N, CD, Q and T N, Q and T N, Q and T Q and T	85 70 60 40 30	325 275 230 200 150
Alloy Steels, E51100, E52100	175-225 225-275 275-325 325-375 375-425	HR, A, CD N, CD, Q and T N, Q and T N, Q and T Q and T	70 65 50 30 20	310 260 220 180 140
Ultra High Strength Steels (Not AISI) AMS 6421 (98B37 Mod.), AMS 6422 (98BV40), AMS 6424, AMS 6427, AMS 6428, AMS 6430, AMS 6432, AMS 6433, AMS 6434, AMS 6436, AMS 6442, 300M, D6ac	220-300 300-350 350-400 43-48 HRC 48-52 HRC	A N N Q and T Q and T	65 50 35 25 10	270 200 150 120 80
Maraging Steels (Not AISI) 18% Ni Grade 200 18% Ni Grade 250 18% Ni Grade 300 18% Ni Grade 350	250-325 50-52 HRC	A Maraged	60 10	300 80
Nitriding Steels (Not AISI) Nitalloy 125 Nitalloy 135 Nitalloy 135 Mod. Nitalloy 225 Nitalloy 230 Nitalloy N Nitalloy EZ Nitrex 1	200-250 300-350	A N, Q and T	70 30	300 225

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.

* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; HB, Brinell hardness number; and HRC, Rockwell C scale hardness number.

in this chapter; therefore, only a few words will suffice here. The tables list the cutting speeds for use with high-speed steel and cemented carbide cutting tools. Although there are slight differences in the cutting speed that can be used with tools made of different types of high-speed steel, these differences are generally small. A greater difference exists between different grades of cemented carbides and it is essential that the correct grade be used. The tables are based on the assumption that the correct grade of uncoated carbide is used. Coated carbides cannot be used to cut

Table 7-2. Recommended Cutting Speeds in Feet per Minute for Turning Tool Steels

Material Tool Steels (AISI Types)	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Water Hardening W1, W2, W5	150-200	A	100	325
Shock Resisting S1, S2, S5, S6, S7	175-225	A	70	300
Cold Work, Oil Hardening O1, O2, O6, O7	175-225	A	70	250
Cold Work, High Carbon High Chromium D2, D3, D4, D5, D7	200-250	A	45	175
Cold Work, Air Hardening A2, A3, A8, A9, A10 A4, A6 A7	200-250 200-250 225-275	A A A	70 55 45	250 200 175
Hot Work, Chromium Type H10, H11, H12, H13, H14, H19	150-200	A	80	300
	200-250	A	65	225
	325-375	Q and T	50	175
	48-50 HRC	Q and T	20	95
	50-52 HRC	Q and T	10	80
	52-54 HRC	Q and T	—	60
Hot Work, Tungsten Type H21, H22, H23, H24, H25, H26	150-200	A	60	250
	200-250	A	50	200
Hot Work, Molybdenum Type H41, H42, H43	150-200	A	55	225
	200-250	A	45	175
Special Purpose, Low Alloy L2, L3, L6	150-200	A	75	325
Mold P2, P3, P4, P5, P6 P20, P21	100-150	A	90	400
	150-200	A	80	350
High Speed Steel M1, M2, M6, M10, T1, T2, T6 M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8 T15, M3-2	200-250	A	65	225
	225-275	A	55	200
	225-275	A	45	170

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.

* Abbreviations designate: A, annealed; Q and T, quenched and tempered; HB, Brinell hardness number; and HRC, Rockwell C scale hardness number.

all materials; however, where they can be used, they can usually be operated at a 20 to 40 percent faster cutting speed than given in the tables and sometimes by as much as 50 percent. Since there are so many grades of cemented carbides, the selection of a specific grade should be made by obtaining the recommendations of a carbide producer.

Work Material. Of equal importance in selecting the cutting speed is the

Table 7-3. Recommended Cutting Speeds in Feet per Minute for Turning Stainless Steels

Material Stainless Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Free Machining Stainless Steels				
(Ferritic), 430F, 430F Se	135-185	A	110	400
(Austenitic), 203EZ, 303, 303Se,	135-185	A	100	350
303MA, 303Pb, 303Cu, 303 Plus X	225-275	CD	80	325
	135-185	A	110	400
(Martensitic), 416, 416Se, 416 Plus X,	185-240	A, CD	100	350
420F, 420F Se, 440F, 440F Se	275-325	Q and T	60	250
	375-425	Q and T	30	125
Stainless Steels				
(Ferritic), 405, 409, 429, 430, 434,	135-185	A	90	300
436, 442, 446, 502				
(Austenitic), 201, 202, 301, 302, 304,	135-185	A	75	225
304L, 305, 308, 321, 347, 348	225-275	CD	65	200
(Austenitic), 302B, 309, 309S, 310,	135-185	A	70	225
310S, 314, 316, 316L, 317, 330				
	135-175	A	95	350
(Martensitic), 403, 410, 420, 501	175-225	A	85	300
	275-325	Q and T	55	200
	375-425	Q and T	35	125
(Martensitic), 414, 431,	225-275	A	60	250
Greek Ascoloy	275-325	Q and T	50	200
	375-425	Q and T	30	125
(Martensitic), 440A, 440B, 440C	225-275	A	55	200
	275-325	Q and T	45	150
	375-425	Q and T	30	125
(Precipitation Hardening)				
15-5PH, 17-4PH, 17-7PH, AF-71,	150-200	A	60	225
17-14CuMo, AFC-77, AM-350,	275-325	H	50	200
AM-355, AM-362, Custom 455,	325-375	H	40	130
HNM, PH13-8, PH14-8Mo,	375-450	H	25	90
PH15-7Mo, Stainless W				

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.

* Abbreviations designate: A, annealed; CD, cold drawn; Q and T, quenched and tempered; H, precipitation hardened; and HB, Brinell hardness number.

work material. Each work material will have a range of cutting speeds at which it can be cut. The cutting speeds recommended in this chapter are based on the expectancy that the material can be cut at the given speed with a reasonable tool life. It has already been explained that most metals can have a different hardness, depending on how the metal has been heat treated or cold worked. For most materials, the cutting speeds in the tables are given for several different hardness levels. Since the hardness is not always known by the machine operator, the material condition associated with the hardness is provided in a separate column. In this way the machinist might estimate the hardness. When in doubt, start at the lower cutting speed and then see if the workpiece can be cut

Table 7-4. Recommended Cutting Speeds in Feet per Minute for Turning Ferrous Cast Metals

Material Ferrous Cast Metals	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Gray Cast Iron				
ASTM Class 20	120-150	A	120	450
ASTM Class 25	160-200	AC	90	350
ASTM Class 30, 35, and 40	190-220	AC	80	275
ASTM Class 45 and 50	220-260	AC	60	200
ASTM Class 55 and 60	250-320	AC, HT	35	125
ASTM Type 1, 1b, 5 (Ni Resist)	100-215	AC	70	225
ASTM Type 2, 3, 6 (Ni Resist)	120-175	AC	65	210
ASTM Type 2b, 4 (Ni Resist)	150-250	AC	50	200
Malleable Iron				
(Ferritic), 32510, 35018	110-160	MHT	130	500
(Pearlitic), 40010, 43010, 45006, 45008, 48005, 50005	160-200	MHT	95	400
	200-240	MHT	75	275
(Martensitic), 53004, 60003, 60004	200-255	MHT	70	250
(Martensitic), 70002, 70003	220-260	MHT	60	225
(Martensitic), 80002	240-280	MHT	50	140
(Martensitic), 90001	250-320	MHT	30	125
Nodular (Ductile) Iron				
(Ferritic), 60-40-18, 65-45-12	140-190	A	100	450
(Ferritic-Pearlitic), 80-55-06	190-225	AC	80	350
	225-260	AC	65	210
(Pearlitic-Martensitic), 100-70-03	240-300	HT	45	175
(Martensitic), 120-90-02	270-330	HT	30	100
	330-400	HT	15	50
Cast Steels				
(Low Carbon), 1010, 1020	100-150	AC, A, N	110	400
(Medium Carbon), 1030, 1040, 1050	125-175	AC, A, N	100	400
	175-225	AC, A, N	90	350
	225-300	AC, HT	70	300
(Low Carbon Alloy), 1320, 2315, 2320 4110, 4120, 4320, 8020, 8620	150-200	AC, A, N	90	350
	200-250	AC, A, N	80	325
	250-300	AC, HT	60	250
(Medium Carbon Alloy), 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175-225	AC, A, N	80	325
	225-250	AC, A, N	70	280
	250-300	AC, HT	55	250
	300-350	AC, HT	45	220
	350-400	HT	30	150

Based on a feed rate of .012 inch per revolution and a depth of cut of .125 inch.

* Abbreviations designate: A, annealed; AC, as cast; N, normalized; HT, heat treated; MHT, malleablizing heat treatment; and HB, Brinell hardness number.

Table 7-5. Recommended Cutting Speed, in Feet Per Minute, for Turning Nonferrous Metals and Alloys

Material	Material Condition	Cutting Speed, fpm	
		HSS	Carbide
All Wrought Aluminum Alloys	CD	600	1200
	ST and A	500	1100
Aluminum Sand and Permanent Mold Casting Alloys	AC	750	1400
	ST and A	600	1200
All Aluminum Die Casting Alloys*	AC	125	550
	ST and A	100	450
*Except Alloys 390.0 and 392.0	AC	80	500
	ST and A	60	425
All Magnesium Alloys	A, CD, AC, ST and A	800	2000
Leaded Commercial Bronze, Leaded Brass, Free-Cutting Brass, Free Cutting Muntz Metal, Forging Brass, Architectural Bronze, Leaded Naval Brass, Free-Cutting Phosphor Bronze	A	300	650
	CD	350	600
Red Brass, Low Brass, Jewelry Bronze, Cartridge Brass, Muntz Metal, Admiralty Brass, Low-Silicon Bronze, High-Silicon Bronze, Manganese Bronze, Aluminum Brass, Leaded Nickel Silver	A	200	500
	CD	250	550
Oxygen-Free Copper, Beryllium Copper, Electrolytic Tough Pitch Copper, Phosphorus Deoxidized Copper, Commercial Bronze, Phosphor Bronze, Aluminum Bronze, Copper Nickel, Nickel Silver	A	100	200
	CD	110	225
Commercially Pure Titanium	110-150 HB	110	400
	180-240 HB	90	300
	250-275 HB	70	250

*Abbreviations designate: A, annealed; AC, as cast; CD, cold drawn; ST and A, solution treated and aged; HB, Brinell hardness number.

faster. Other work material factors that influence the cutting speed are surface scale, hard spots in castings and weldments, and hard work-hardened layers resulting from previous machining or cold working operations. Some variations in the hardness and microstructure may occur within a single bar of a wrought material, such as steel. Noticeable variations in the ability to be cut by machining may occur between bars of the same type of steel made from different heats, as a result of steel mill practices.

Tool Life. Tool life is the length of time that a cutting tool will cut before it becomes dull or requires replacement. When cutting most metals, cutting tools will operate successfully over a rather wide range of cutting speeds, up to a cutting speed where the tool life is too short to be acceptable. Below this speed the tools will behave in the following manner when normal flank wear occurs and an abnormal type of tool failure does not occur.

A reduction in the cutting speed will cause a much larger increase in the tool life. Conversely, increasing the cutting speed will cause a much larger reduction in the tool life.

This is a very important performance, characteristic of all metal cutting tools, that should be remembered. For example, when the cutting speed is reduced from 350 fpm to 300 fpm while turning AISI 1040 steel with a cemented carbide tool, the tool life will be increased from approximately 30 minutes to 50 minutes. This is a 67 percent increase in tool life obtained by a 14 percent reduction in the cutting speed. When, in this case, the cutting speed is reduced about 29 percent, from 350 fpm to 250 fpm, the tool life will be increased 200 percent to approximately 90 minutes. Even larger percentage increases will occur when using high-speed steel cutting tools.

Cutting tools can perform successfully over a rather wide range of cutting speeds, except when cutting very hard materials and certain other difficult-to-machine materials. The cutting speed selected should be based on a desired tool life. If the cutting speed used is found to result in a tool life that is too short, the first remedial step should be to reduce the cutting speed. While a slow cutting speed will result in a very long tool life, this is always at the expense of an unnecessarily long cutting time and an increased part cost.

Feed and Depth of Cut. The feed rate and the depth of cut also have an effect on the tool life and, therefore, on the cutting speed. The cutting speeds in Tables 7-1 through 7-5 are based on using a feed rate of .012 in./rev (0.30 mm/rev) and a depth of cut of .125 in. (3.18 mm). When a different feed or depth of cut is to be used, the cutting speed should be changed accordingly in order to have approximately the same tool life as before. The amount of change in the cutting speed can be calculated by applying the factors for feed and depth of cut in Table 7-6. These factors should not, however, be used with Table 7-7 because their validity for many nonferrous metals and alloys is uncertain. Applying the feed and depth of cut factors will show that when light cuts are taken the cutting speed can be increased and when heavy roughing cuts are taken it should be decreased.

Cutting Tool Geometry. This topic is treated at length elsewhere in this

Table 7-6. Cutting Speed Feed and Depth of Cut Factors for Turning *

Feed, in./rev.	Feed Factor, F_f	Depth of Cut, in.	Depth of Cut Factor, F_d
.002	1.50	.005	1.50
.003	1.50	.010	1.42
.004	1.50	.016	1.33
.005	1.44	.031	1.21
.006	1.34	.047	1.15
.007	1.25	.062	1.10
.008	1.18	.078	1.07
.009	1.12	.094	1.04
.010	1.08	.100	1.03
.011	1.04	.125	1.00
.012	1.00	.150	.97
.013	.97	.188	.94
.014	.94	.200	.93
.015	.91	.250	.91
.016	.88	.312	.88
.018	.84	.375	.86
.020	.80	.438	.84
.022	.77	.500	.82
.025	.73	.625	.80
.028	.70	.688	.78
.030	.68	.750	.77
.032	.66	.812	.76
.035	.64	.938	.75
.040	.60	1.000	.74
.045	.57	1.125	.73
.050	.55	1.250	.72
.060	.50	1.375	.71

* For use in conjunction with Table 7-1, 7-2, 7-3, and 7-4 only.

chapter. It is, therefore sufficient to say here that the shape of the cutting tool has some effect on the cutting speed.

Cutting Fluids. Coolant type cutting fluids, when correctly applied, will lower the temperature of the cutting tool and tend to increase the tool life. As an alternative, the coolant will cool the tool enough so that a higher cutting speed can be used without overheating. The degree of improvement that can be obtained depends on the type of coolant used.

Selecting the Cutting Conditions.

The tool life of a cutting tool is most affected by the cutting speed, secondly by the feed rate, and least by the depth of cut. This important principle of metal cutting should be memorized. Stated in another way, reducing the cutting speed will increase the tool life more than a reduction in the feed rate; and a reduction in the feed rate will increase the tool life more than a reduction in the depth of cut. Conversely, increasing the depth of cut will reduce the tool life less than an increase in the feed rate; and, increasing the feed rate will reduce the tool life less than an increase in the cutting speed. The latter principle leads to the logical sequence of selecting the cutting conditions, which are given below:

1. Select the depth of cut. Use the largest depth of cut possible, as determined by the amount of metal to be removed from the part and by the available power on the machine.
2. Select the feed rate. Use the heaviest feed rate possible consistent with the surface finish requirement on the workpiece, the rigidity of the machine and workpiece, and the available power on the machine.
3. Select the cutting speed. Use an appropriate table of cutting speeds as a guide, such as provided in this book.
4. Find the appropriate feed and depth of cut factors, when available. Use Table 7-6 in this book, but only with the cutting speed tables provided in this book and in *Machinery's Handbook*. Do not use this table for nonferrous metals and alloys.
5. Calculate the cutting speed and machine spindle speed. How this is done will be shown in the next section of this chapter.
6. Check to see if the machine has sufficient power. This step is optional. In actual practice, this is not done by many persons. When taking only a medium or light cut it is unnecessary. When taking a very heavy cut it is a useful step to determine if the planned cut is within the capacity of the machine. Details on how this is done are given further on in this chapter.

Calculating the Cutting Speed

The formulas used to calculate the cutting speed and the spindle speed of the lathe are given below for both customary inch and metric units. Either inch or metric units can be used in Equation 7-1. The answer will be in terms of the units used for the cutting speed selected from the table.

$$V = V_o F_f F_d \quad (7-1)$$

$$N = \frac{12 V}{\pi D} \quad (\text{Inch units only}) \quad (7-2)$$

$$N = \frac{1000 V}{\pi D} \quad (\text{Metric units only}) \quad (7-3)$$

Where:

V = Cutting speed as modified for feed and depth of cut, or from Table 7-5, fpm or m/min

V_o = Cutting speed from Tables 7-1 through 7-4, fpm or m/min

F_f = Feed factor, from Table 7-6

F_d = Depth of cut factor, from Table 7-6

N = Spindle speed of lathe, rpm

D = Diameter of workpiece, in. or mm. (This is the diameter before turning, not the finish turned diameter.)
 $= 3.14$ (pi)

Example 7-1:

Calculate the cutting speed and the spindle speed for turning a 1.500 in. diameter bar of AISI 4340 steel having a hardness of 220 HB using a cemented carbide (uncoated) cutting tool. The depth of cut selected is .200 in. and the feed selected is .020 in./rev.

$$\begin{aligned}
 V_o &= 275 \text{ fpm} && \text{(from Table 7-1)} \\
 F_f &= .80 && \text{(from Table 7-6)} \\
 F_d &= .93 && \text{(from Table 7-6)} \\
 V &= V_o F_f F_d = 275 \times .80 \times .93 \\
 &= 200 \text{ fpm} \\
 N &= \frac{12 V}{\pi D} = \frac{12 \times 200}{\pi \times 1.500} \\
 &= 500 \text{ rpm}
 \end{aligned}$$

In calculations of this type the answers are always rounded-off; e.g., the calculated cutting speed is 204.6 fpm, which is rounded-off to 200 fpm. The actual spindle speed that would be used is the closet available on the lathe. Sometimes, but not always, when the closest available speed is much faster than the calculated spindle speed, the closest lower spindle speed is used.

Example 7-2:

Calculate the cutting speed and the spindle speed for turning a 200 mm diameter Class 30 gray iron casting with a high-speed steel cutting tool. The depth of cut selected is 3.80 mm (.150 in.) and the feed selected is 0.38 mm/rev. (.015 in./rev).

$$\begin{aligned}
 V_o &= 80 \text{ fpm} && \text{(From Table 7-4)} \\
 F_f &= .91 && \text{(From Table 7-6)} \\
 F_d &= .97 && \text{(From Table 7-6)} \\
 V_o &= 80 \times .3048 = 24 \text{ m/min} \\
 V &= V_o F_f F_d = 24 \times .91 \times .97 \\
 &= 21 \text{ m/min} \\
 N &= \frac{1000 V}{\pi D} = \frac{1000 \times 21}{\pi \times 200} \\
 N &= 33 \text{ rpm}
 \end{aligned}$$

Cutting Tool Materials

Cutting tool materials must have the following properties at the high temperatures encountered in metal cutting: 1. high hardness; 2. good

Table 7-7. Chemical Composition of Frequently Used High-Speed Steels *

AISI Designation	Nominal Chemical Composition, Percent					
	C	W	Mo	Cr	V	Co
M1	0.80	1.50	8.00	4.00	1.00	
M2	0.85	6.00	5.00	4.00	2.00	
M6	0.80	4.00	5.00	4.00	1.50	12.00
M42	1.10	1.50	9.50	3.75	1.15	8.00
T1	0.70	18.00	...	4.00	1.00	
T6	0.80	20.00	...	4.50	1.50	12.00
T15	1.50	12.00	...	4.00	5.00	5.00

* C-Carbon, W-Tungsten, Mo-Molybdenum, Cr-Chromium, V-Vanadium, Co-Cobalt

abrasion resistance; and 3. resistance to chemical-metallurgical interaction with the work material. Cutting tools must also be able to withstand mechanical and thermal shock. The room temperature properties of cutting tool materials serve only as a rough guide to their ability to perform as a cutting tool; the only reliable guide is their actual performance when cutting. While high-speed steel can be broadly applied as a cutting tool material, where higher cutting speeds are required it is necessary to be able to select from other cutting tool materials the material to suit a particular job.

High-Speed Steel

High-speed steel is the name given to a group of very similar tool steels by virtue of their excellent red hardness. Red hardness is the ability of a steel to retain its hardness at a high temperature. High-speed steels will retain sufficient hardness at temperatures up to 1100 F to 1200 F which will enable them to cut other materials while at these temperatures. When cooled, the high-speed steels will return to their room temperature hardness, which is 63 to 65 HRC for cutting tools made of this material. High-speed steels are very deep hardening; this enables them to be ground to a tool shape from solid stock and to be resharpened, when required, without a loss of hardness. They can be softened by annealing and then machined into complex cutting tool shapes, such as twist drills, reamers, and milling cutters.

While the types of high-speed steels that are available are designed to meet some special requirement or to provide a special advantage, as a group they are almost a universal cutting tool material inasmuch as they are not too sensitive to the characteristic of the material being cut. Most work materials can be cut successfully with any type of high-speed steel.

It is not necessary to be overly concerned about the type of high-speed steel used. Exceptions are the high-temperature alloys and other exotic materials. The limitations of high-speed steel relative to other cutting tool materials are its lower hardness and the slower cutting speeds that must be used.

The composition of a few typical high-speed steels is given in Table 7-7. The alloying elements in high-speed steels are tungsten, molybdenum, chromium, vanadium, and carbon. There are two classes of high-speed steel. Those designated by the letter T have tungsten as their principal alloying element. To conserve tungsten, the class designated by the letter M has some of the tungsten replaced by molybdenum. The original type of high-speed steel was T1, which is still used to make cutting tools. The most widely used type for making cutting tools is M2.

The addition of cobalt to high-speed steels increases its red hardness; i.e., it will have a higher hardness and will be more wear resistant than other types at the metal cutting temperatures. Cobalt high-speed steels, such as T6 and M6, are, therefore, recommended for taking roughing cuts in abrasive materials and through abrasive surface scale. They are, however, more brittle and susceptible to cutting edge chipping. They are thus not recommended for fine finish cutting operations.

The M40 series and T15 high-speed steels (see M42 in Table 7-8) are a group of high-hardness, or so-called super-high-speed steels. They can be hardened to 70 HRC, although for cutting tool applications they are usually hardened to 67 to 68 HRC to reduce their brittleness and tendency to chip. They are designed specifically to cut high temperature alloys and other high-strength alloys. Some high-speed steels are made by the particle metallurgy process. Tools made from these steels can be about 1 HRC harder without sacrificing toughness. For high-speed steel cutting tools, 1 HRC is a significant increase in hardness. These high-speed steels are particularly useful on applications where the tool life is limited by chipping of the cutting edge.

Cemented Carbides

Cemented carbides, also called sintered carbides or just carbides, are much harder and more wear resistant than high-speed steel. They can retain their hardness at a higher temperature (1400 F and higher) than high-speed steel which enables them to cut at much faster cutting speeds. They are produced by the powdered metallurgy process. They are, however, as compact and dense as other metals. Like most very hard materials, carbides are somewhat brittle. Cemented carbides are expensive; for this reason carbide cutting tools are made in the form of small tips brazed onto

a steel shank, or small inserts that are mechanically clamped in place. Although not common, some solid cemented carbide cutting tools, such as small twist drills, are made.

Many different grades of cemented carbides are available and the importance of selecting the correct grade cannot be overemphasized. The cutting speeds for carbides given in the tables of this book are based on the assumption that the correct grade of carbide is being used. To select a carbide grade, it is best to seek the recommendation of a carbide producer. As a guideline in selecting a grade, it is first necessary to find the grades having the correct chemical and metallurgical specification for cutting the workpiece material. Then within this group, a grade having a suitable hardness is selected. Harder grades are selected for light finishing cuts, and for roughing cuts less hard but tougher grades are selected. In general, select the carbide grade having the highest hardness with sufficient strength to prevent the cutting edge from chipping.

The principal types of cemented carbides are straight tungsten carbides, crater resistant carbides, titanium carbides, and coated carbides. Straight tungsten carbide grades contain from 94 to 97 percent tungsten carbide with the remainder being cobalt. The higher cobalt bearing grades are tougher and more shock resistant; they are recommended for heavy roughing cuts. The higher tungsten bearing carbides are harder and more wear resistant; they are recommended for taking lighter, finishing cuts. As a class, straight tungsten carbides are recommended for machining gray cast iron, ferritic malleable iron, austenitic stainless steels, high temperature alloys, and plastics. They should not be used to cut plain carbon and alloy steels because these materials will form a crater on the face of the tool which will result in a very short tool life.

Plain carbon and alloy steels should be cut with a crater resistant grade of carbide. Crater resistance is imparted in a carbide by the addition of titanium carbide. In addition, tantalum carbide is usually added to these grades to prevent the cutting edge from deforming at the temperature and pressure encountered when taking a heavier cut. Crater resistant grades of carbides are used to cut all of the plain carbon and alloy steels, alloy cast irons, pearlitic malleable irons, nodular iron, Monel metal, martensitic stainless steel, ferritic stainless steel, and all of the tool steels.

Titanium carbide grades contain no tungsten carbide; they are made from titanium carbide and a small quantity of nickel or molybdenum. These grades have excellent crater and heat resistance which enables them to be used at higher cutting speeds. They will produce an excellent surface finish on the workpiece. Their use is limited by their low resistance

to shock, thermal cracking, and to oxide films and surface scale. Titanium carbides are used to take finish and semi-finishing cuts on low carbon-low alloy steel, wrought aluminum, and cast iron. They should not be used to machine abrasive materials because of their low overall resistance to abrasion.

Coated cemented carbides consist of a cemented carbide insert, called the substrate, on which a thin coating of titanium carbide (TiC), aluminum oxide (Al_2O_3), or titanium nitride has been deposited. The thickness of the coating is only .0002 to .0004 inch (0.005 to 0.010 mm), which is firmly bonded to the substrate. Usually a single coating is used although some inserts are double and triple coated. Since grinding would remove the coating, coated carbides are available only in the form of disposable inserts. The advantage of coated carbides lies in the fact that a wear resistant coating can be applied to a selected substrate to obtain a combination of favorable performance characteristics such as wear resistance, toughness, strength, and resistance to thermal shock, crater formation, and deformation. Where they can be used, coated carbides will generally operate at a higher cutting speed without a sacrifice of tool life; conversely, at the same cutting speed as an uncoated grade, the coated insert will give a longer tool life. In some cases the cutting speed can be increased 20 to 40 percent and sometimes up to 50 percent when using a coated carbide. Coated carbides have been developed to take interrupted cuts and to perform milling operations, although they are slightly less shock resistant than the tougher uncoated grades of carbides.

There are different grades of coated carbides and the correct grade must be selected for each operation. In general, coated carbides are recommended for taking medium and heavy-duty roughing cuts and high-speed finishing cuts on carbon, alloy, and tool steels, free machining steels, alloy cast iron, and the 400-500 series stainless steels. They are also recommended for medium roughing and high-speed finishing cuts on gray cast iron and the 200-300 series stainless steels. They are not recommended for the machining of cobalt, nickel, or iron base high-temperature alloys, titanium base alloys, refractory metals, free machining brass, bronze, aluminum-bronze, manganese bronze, monel, cupro-nickel, silicon-aluminum, nickel, copper, magnesium alloys, zinc base alloys, and nonmetals such as carbon, fiberglass, and plastics.

Ceramics and Cermets

Ceramic cutting tools are made by sintering fine grains of aluminum

oxide into a dense structure. Cermetes are a mixture of approximately 70 percent aluminum oxide and 30 percent titanium carbide. Ceramic cutting tools are quite brittle and sensitive to shock. They can be used to cut at extremely high cutting speeds. As an example, ceramic cutting tools have been used to cut AISI 1040 steel at 18,000 fpm (5500 m/min) on a lathe designed to take high-speed cuts. At this speed a very good tool life was obtained. Ordinary machine tools cannot, however, cut at these speeds. Ceramic inserts should be honed or ground along the cutting edge to prevent chipping. It is customary to grind a "K" land on ceramic tools, which is a narrow land .002 in. to .006 in. (0.05 mm to 0.15 mm) wide and ground at an angle of 30 degrees with respect to the face of the insert. Cermetes are stronger than ceramics and can be used for interrupted cuts. They have been used successfully in milling. To be effective, however, they should be used at very high cutting speeds. Ceramic and cermet cutting tools can be used to machine plain carbon and alloy steels, gray cast iron, malleable iron, nodular iron, and stainless steel. They should not be used to cut aluminum or aluminum alloys because the aluminum oxide in these tools will react with these metals.

Other Cutting Tool Materials

Several other cutting tool materials are firmly established by virtue of their superior performance on special classes of work.

Diamonds. Single crystal diamond cutting tools are extremely hard and very wear resistant. They produce an outstanding surface finish on all materials that they are capable of cutting because diamonds will not react with other materials at the cutting temperatures (see Fig. 9-32). They can operate at fast cutting speeds; however, the diamond may disintegrate if the temperature at the cutting edge becomes too high. For this reason they should not be used to cut steel, cast iron, or other iron-base materials. The diamond is also susceptible to chipping and interrupted cuts are not normally recommended. Diamond cutting tools are able to cut some very hard materials and are especially useful for cutting abrasive materials. Their excellent wear resistance enables them to produce a fine surface finish and a uniformly precise dimension when taking long cuts. One outstanding characteristic of these tools is their ability to produce excellent surface finishes on materials that are difficult to finish by other means, such as aluminum, magnesium, copper, and babbit. They are also used to finish brass, bronze, stainless steel, titanium, and the precious metals—gold, silver, and platinum. The finish obtained can be comparable or better than that obtained by grinding, fine honing, or lapping.

Poly-Crystalline Diamonds. These cutting tools are made in the form of

inserts from diamond powders that have been compacted and sintered together. Another type consists of a layer of poly-crystalline powders supported on a cemented carbide base. They are more resistant to chipping and can be resharpened by grinding. Otherwise, their properties and field of application are similar to single-crystal diamonds.

Cubic Boron Nitride. The trade name of this material is Borozon, by which it is better known. It is used as an abrasive in grinding wheels and is compacted to make edge-type cutting tools in the form of inserts. Borozon is very hard and can operate for long periods of time at a temperature of 2000 F. It will cut the difficult-to-machine high temperature alloys at a cutting speed that is four to six times faster than cemented carbide tools. The temperature developed at the cutting edge is high enough to produce a small ball of fire and to reduce the strength of the work material, making it easier to cut. A water base cutting fluid should be used to prevent the chips from welding onto the surface of the workpiece.

Cast Nonferrous Alloys

Cast nonferrous alloys can be used for cutting at speeds about 20 to 50 per cent higher than high-speed steels. They are not as tough, but are more wear resistant than high-speed steel. They cannot cut as fast as cemented carbides and their application is limited. The composition of these alloys is approximately 40 to 53 per cent cobalt, 30 to 33 per cent chromium, and 14 to 22 per cent tungsten.

Single-Point Cutting Tool Nomenclature

The nomenclature of a single-point cutting tool is shown in Fig. 7-1. Most cutting tools have two cutting edges; a side-cutting edge and end-cutting edge (see View A, Fig. 7-1). When doing ordinary turning the cutting is done with the side-cutting edge. The nose is that portion of the cutting edge which serves to connect the two cutting edges. It is a very critical part of the cutting edge because it produces the finished surface in turning. The flank of the tool is the surface on the side of the tool below the cutting edges. It must be relieved in order to permit the cutting edges to penetrate into the metal being cut. If the flanks below the side-cutting edge, nose, and end-cutting edge were not relieved, the tool would rub and could not penetrate into the workpiece. The face of the cutting tool provides a surface over which the chip that is formed by the cut slides. It also has a great influence in determining the angle of the shear plane on which the grains of the metal are deformed in order to form the chip. This will be described later in this chapter. The metal remaining in back of the cutting edge is the shank. It supports the cutting edge and provides a surface upon which the cutting tool can rest and be held.

The cutting tool in Fig. 7-1 is a right-hand tool. A right-hand cutting tool can be identified by having the side-cutting edge on the left when

viewed from the shank. A left-hand cutting tool has the side-cutting edge on the right side when viewed from the shank. The tool shown is also classified as a side-cutting-edge tool because it is provided with a side-cutting edge and because it is intended that most of the cutting will be done with the side-cutting edge. End-cutting-edge tools (see H, J, and K, Fig. 7-14) are intended to cut exclusively with their end-cutting edges and are made without a side-cutting edge.

When specifying a single-point cutting tool, it is necessary to specify the width and height of the shank. In addition, it is necessary to specify the radius of the nose and the six principal tool angles which are: 1. end relief angle; 2. side relief angle; 3. back rake angle; 4. side rake angle; 5. end-cutting-edge angle; and 6. side-cutting-edge angle. These angles are all shown in View B, Fig. 7-1.

Relief Angles

The side relief angle provides the relief for the side-cutting edge in order to allow it to penetrate into the workpiece so that it can form the chip. The end relief angle allows the end-cutting edge to penetrate into the workpiece. The relief angle below the nose is a blend of both the end relief and the side relief angles. Usually, both relief angles are equal although this is not always the case. The size of the relief angles has a very pronounced effect on the performance of the cutting tool; if they are too small they will cause a decided decrease in the tool life; if too large, the support below the cutting edge will be weakened to the extent that the cutting edge may break or chip. For average work, a relief angle of 10 degrees is recommended. For high-speed-steel cutting tools the relief angle should be in the range of 8 to 16 degrees. Harder materials may require a smaller relief angle while softer materials are more successfully cut when a larger relief angle is used. The relief angle for cemented-carbide cutting tools should be in the range of 5 to 12 degrees. Since carbides are harder and more brittle than high-speed steel, a better support under the cutting edge is required.

Rake Angles

The slope of the face of the cutting tool is determined by the back-rake and side-rake angles. The side-rake angle is measured perpendicular to the side-cutting edge and the back-rake angle is measured parallel to the side-cutting edge. The rake angles may be positive, as shown at B in Fig. 7-1, or may be negative, as shown at C. Note that the positive rake angles cause the face of the cutting tool to slant downward when moving away from the cutting edges, and that negative rake angles cause this surface to slant upward when moving away from the cutting edges. Positive side-rake angles are generally preferred because less cutting force is required in order to take a given size cut as compared to a tool with a negative side-rake angle.

The cutting force required to take a given cut will increase approxi-

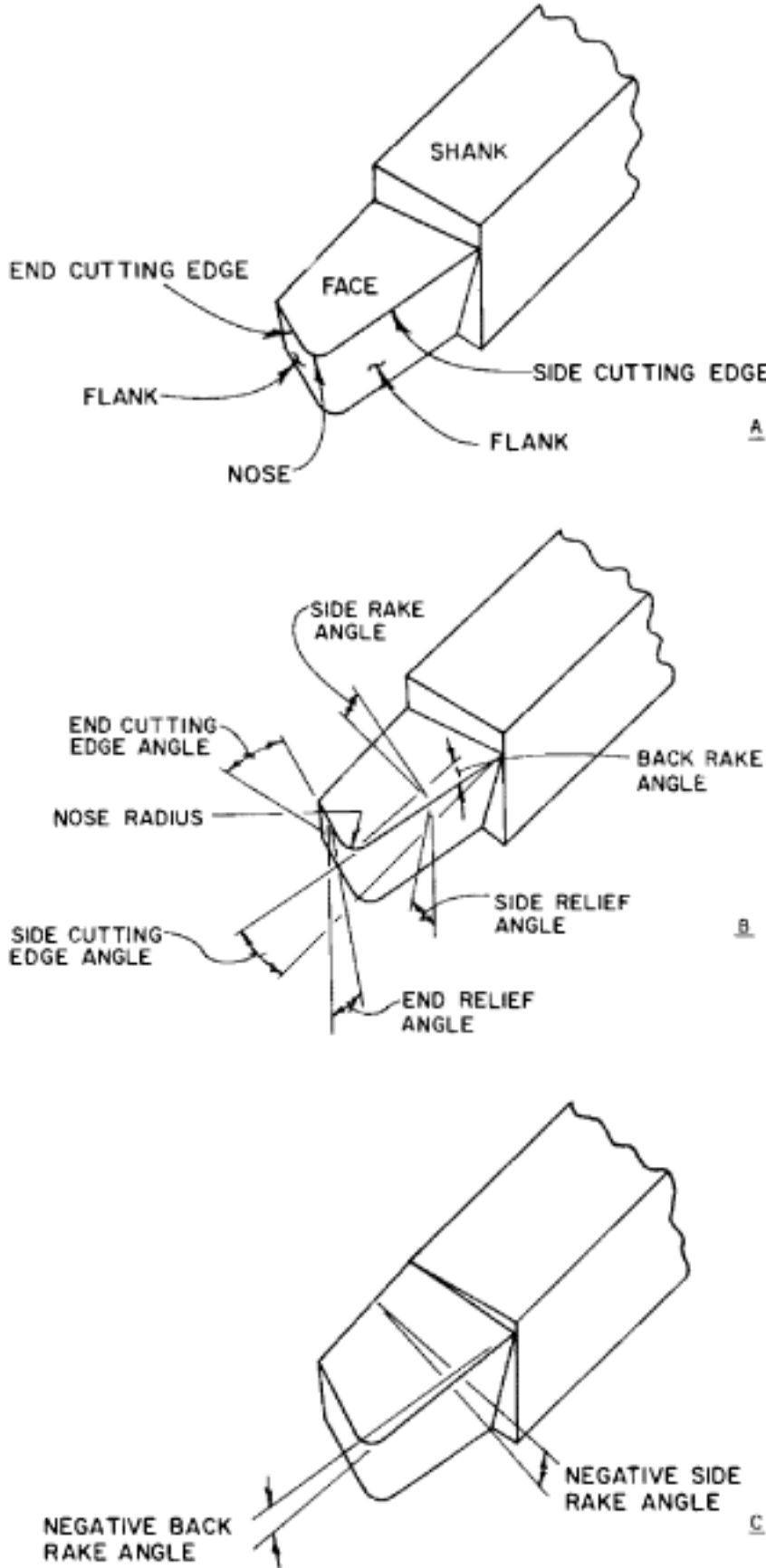


Fig. 7-1. The nomenclature of the single-point cutting tool.

mately one per cent per degree that the size of the positive rake angle is decreased, or that the size of the negative side-rake angle is increased. For example, if a tool with a 5-degree, positive side-rake angle is replaced by one with a 5-degree negative side-rake angle, the force required to take the cut will be increased approximately ten per cent. The back-rake angle does not have as much effect on the cutting force. The positive side-rake angle tool will generate less heat and will have a longer tool life than the negative side-rake angle cutting tool. The side-rake angle on most cutting tools should be made with as large a positive rake-angle as possible, up to a certain limit, for if it is made too large, the cutting edge will be weakened and will readily fail. The advantage of having a negative side-rake angle, and a negative back-rake angle, is that the cutting edge is strengthened; thereby enabling it to withstand more severe cutting loads. This is an advantage when cutting very hard and tough materials and when taking interrupted cuts. Negative rake angles are often preferred on disposable or throw-away carbide cutting tools (see Fig. 7-6) because relief angles do not have to be ground on the inserts; also, the edges on both the top and the bottom of the inserts can be used as cutting edges. This provides twice as many cutting edges as a comparable, positive rake cutting tool which has cutting edges avail-

Table 7-8. Recommended Rake Angles

Material	Hardness, HB	High-Speed Steel		Carbide	
		Back Rake Angle, deg.	Side Rake Angle, deg.	Back Rake Angle, deg.	Side Rake Angle, deg.
Plain Carbon Steel	100 to 200	5 to 10	10 to 20	0 to 5	7 to 15
	200 to 300	5 to 7	8 to 12	0 to 5	5 to 8
	300 to 400	0 to 5	5 to 10	-5 to 0	3 to 5
	400 to 500	-5 to 0	-5 to 0	-8 to 0	-6 to 0
Alloy Steel	100 to 200	5 to 10	10 to 16	0 to 5	5 to 15
	200 to 300	5 to 7	8 to 12	0 to 5	5 to 8
	300 to 400	0 to 5	5 to 10	-5 to 0	3 to 5
	400 to 500	-5 to 0	-5 to 0	-6 to 0	-6 to 0
Aluminum					
Non Heat-Treated	...	10 to 20	30 to 35	0 to 15	15 to 30
Heat Treated	...	5 to 12	15 to 20	0 to 5	8 to 15
Magnesium	...	5 to 10	10 to 20	0 to 5	10 to 20
Stainless Steel					
Ferritic	130 to 190	5 to 7	8 to 10	0 to 5	5 to 7
Austenitic	130 to 190	5 to 7	8 to 12	0 to 5	5 to 7
Martensitic	130 to 220	0 to 5	5 to 8	-5 to 5	6 to 15
Gray Cast-Iron	100 to 200	5 to 10	10 to 15	0 to 5	6 to 15
	200 to 300	5 to 7	5 to 10	0 to 5	5 to 7
	300 to 400	-5 to 0	-5 to 0	-6 to 0	-6 to 0
Malleable Iron					
Ferritic	110 to 160	5 to 15	12 to 20	0 to 10	7 to 15
Pearlitic	160 to 200	5 to 8	10 to 12	0 to 5	5 to 8
	200 to 280	0 to 5	-5 to 8	-5 to 5	-5 to 8
Brass-Free Cutting	...	-5 to 5	0 to 10	...	...
Brass					
Red, Yellow, Naval, Manganese Bronze, Nickel Silver	...	-5 to 5	-5 to 5	-5 to 5	-5 to 5
Hard Phosphor Bronze	...	-5 to 0	-6 to 3	-5 to 0	-5 to 5

able on only one face of the insert. The rake angles recommended for cutting different work materials are given in Table 7-8.

End-Cutting-Edge Angle

The end-cutting-edge angle is the amount that the end-cutting edge slopes away from the nose of the tool, so that it will clear the finished surface on the workpiece, when cutting with the side-cutting edge. The size of this angle is very important, particularly when cutting materials that tend to form a large crater on the face of the tool. This crater will then tend to enlarge toward the end-cutting edge where it will eventually break through and cause the tool to fail. This is shown at D, Fig. 7-24. When severe cratering occurs, the size of the end-cutting edge should be limited to 8 to 15 degrees; otherwise, it can be made as large as 45 degrees without an adverse effect on the performance of the tool.

Side-Cutting Edge and Lead Angles

Since the chip will flow approximately perpendicular to the side-cutting edge, the side-cutting-edge angle, thereby, influences the direction of the chip flow. It also affects the thickness of the chip and the direction of the longitudinal component of the cutting force. These, however, are more affected by the lead angle. An understanding of the relationship of the lead angle to the side-cutting-edge angle is necessary in order to understand how single-point cutting tools perform.

The lead angle is defined as the angle at which the side-cutting edge is positioned, with respect to a plane that is perpendicular to the axis of the workpiece. (See Fig. 7-2.) When the cutting tool is positioned perpendicular to the workpiece, the lead angle is equal to the side-cutting-edge angle as shown at B. In Fig. 7-3, the tool at A, has a zero degree lead angle and the tool at B, has a larger lead angle. The feed, or distance that the cutting tool advances per revolution, and the depth of cut are

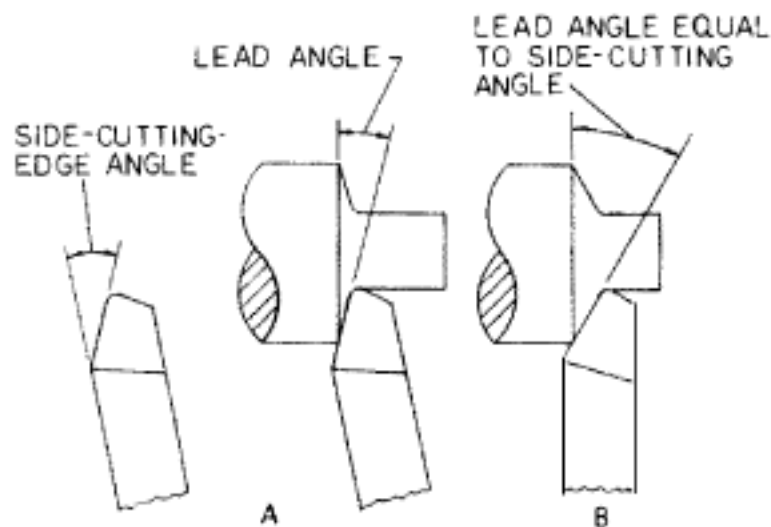


Fig. 7-2. Definition of the lead angle.

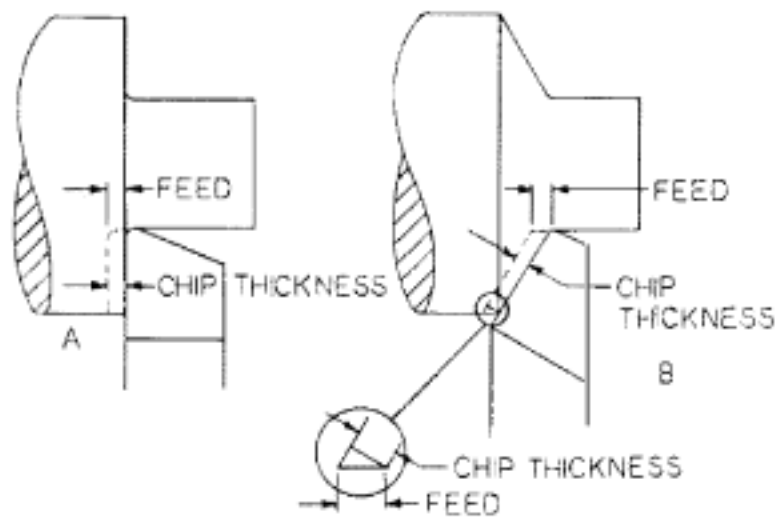


Fig. 7-3. The effect of the lead angle on the chip thickness.

equal, in both cases. The chip thickness is measured perpendicular to the side-cutting edge of the tool. At A, the chip thickness is equal to the feed; while at B, the chip thickness is less than the feed. In each case the same amount of metal is removed—in the form of a chip—for each revolution of the work; however, the chip at B, is longer and thinner. Thus, the effect of the lead angles is to produce a longer and thinner chip, which can be considered the equivalent of increasing the depth of cut and decreasing the feed rate. A study of the effect of feed on the cutting speed, in Table 7-6, will show that the thinner chip, or lower feed rate, will permit the cutting speed to be increased; or, if the cutting speed remains the same, the feed can be increased. In either case, an increase in the lead angle can bring about an increase in the production rate that can be attained. The effect of the depth of cut is relatively small, and can usually be ignored.

There is, however, an adverse effect caused by a large lead angle. This is shown in Fig. 7-4. A force is required to feed the cutting tool into the workpiece; this force is called the “longitudinal” force. It is perpendicular to the side-cutting edge. Although shown in Fig. 7-4, as reacting against the work, it reacts with an equal magnitude against the cutting tool. At A, this force is parallel to the axis of the workpiece and there is very little tendency to cause it to bend the work. At B, this force is directed so that it will cross the workpiece axis which will cause the workpiece to bend away from the tool. The elasticity in the workpiece material will cause it to spring back into its original position. When this cycle is repeated at certain frequencies a very noticeable vibration, called chatter, will set in. Chatter impairs the surface finish on the work and causes a rapid breakdown of the cutting edge as well. In all instances it must be eliminated. It can be caused by many factors, such as: loose headstock-spindle bearings, a loose fit on the cross slide, too much overhang of the cutting tool, or a very dull cutting tool. However, the most frequent cause

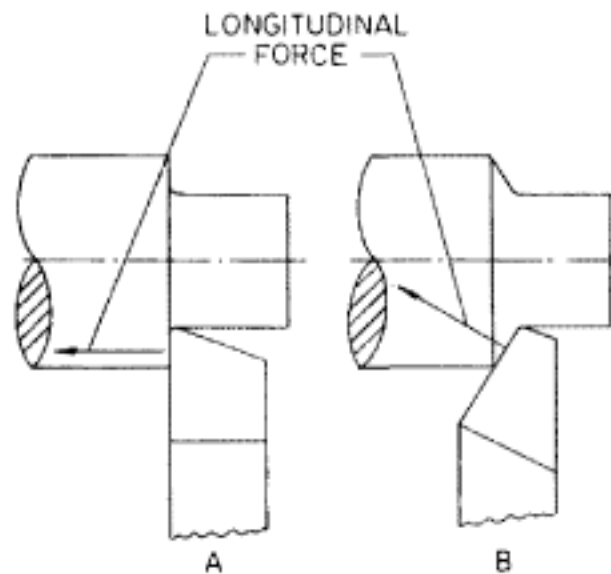


Fig. 7-4. The direction of the reaction of the longitudinal cutting force is perpendicular to the side-cutting edge. An equal and opposite force that is not shown reacts against the cutting edge.

is a lead angle that is too large; especially when turning a long and slender workpiece. The first step to take is to examine the lead angle used and to reduce it if it is too large. In a great number of instances, this will eliminate the chatter completely. When boring with a cutting tool having a large lead angle, the reaction of the longitudinal force against the cutting edge will deflect the boring bar, thus causing chatter or some difficulty in boring the hole to a desired dimension. For this reason the lead angle on boring tools should be kept small. When boring either very deep or small diameter holes, it is best to use a zero degree lead angle.

Nose Radius

The nose is an extension of the cutting edges. Its point must be formed into a radius in order to prevent the formation of a sharp threadlike groove on the surface of the work. When taking a cut, the chip thickness of the nose is equal to the chip thickness of the side-cutting edge at the point where they join. At the point where the nose is tangent to the workpiece, the chip thickness becomes zero in magnitude. Thus, the chip becomes increasingly thinner as the point of tangency between the nose and the work is approached. If the nose radius is large, there will be a large area where a very thin chip is formed. When this occurs the cutting edge may fail to penetrate the work in this region, thus causing it to rub. This is very apt to occur if the cutting edge is dull. Should the cutting edge fail to penetrate the work, chatter may be set up or the tool will get dull rapidly, in this region. Thus, we have another cause of chatter, a nose radius that is too large. Hence, reducing the size of the nose radius will reduce the tendency to chatter. When finish turning, the nose

radius must be made large enough to prevent any formation of thread-like grooves that have the form of the nose. If the size of the nose radius cannot be increased, the feed must be reduced to obtain a good surface finish. The size of the nose radius is then dependent upon the feed rate, the surface finish required on the work, and the requirement to prevent the occurrence of chatter. It should be pointed out that the life of the cutting tool can be adversely affected by a nose radius that is either too large or too small. For average conditions when turning, a nose radius of $\frac{1}{64}$ to $\frac{1}{8}$ inch is recommended, although larger nose radii can sometimes be used successfully.

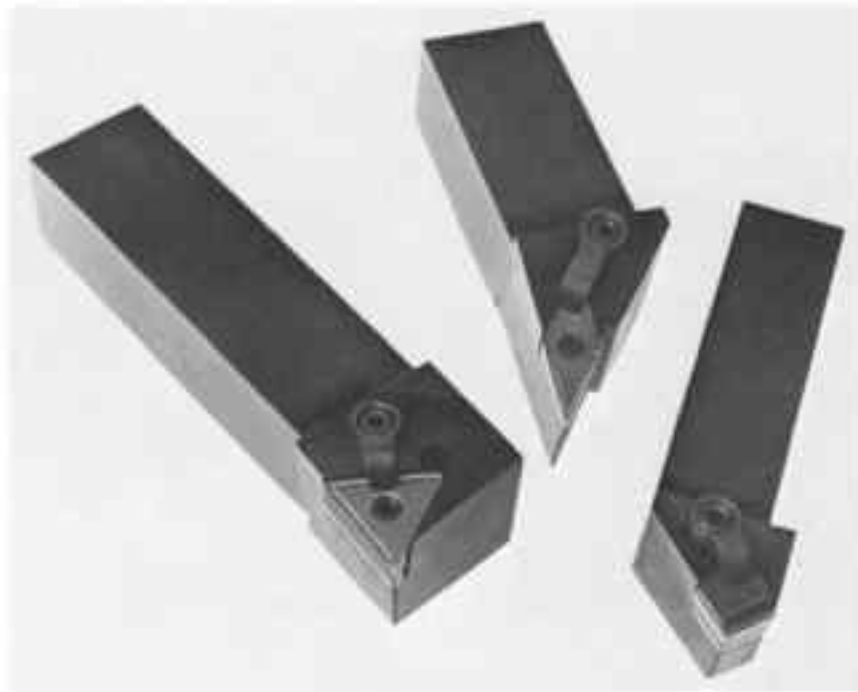
Tool Holders

In most cases the cutting tool is clamped in a tool holder; however, single-point cutting tools are sometimes clamped directly on the machine. The tool holder is made from a less expensive steel than the cutting tool. It is heat treated to increase its strength and to resist the penetration of the screws which clamp it in place. Although there is a great variety of tool holders, only representative examples of commonly used designs are illustrated here. One common but very effective tool holder for holding high-speed-steel tool bits, is shown in A, Fig. 7-5. Cemented carbide tips are sometimes brazed on steel shanks as in B, Fig. 7-5.

Most cemented carbide cutting tools are in the form of indexable inserts, which are also called disposable inserts and throw-away inserts. They are held in pockets machined in the tool holders which nest the insert. Three typical indexable insert tool holders are shown in Fig. 7-6. Each insert has several cutting edges. After a cutting edge has become dull, the insert is indexed to bring another cutting edge into the position on the tool holder where it can cut. When all the cutting edges are used up, the insert is replaced. There are positive rake and negative rake tool holders and inserts. A positive rake tool holder is shown at A, Fig. 7-7. The positive rake insert must be provided with a relief angle



Fig. 7-5: A. (Left) Toolholder for high-speed steel tool bits, B. (Right) A cemented-carbide cutting tool with a brazed-on insert.



Courtesy of Carbide Systems Dept.-General Electric Co.

Fig. 7-6. Three typical indexable insert toolholders.

on the flank. For this reason only the edges on the top face of the insert can be used as cutting edges. The relief angle on the negative rake insert, shown at B, Fig. 7-7, is provided by the inclination at which the insert is held on the tool holder. Held in this position, the flanks of the insert can be perpendicular with both faces and all of the edges on both faces can be used as cutting edges. Therefore, negative rake inserts have twice as many cutting edges as comparable positive rake inserts. Another advantage of negative rake inserts is that they are stronger than positive rake inserts and more able to withstand shock loads, such as encountered when taking interrupted cuts. The advantage of positive rake inserts is that when cutting, the cutting force is significantly less. When the cutting force must be kept as low as possible, as when cutting thin material sections, a positive rake insert should be used. Positive-negative rake inserts, view C, Fig. 7-7, are held on negative rake tool holder but have an effective positive rake angle, provided by a groove on the face of the insert. As shown in Fig. 7-8, the groove also acts as a chip breaker. Ductile materials produce an unbroken continuous chip. This chip will move off from the tool very rapidly when cutting at a high speed with cemented carbides, thereby creating a hazard to the machine operator. To eliminate this hazard, the chips must be broken into small segments by providing a chip breaker on the tool.

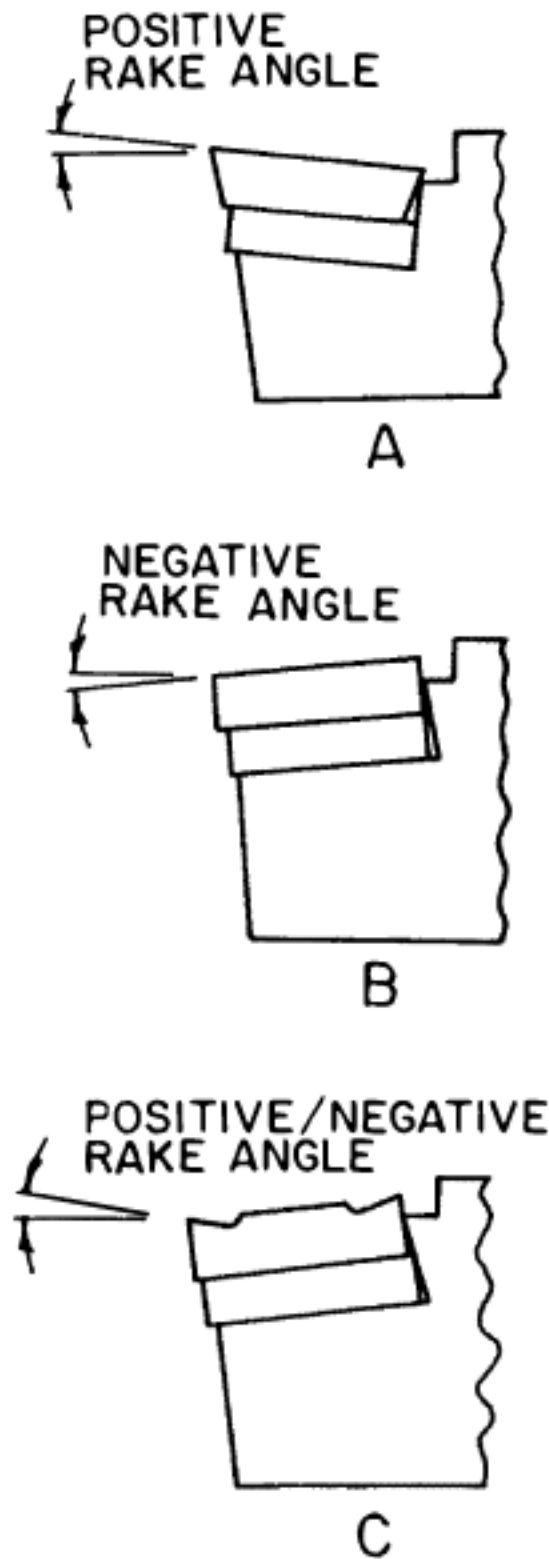
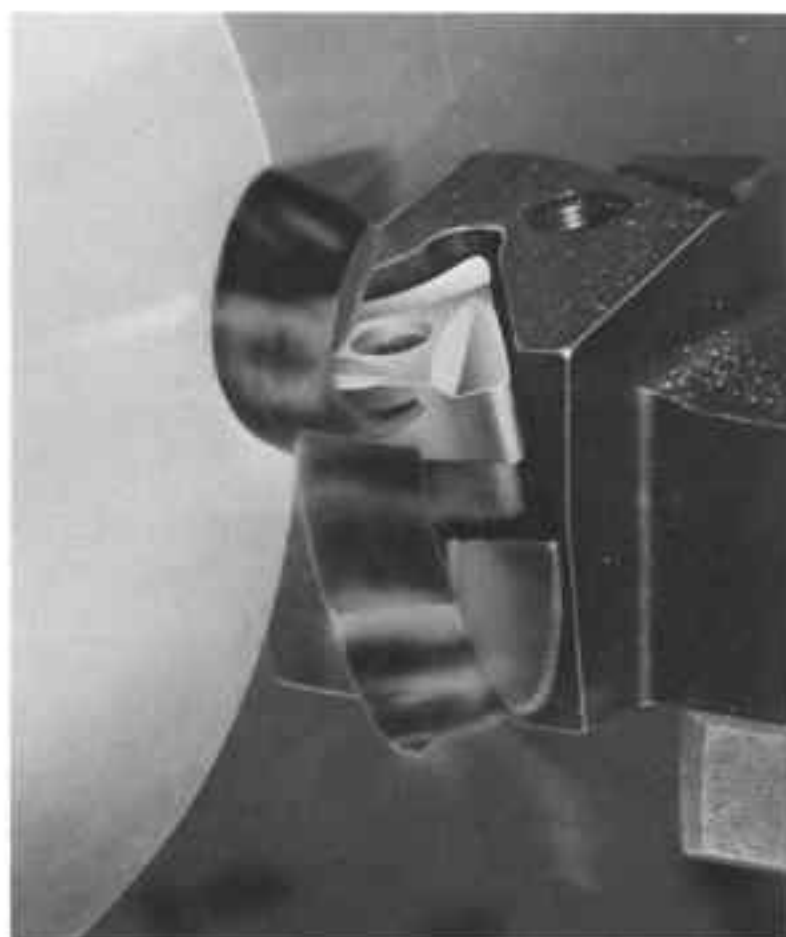


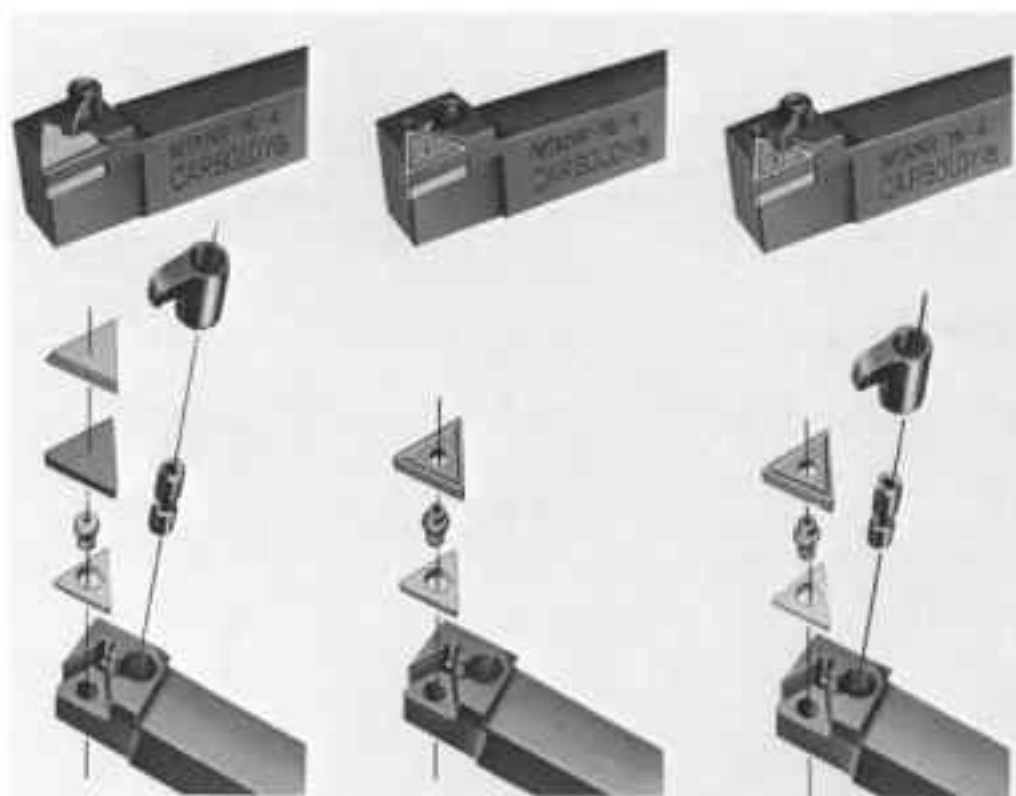
Fig. 7-7. Orientation of inserts in toolholder pockets. A. Positive rake insert; B. Negative rake insert; C. Positive/Negative rake insert.



Courtesy of the Valenite Division of the Valeron Corp.

Fig. 7-8. Positive/Negative rake insert shown cutting ductile material (steel). Groove on face provides positive rake angle and acts as chipbreaker.

The methods of clamping the inserts on the tool holder are shown in Fig. 7-9. The insert is always placed on a cemented carbide seat which rests on the bottom of the tool holder and provides a firm flat bearing surface for the insert. A pin, pin-type cam, or a screw holds the seat in place on the holder. In Fig. 7-9A the insert is clamped by a top clamp. When cutting ductile materials, a plate-type chip breaker is clamped on top of the insert. The insert at B has a chip breaker groove on the face and, therefore, does not require a separate plate-type chip breaker. It is clamped to the tool holder by a pin-type cam which fits into the hole provided in the insert. By turning the pin-type cam with an Allen wrench, the insert is clamped firmly to the seat and against the sides of the pocket. Other pin-type clamping arrangements are also used. The tool holder in Fig. 7-9 can be adapted to clamp the insert by both methods. At A it is used as a top clamp-type tool holder. At B it is a pin-type tool holder, and at C both methods are used simultaneously.



Courtesy of Carbide Systems Dept., General Electric Co.

Fig. 7-9. Methods of clamping indexable inserts on toolholder. A. Top clamp; B. Pin-type clamp; C. Using top clamp and pin-type clamp simultaneously.

Indexable inserts used to cut screw threads and shallow grooves are usually mounted on the side of the tool holder, as shown in Fig. 7-10. A tool used for cut-off and for deep grooving operations is shown in Fig. 7-11. Any of four different heads holding an equal number of different cemented carbide cut-off and deep grooving inserts can be mounted on a single shank. A group of typical boring bars used to hold cemented carbide indexable inserts are shown in Fig. 7-12. Two typical boring bars for holding high-speed steel tool bits are shown in Fig. 7-13.

Tool Shapes

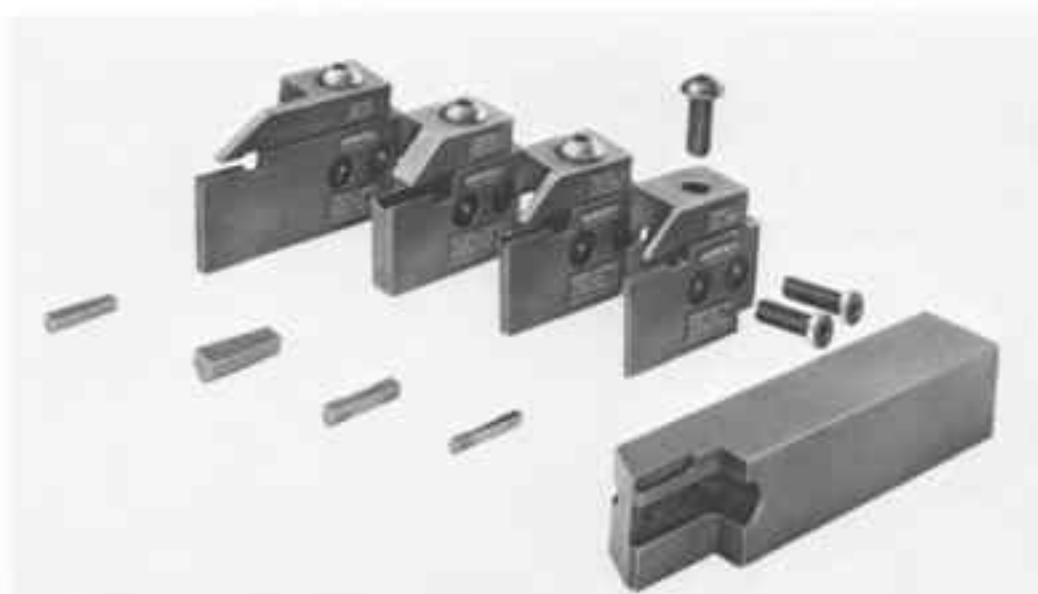
Many different tool shapes have been successfully used for turning operations but only a few of the basic shapes can be shown here. Regardless of the shape of the single-point tool, it must have the optimum tool angles that will allow it to penetrate the metal and cut efficiently. This section will deal only with the *shape* of the tool, since the tool angles have been described in a previous section.

High-speed steel single point cutting tools are usually ground to shape from solid blanks called tool bits. The basic high-speed steel cutting tool



Courtesy of Carbide Systems Dept., General Electric Co.

Fig. 7-10. Toolholder and inserts for thread cutting and for cutting shallow grooves. Insert is clamped on side of toolholder.



Courtesy of Carbide Systems Dept., General Electric Co.

Fig. 7-11. Deep grooving and cut-off tool, showing the cemented-carbide inserts and the four styles of heads that may be assembled on a single shank.



Courtesy of Carbide Systems Dept., General Electric Co.

Fig. 7-12. Typical cemented-carbide, indexable insert boring bars.

shapes and how they are applied are shown in Fig. 7-14 and are described below:

- A. *Turning Tool.* This is a basic high-speed steel single point cutting tool shape; it is a right-hand turning tool used for both rough and finish turning operations. The side cutting edge angle is zero degrees providing a zero degree lead angle when the tool is positioned as shown. It is often used to turn a cylindrical surface and to form a square shoulder at the end of the cut.
- B. *The Lead Angle Turning Tool.* The lead angle (side cutting edge angle) turning tool is one of the most frequently used single point cutting tools for roughing and for finishing operations. It may be positioned at an angle, as shown at C, to form a square shoulder at the end of a turning cut.



Courtesy of Armstrong Brothers Tool Company

Fig. 7-13. Boring bars for high-speed steel tool bits.

- C. *Turning Tool with Chip Control.* This tool is used to turn ductile materials, which form continuous chips that sometimes have a tendency to snarl. The chip control groove will curl the chip and cause it to flow away from the workpiece. There are several variations of this tool, one of which does not have the small flat on the face of the cutting edge. However, the flat is frequently used as it strengthens the cutting edge and tends to increase the tool life. It should be approximately .016 to .047 inch (0.40 to 1.2 mm) wide. The width of the flat and size of the groove depend on the material being cut and the feed rate used.
- D. *Round Nose Turning Tool.* The round nose turning tool is used extensively when turning large diameter workpieces; however, it can be used equally effectively on smaller diameter workpieces. The curved side cutting edge is, in effect, a large lead angle allowing a heavy feed rate to be used, while the large nose radius produces a very good surface finish on the workpiece. It is used for both heavy rough turning cuts and for light finish turning cuts.
- E. *Turn and Facing Tool.* This is merely a variation of the application of the turning tool at A, although for this purpose the tool is often provided with a slightly larger end cutting edge angle. Positioned as shown, it is used to turn a cylindrical surface up to a shoulder. When against the shoulder it is fed outward away from the workpiece center to take a facing cut—called back facing—in order to cut the shoulder perfectly square. If necessary, a facing

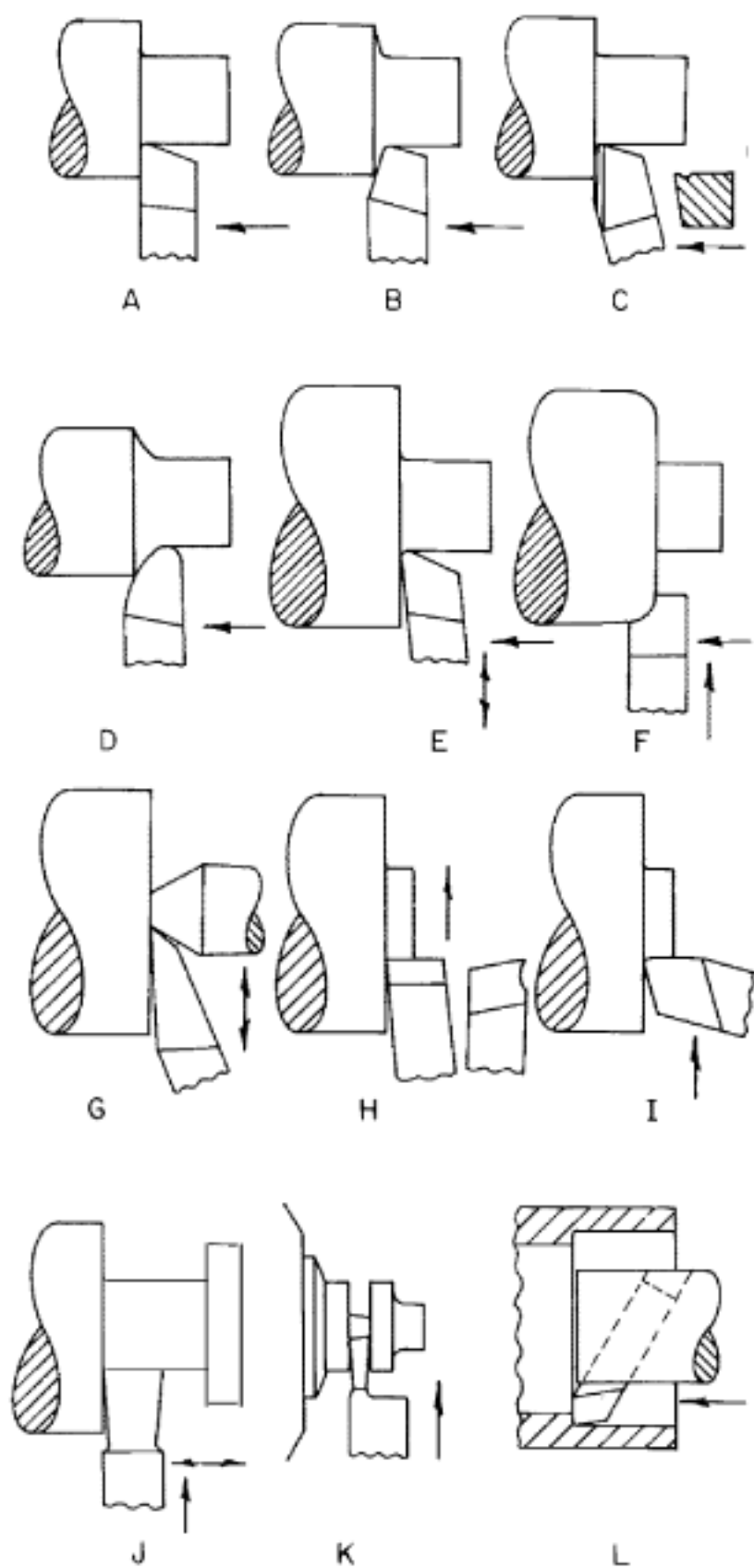


Fig. 7-14. Typical tool shapes ground on high-speed tool bits and how they are used.

cut can also be taken by feeding the tool inward toward the center of the workpiece.

- F. *Form Tool*. Form tools are used to turn a contoured shape on the workpiece, such as the radius in the illustration.
- G. *End Facing Tool*. This tool is used to face the end of the workpiece while it is mounted on a tailstock center (see also Fig. 8-19).
- H. *Facing Tool*. The facing tool shown is used to take heavy facing cuts requiring a large amount of stock removal. It has a "hook" ground on its face to provide a rake angle to the end cutting edge. Sometimes it is provided with a narrow flat on the cutting edge, similar to the flat on the tool shown at C. The side flank adjacent to the shoulder has a 5- to 8-degree relief angle.
- I. *Facing Tool*. Actually this is a left-hand turning tool which when used as shown is sometimes called a facing tool. It is used to take heavy or light facing cuts. Having a nose radius it will usually produce a better surface finish than the tool at H.
- J. *Grooving or Necking Tool*. The end cutting edge is the primary cutting edge of this tool; the sides have a slight back clearance angle to prevent their rubbing against the sides of the groove. Wide grooves are cut by taking several plunge cuts into the workpiece. The groove is then finished by taking light finishing cuts on the cylindrical surface using the carriage feed and on the sides using the cross slide feed. Narrow grooves are cut by a single plunge into the workpiece, with the width of the tool being made equal to the width of the groove in the workpiece.
- K. *Cut-Off Tool*. Cut-off tools, also called parting tools, are used to cut off the ends of stock in a lathe. Usually the portion that is cut off is a partially or completely finished workpiece.
- L. *Boring Tool*. Boring tools are used to enlarge holes. There are many different styles of boring tools of which the tool illustrated is one example. Three features of these tools must be carefully controlled: 1. the end relief angle must be large enough to provide an adequate clearance with respect to the wall of the hole; 2. the nose radius should be as small as possible; and, 3. the lead angle should be small, preferably zero degree for small diameter boring bars. The nose radius and the lead angle tend to deflect the boring bar away from the wall of the hole. They should, therefore, be kept as small as possible.

Figure 7-15 illustrates a group of typical cemented carbide indexable insert cutting tools and how they are applied. When selecting negative rake indexable insert cutting tools it is necessary to know where the relief angles are located since the tool can only cut with an edge that has a

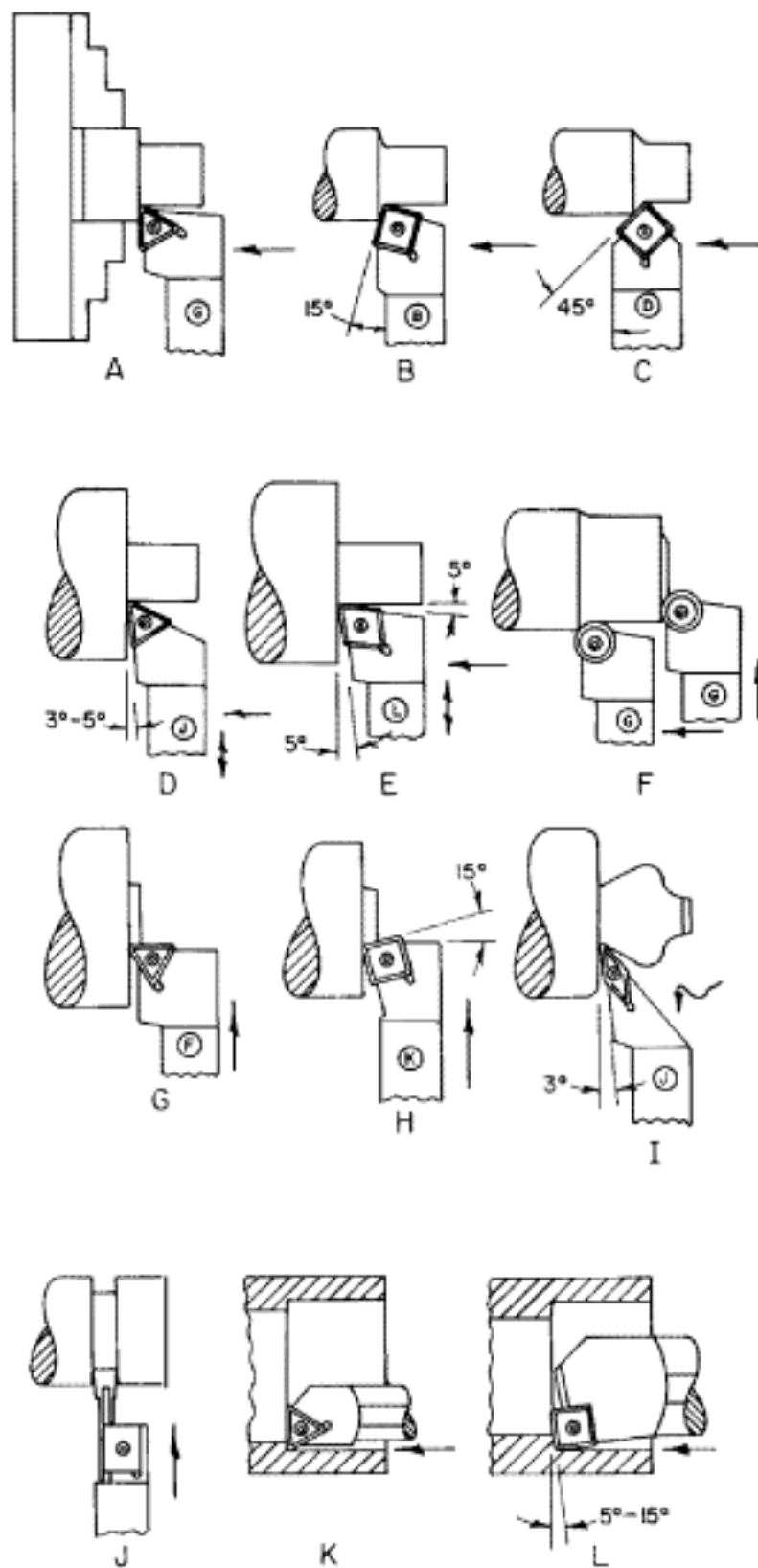


Fig. 7-15. Typical tool shapes and applications for cemented-carbide, indexable-insert cutting tools. Encircled letters are the standard toolholder designations.

relief angle. The relief angle on these inserts is provided by holding them at a compound angle on the tool holder. There will always be one or two edges of the insert that have a negative relief angle. Usually these edges are positioned on the body of the tool holder, but not always. The arrows on the illustrations show the direction in which negative rake tools will cut. Since the relief angles are ground or formed on positive rake inserts, they are not a cause of concern if the tool holder is correctly made. Standard indexable insert tool holders are designated by a letter symbol that is specified by an American National Standard (ANSI B94.26-1969). The ANSI designation of the tool holders shown in Fig. 7-15 are given by the letter enclosed within the circle. Each tool in Fig. 7-15 will be described below on the basis of the operations they are performing:

- A. *Turning.* The G style tool holder illustrated has an offset to allow the triangular insert to cut close to chuck jaws or to a large shoulder. An A style tool holder is similar but does not have the offset. These tool holders have a zero degree lead angle and can be used to form a square shoulder following a turning operation.
- B. *Turning.* The B style tool holder provides a 15-degree lead angle and allows a square insert to be used, which has more cutting edges available than the triangular insert.
- C. *Turning.* The D style tool holder provides a 45-degree lead angle which allows a heavier feed rate to be used.
- D. *Turn and Back Face.* This operation is used to produce shoulders that are precise with respect to squareness and location. The J style tool holder positions the insert to have a negative lead angle; i.e., it will cut with the nose leading. The procedure used is to turn the cylindrical surface to the shoulder, and then to backface the shoulder. If necessary, this tool can also be used to take a light facing cut by feeding inward toward the center of the workpiece.
- E. *Turn and Backface.* In all respects the operation of this tool is identical to the tool shown at D. The insert is diamond shaped having an acute angle of 80 degrees. When held on the L style tool holder it has only two cutting edges per face available. However, this holder holds the insert very firmly and allows a heavier turning cut to be taken than is usually possible with the tool at D. The size of the backfacing cut is dependent primarily on the negative lead angle, although on tough materials the L style holder will hold the insert more firmly when backfacing.
- F. *Turn and Facing.* Round inserts provide more usable cutting edge, or indexes, per face before they are used up than the other insert shapes. Furthermore, the large radius produces a good surface finish. When used on the G style tool holder, the round inserts can be used to take medium and light turning and facing cuts.

- G. *Facing*. The F style tool holder and the triangular insert are used to take light and heavy facing cuts.
- H. *Facing*. The K style tool holder utilizes a square insert to take light and heavy facing cuts.
- I. *Contour Turning*. The diamond insert shown is specifically designed for contour turning, although other inserts are also used for this purpose depending on the shape of the contour. There are two types of diamond inserts designed for this purpose: one has a 35-degree acute angle while the other has a 55-degree acute angle. A J style tool holder is shown; other style tool holders are available which hold the insert in a different position.
- J. *Cut-Off and Grooving*. Cut-off and grooving tool holders do not have an ANSI designation. Many different designs of carbide cut-off and grooving tools are available (see Figs. 7-10 and 7-11).
- K. *Boring*. A large selection of indexable insert boring (Fig. 7-12) bars are available. When the lead angle is zero degrees a triangular insert is used, as shown.
- L. *Boring*. Boring bars having a lead angle can utilize square inserts. The lead angle should be kept small on smaller diameter bars.

Some very useful single point tool shapes are shown in Fig. 7-16. The tool at A, is a combination "rough and finish" turning tool. A small flat is ground on the end-cutting edge, which is used to take a very light finishing or scraping cut, while the side-cutting edge is used to take a heavier cut. The width of the flat should be from $\frac{1}{16}$ to $\frac{3}{16}$ inch. The side rake angle of this tool must be zero degrees. It may be given a positive back rake angle up to 10 degrees. A similar tool, having a 45-degree, side-cutting-edge angle is shown at B. Either tool can be made from high-speed steel or from cemented carbide. The flat of each of these tools can be used to take a light finishing cut when desired, although, primarily, they are designed to take combined roughing and finishing cuts.

A broad nose finishing tool is shown at C, Fig. 7-16. When properly ground, honed, and applied it will produce an excellent surface finish. The high-speed steel, broad-nose tool has a deep hook ground on the face, which, in effect, gives it a very large rake angle. A small radius is ground or honed at each end of the cutting edge. The cutting edge should be honed to a sharp edge and a fine finish. Cemented carbide broad-nose tools should have a back rake angle from -6 to $+15$ degrees, depending on the hardness of the material being cut. The tool must be positioned with the end-cutting edge parallel to the work, as shown in the illustration, in order to contact the cylindrical work surface along the tool's entire length. It is used with a very light depth of cut which varies from approximately .001 to .005 inch. The recommended feed rate is .015 to .750 inch per revolution. Sometimes, even heavier feeds are used. The lighter feeds generally result in a better surface finish, while the heavier feeds result in higher production rates. The cutting speed used with broad-

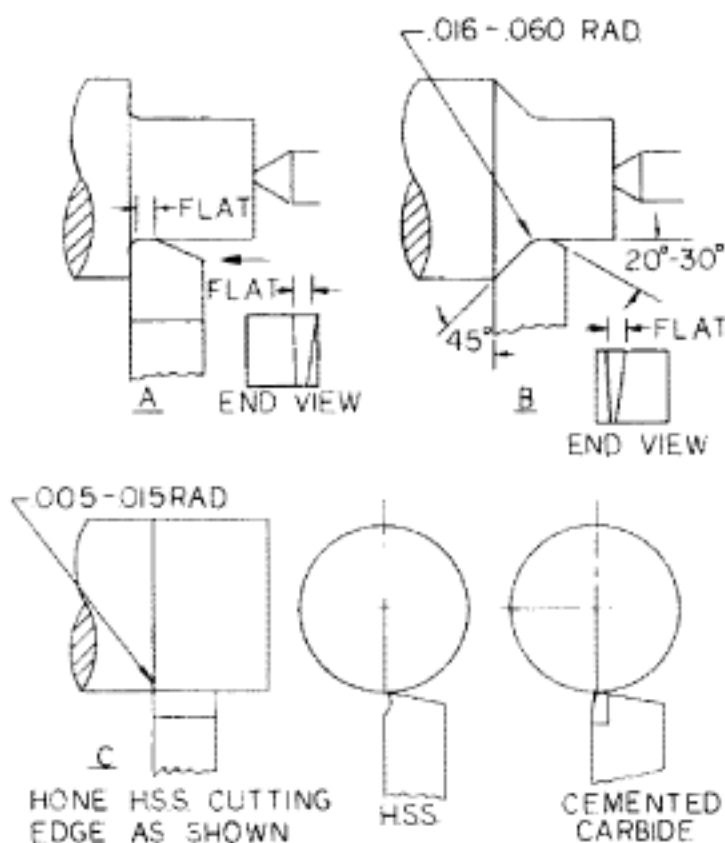


Fig. 7-16. A. Combination rough- and finish-turning tool. B. Combination rough- and finish-turning tool with lead angle. C. Broadnose finishing tool.

nose tools should generally be lower than recommended for turning tools. The high feed rates possible with this tool cause it to traverse the length of the turned surface much more rapidly than the turning tools; therefore, there is less wear on the cutting edge which results in greater accuracy when turning long, cylindrical surfaces. The short cutting time, and the greater accuracy obtained, make the broad-nose tool very useful when turning long, cylindrical or tapered surfaces.

Grinding Single-Point Cutting Tools

Single-point cutting tools are generally ground by hand on pedestal or bench type grinders. Some of these are specifically designed to grind single-point cutting tools and have a table on which the tools can be placed while they are held against the grinding wheel by hand. The grinding wheel used on these machines is designed for grinding on the side of the wheel in order to produce a true plane surface on the workpiece. The table can be set at an angle, with respect to the side of the wheel, thereby enabling the desired angle to be ground on the tool. Provisions are usually made to have a flow of coolant available in order to prevent the tool from overheating while grinding. Most single-point, cemented-carbide cutting tools are ground on this type of grinder using a diamond-

impregnated grinding wheel. High-speed-steel cutting tools can also be ground on these machines using an aluminum oxide grinding wheel.

Most high-speed-steel cutting tools, however, are ground on a simple pedestal or bench grinder. The tool bit is held by hand against the peripheral face of the wheel, at the angle required. This method is called off-hand grinding and it requires a considerable amount of skill. This skill can be acquired by analyzing what is to be accomplished and by practice.

A grinding wheel suitable for grinding high-speed steel should be mounted on the pedestal grinder. For this type of grinding a typical wheel specification is A-36-O-5-V for rough grinding, and A-60-M-5-V for finish grinding. (See Chapter 14, Volume 2, for the meaning of grinding-wheel specifications.) The grinding wheel should be dressed so that it is straight and that it runs true. Occasionally, it must be redressed in order to keep it sharp and true. A dull grinding wheel will cut more slowly and can cause the tool to become overheated. It is desirable to use a grinder equipped to supply a flow of coolant over the face of the wheel. If a coolant is not available on the grinder, the tool should be dipped in water, occasionally, to cool it. It should never be allowed to overheat to the extent of becoming discolored as this can cause metallurgical damage which will reduce the life of the tool. The most effective way of preventing the tool from becoming overheated is to control the pressure used to hold the tool against the face of the grinding wheel. A very heavy pressure can cause the tool to overheat even if a coolant is used on the machine and the grinding wheel is sharp.

A general-purpose, high-speed steel turning tool is shown in Fig. 7-17. Starting with a high-speed steel tool bit, the procedure for grinding this tool is shown in Fig. 7-18. When grinding, the tool bit should be held with the cutting edge facing upward so that it can be seen. Held in this manner, the grinding wheel will "run on" to the cutting edge instead of running off from it. This prevents the possibility of chipping and reduces the tendency to form a fine almost microscopic feather on the cutting edge. All of the angles on the tool should first be rough ground almost up to the cutting edges; then the angles are finish ground and the cutting edges sharpened. The nose radius is ground last. It is ground by holding the tool at the correct angle with a light pressure and then swinging the tool bit slowly back and forth between the side and end flanks.

Chip Formation

The three basic types of chips formed in metal cutting are the continuous chip, the discontinuous chip, and the segmental chip, Fig. 7-19. When the cutting tool first makes contact with the metal, it compresses the metal ahead of the cutting edge. As the tool advances, the metal ahead of the cutting edge is stressed to the point where it will shear internally, causing the grains of the metal to deform and to flow plastically along a plane called the shear plane, or roughly perpendicular to the

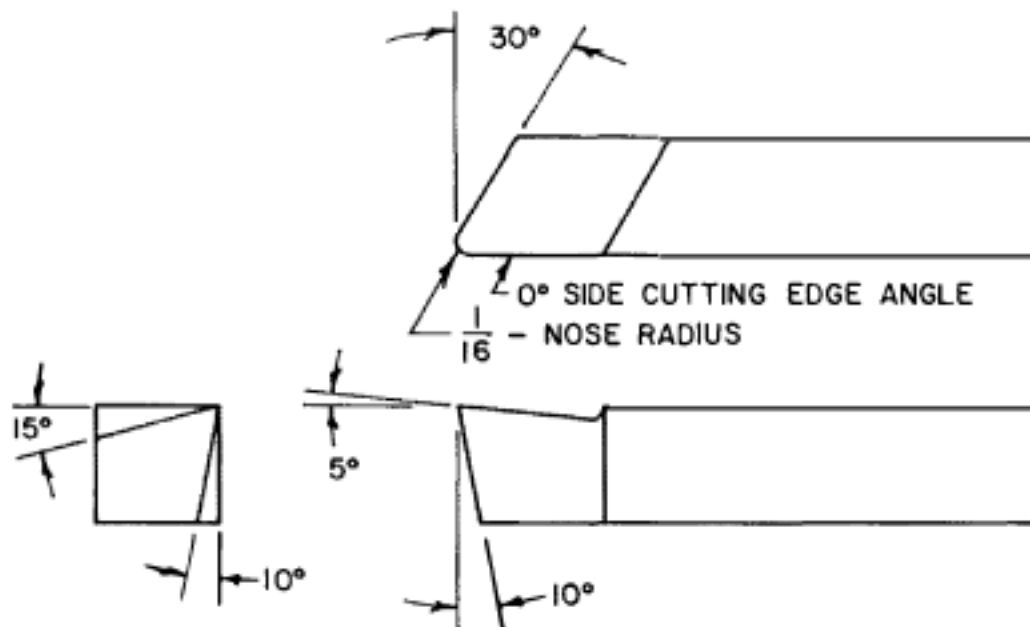


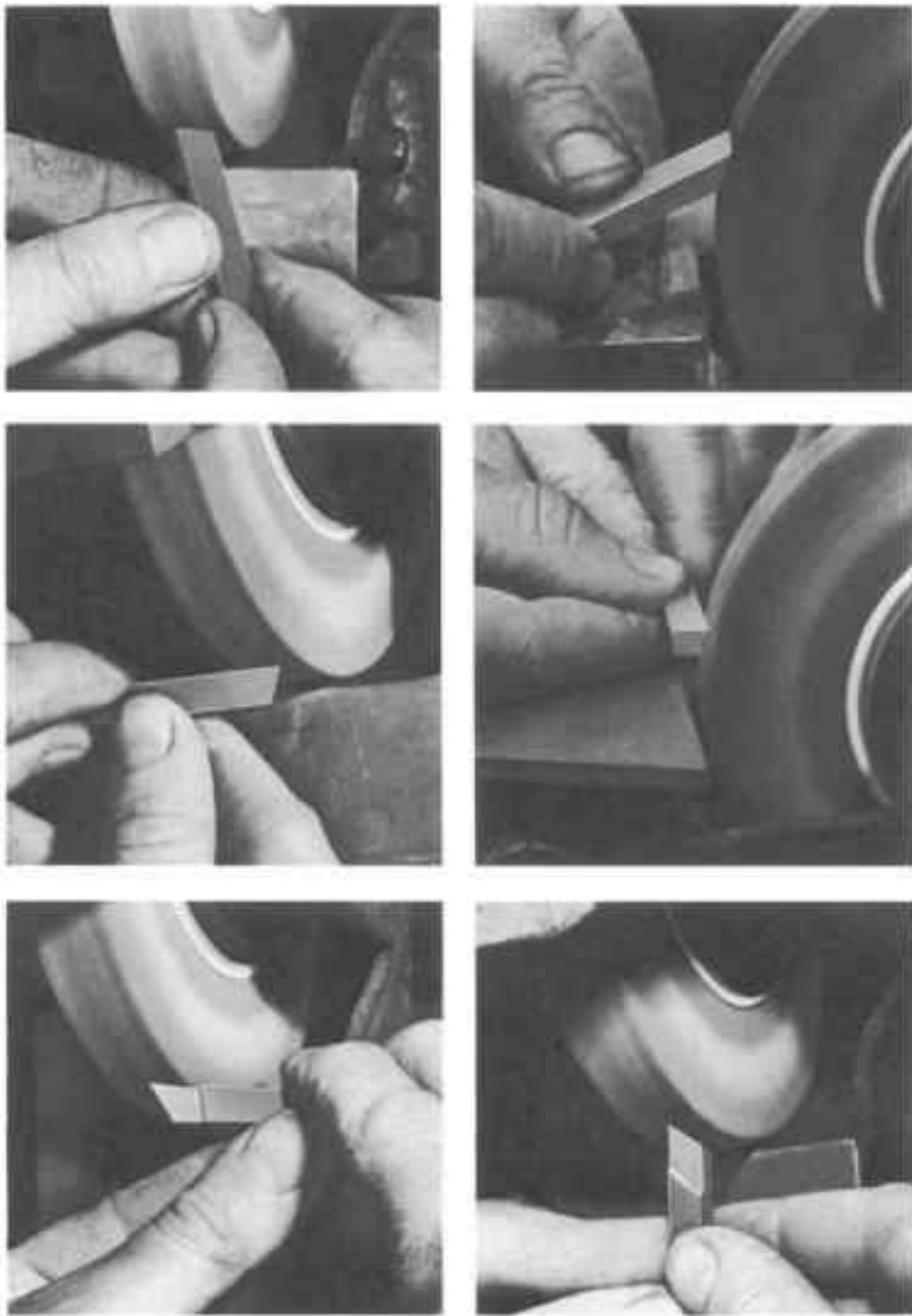
Fig. 7-17. General-purpose, high-speed steel turning tool. The sequence of grinding this tool is shown in Fig. 7-18.

face of the tool.* When the type of metal that is being cut is ductile, such as steel, the chip will come off in a continuous ribbon, as at A, Fig. 7-19. The internal shearing action of this type of chip, called a continuous chip, causes plastic flow but it does not cause the metal to separate, or fracture; the chip escapes by flowing over the face of the tool. The movement of the chip over the tool face, under high pressure, does further work to that portion of the chip in contact with the face. This surface, located on the underside of the chip, is work hardened and will have a shiny appearance.

If the metal being cut is brittle, such as gray cast-iron, the stresses on the shear plane will eventually cause the metal to fracture because it is unable to flow plastically as can a more ductile material. This causes the chip to break up into particles as shown at B, Fig. 7-19. Such a chip is called a discontinuous chip. The formation of a discontinuous chip is illustrated in Fig. 7-20.

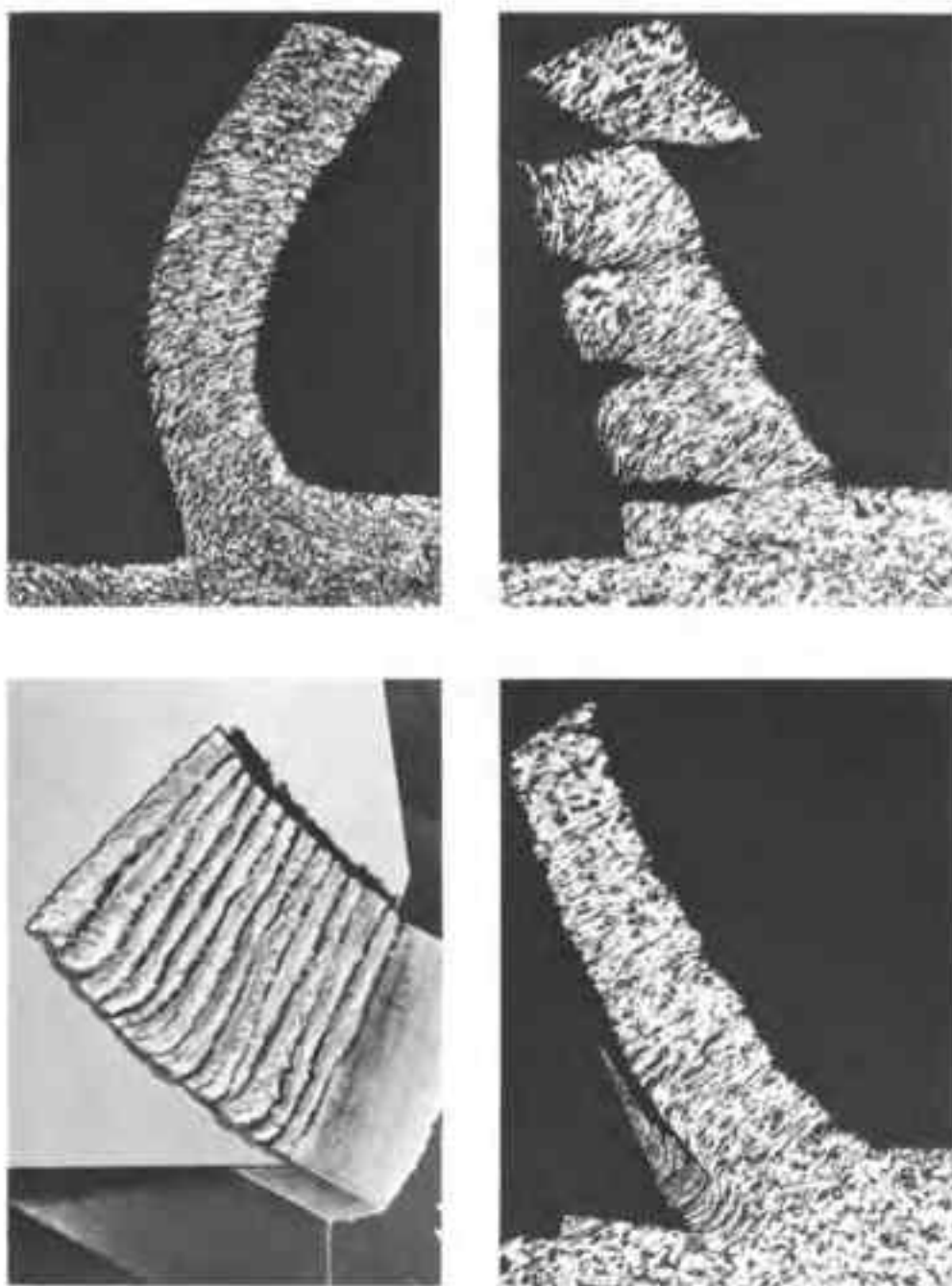
A segmental chip is frequently encountered when machining ductile materials. It is shown at C, Fig. 7-19. When a heavy feed is used, the stresses on the shear plane cause the metal in this plane to eventually fracture internally. This may be further abetted either by low frequency and high amplitude vibrations of the cutting tool or the work, or both. The internal fractures will spread to the outside surfaces; however, they are also quickly welded together again by the action of the pressures existing in this region. The result is a continuous chip that has a file-

* The shear plane is not a true plane inasmuch as it usually is slightly curved. Furthermore, it is not a surface, but a region within the metal along which internal shear occurs.



* Fig. 7-18. Sequence for grinding general-purpose turning tool. (Upper left) Top view of simultaneously grinding the end cutting edge and end relief angle. (Upper right) Side view of same operation. (Center left) Top view of simultaneously grinding side-relief angle and side cutting edge angle (0°). (Center right) Side view of same operation. (Lower left) Simultaneously grinding the side-rake and back-rake angles. (Lower right) Grinding the nose radius.

* The views in Fig. 7-18 were obtained through the kind cooperation of Mr. Harold D. Manning, Chairman, Industrial Arts Dept., and Mr. Paul Vogel, student, at the James E. Sperry Senior High School, Henrietta, N.Y.

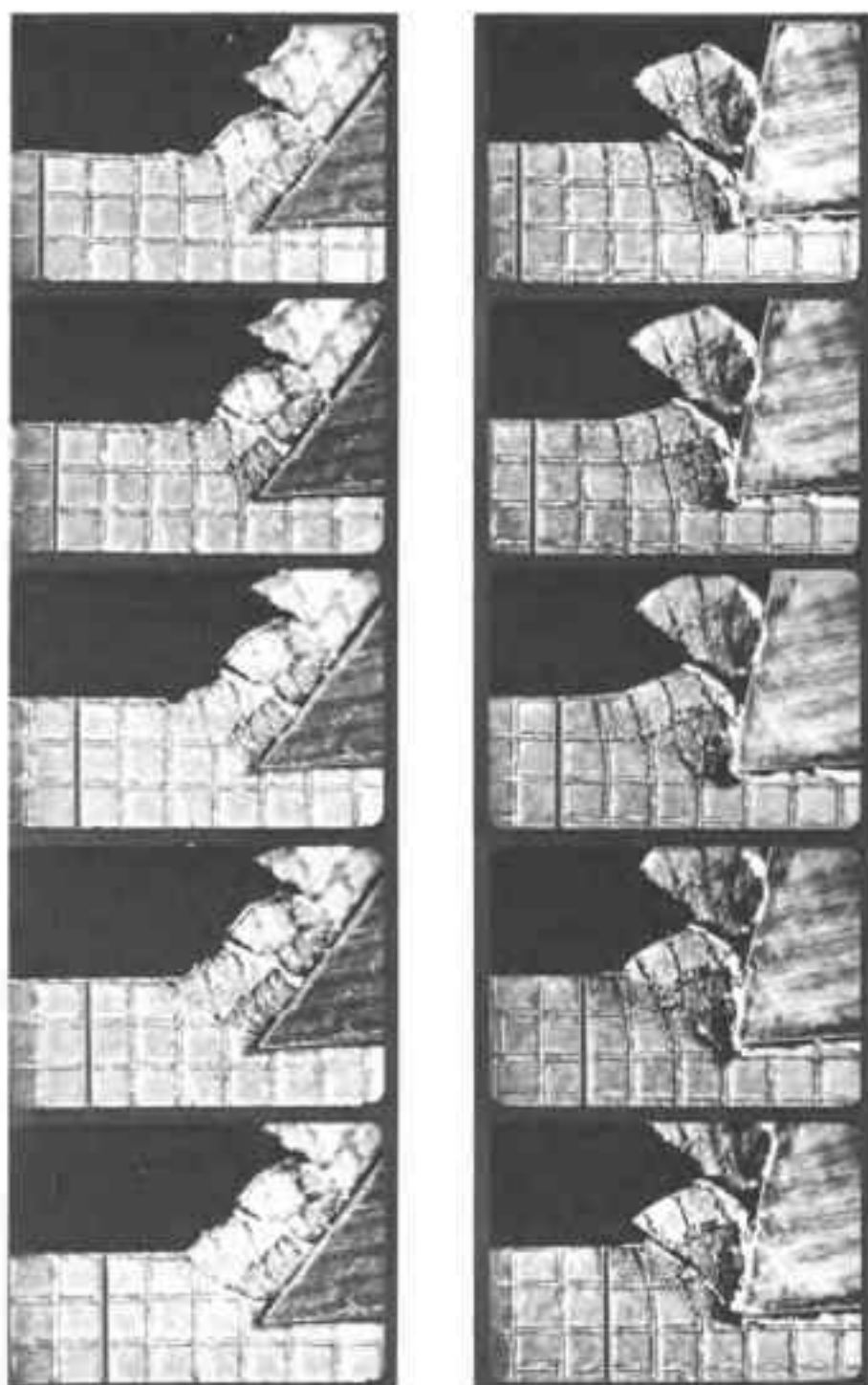


Courtesy of Cincinnati Milacron

Fig. 7-19. A (Upper left) Continuous chip. B (Upper right) Discontinuous chip. C (Lower left) Segmental chip. D (Lower right) Continuous chip with built-up edge.

tooth appearance on one side. The side of the chip that passes over the tool face is more completely welded together, and is burnished by the face so that it has a smooth appearance. The degree and depth of the segmentation of a chip can vary greatly because of the feed, the cutting speed, and the nature of the metal being cut.

Another phenomenon associated with the formation of the chip is the



Courtesy of Cincinnati Milacron

Fig. 7-20. The formation of a discontinuous chip. (Left) Cutting tool with large rake angle. (Right) Cutting tool with small rake angle.

built up edge shown at D, Fig. 7-19. This generally occurs when cutting ductile materials at relatively low (approximately, 150 to 200 fpm, or less) cutting speeds. The built-up edge is a highly work-hardened layer

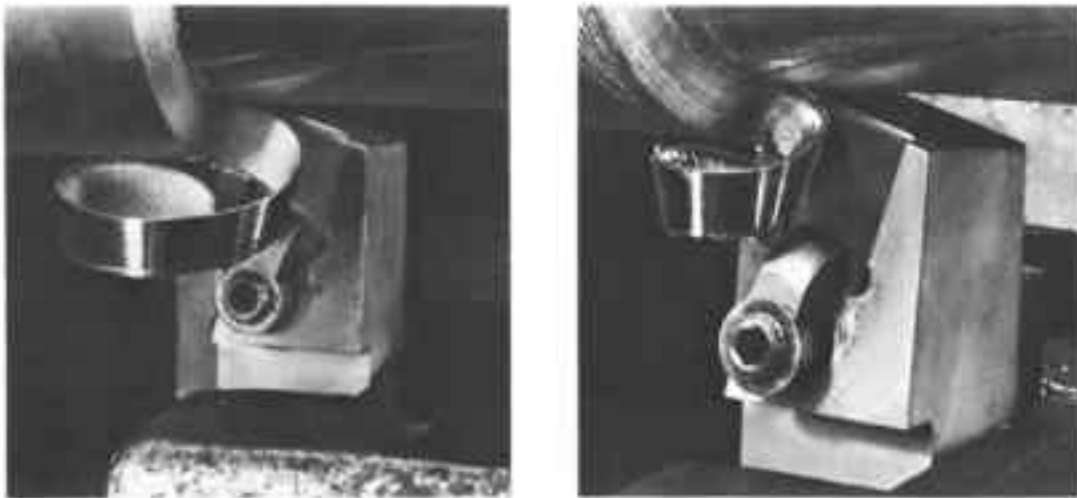
of the work material that has been welded onto the cutting edge. It starts out as a small layer that gradually builds up in size by having more work material continuously added to it by pressure welding. Eventually, the built-up edge enlarges to the point where it becomes unstable and it breaks off. When this happens, it may pass off with the chip or it may wedge between the work and the cutting edge to score the work surface. It can also pluck out small particles of the cutting tool material as it breaks off, resulting in a deterioration, or wear, of the cutting edge. When the built-up edge has broken off, a new built-up edge starts to form and this cycle is repeated. In some cases, the size of the built-up edge becomes quite large and protrudes beyond the cutting edge to score the surface of the work while it is still attached to the tool.

The effect of the built-up edge is to cause the machined surface of the work to be rough. *The built-up edge is the principal cause of surface roughness on the machined surface.* Efforts to achieve a good surface finish should be directed toward eliminating the formation of the built-up edge. It will *not* form when the cutting speed exceeds a certain critical rate which can be easily reached with cemented-carbide cutting tools. When these tools are operated above this speed, an excellent surface finish is obtained without difficulty. As high-speed steel must usually operate below this cutting speed, in order to obtain a good surface finish, a cutting fluid is used with high-speed steel tools, to reduce or to eliminate the built-up edge. Using a large rake angle, a light feed rate, and having a smooth surface finish on the cutting edge will also help. Since the formation of the built-up edge requires a metallurgical interaction between the work material and the cutting tool material, using a cutting tool material that does not react with the work material will also reduce or eliminate the built-up edge. Industrial diamonds fall into this category; they will produce excellent surface finishes; however, their use is restricted to very light finishing cuts.

Chip Control

Chip control is not a problem when cutting metals that form a discontinuous chip. It is only troublesome when machining metals that form a continuous or a segmental chip with high-speed-steel cutting tools. Chip control, on the other hand, is essential when forming continuous or segmental chips with cemented-carbide cutting tools. The high cutting speeds used cause such a rapid formation of the chip that both the machine operator and the equipment are endangered.

The action of a plate type chip breaker is shown in Fig. 7-21. The plate deflects the chip, causing it to curl, and to strike against the flank of the tool holder or the chip breaker. When the chip strikes the tool holder it breaks off into short lengths which are curled and have an appearance like a figure 9. The size of the broken off chips depends on the feed rate and the design of the chip breaker. Chip breakers may be ground into high speed steel and into cemented-carbide tools to cause the chip



Courtesy of Kennametal Inc.

Fig. 7-21.

to curl and break off as described. The three basic types of chip breakers are shown in Fig. 7-22. Many cemented carbide inserts for turning ductile materials that form a continuous chip have pressed-in chip breakers or a chip control geometry that is formed in the insert when it is made. A chip breaker plate is not used with these inserts.

Tool Wear

All metal cutting tools begin to wear at the instant they start to cut. Tool wear is unavoidable. However, the rate of tool wear must be kept within acceptable limits to obtain a reasonable tool life. Fundamentally, tool wear is caused by some very complex mechanical, chemical, and metallurgical interactions between the cutting tool material and the work material. The visible manifestation of these interactions provides useful information in improving the performance of the cutting tool. They will now be described.

Flank Wear. This type of tool wear pattern is normal and will always occur. It cannot be avoided, but the rate at which it progresses must be controlled to obtain a reasonable tool life. Flank wear is illustrated in Fig. 7-24A. The worn surface on the flank just below the cutting edge is called the flank wear land. It is a very uniform band that can readily be seen with a low powered magnifying glass or magnifying mirror. The width of the wear land can be measured with a low powered toolmakers microscope, and the progress of the wear as the tool cuts can be shown on a graph, as in Fig. 7-23. When a new tool starts to cut it will wear very rapidly during the first minute or two of its operation; this is called the

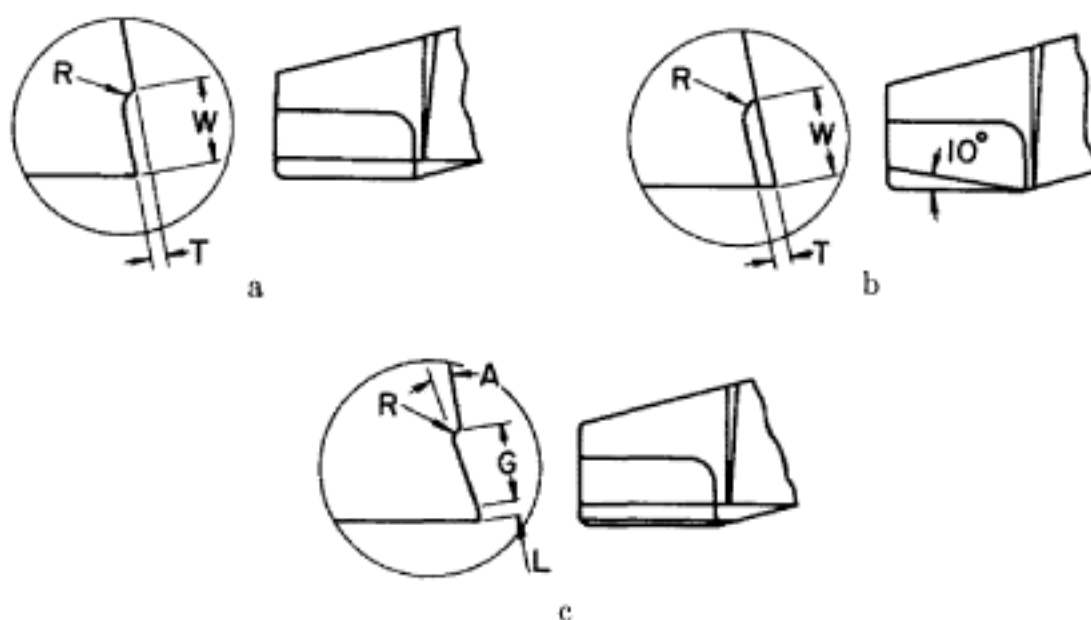


Fig. 7-22. Basic types of chip breakers: a. Parallel, b. Angular, c. Groove type.

initial or break-in flank wear. While the rate of wear during this period is rapid, under normal operating conditions, the amount of wear is not usually great. After the break-in wear has occurred, there is a long period of time during which the rate of wear is very uniform; i.e., the width of the wear land increases at a uniform rate. The rate at which the wear land increases is primarily affected by the cutting speed, then on the feed rate, and least of all on the depth of cut. As the tool cuts, the wear land rubs against the work material and the resulting friction creates heat. When the wear land is small, the heat generated by the rubbing action is much less than the heat generated in forming the chip; however, when the wear land reaches a certain width (approximately .035 to .050 in., or 0.89 to 1.27 mm), the heat generated by the flank wear land increases rather rapidly causing the flank wear rate to also increase rapidly. Then the combined effect of the heat generated by the flank wear land and by the formation of the chip causes the temperature of the cutting edge to rise to the point where the tool material becomes soft and unable to cut. At this point a catastrophic failure of the cutting edge occurs. Before catastrophic failure occurs, the rubbing action of the wide wear land will burnish the workpiece, giving it a shiny appearance.

The cutting tool should be replaced or resharpened before catastrophic failure takes place. Experience has shown that this should be done when the width of the wear land reaches .010 to .020 in. (0.25 to 0.50 mm) for finishing cuts, .030 in. (0.76 mm) for most medium and heavy roughing cuts, and .040 in. (1.02 mm) for extremely heavy roughing cuts, such as

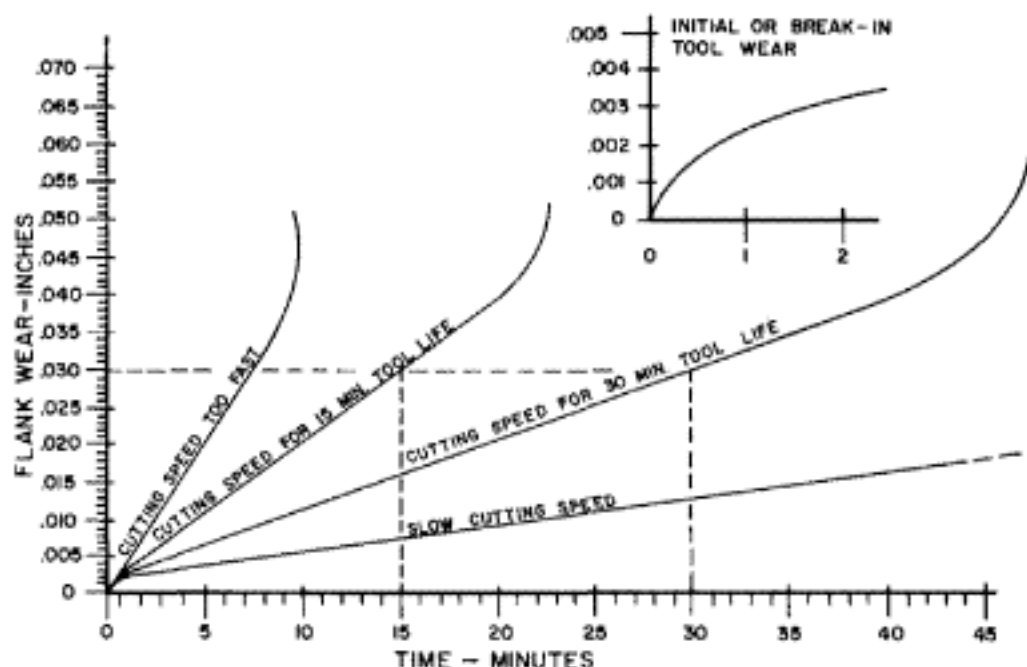


Fig. 7-23. Typical graph showing how the flank of a metal cutting tool wears. Higher cutting speeds cause flank wear rate to increase; slower cutting speeds reduce flank wear rate. Insert shows initial, or break-in, wear occurring during first minute or two of cutting time.

encountered when turning steel mill rolls. The length of time that the tool will cut before reaching the prescribed amount of flank wear defines its tool life. Therefore, decreasing the rate of flank wear will increase the tool life, as shown in Fig. 7-23. Flank wear rate and tool life depend on the combined effect of the work material, the work material hardness and heat treatment, the cutting tool material, the cutting speed, feed, and depth of cut. For a given combination of work material and tool material, the most effective way to increase the tool life is to decrease the cutting speed. Next in effectiveness is to reduce the feed rate and least effective is to reduce the depth of cut. A coolant-type cutting fluid will usually reduce the flank wear rate and increase the tool life. The flank wear rate is very sensitive to the grade of cemented carbide used in the cutting tool; tool life can be increased significantly by selecting the best grade of carbide for the particular application.

Built-Up Edge. The built-up edge (Fig. 7-24B) is the principal cause of roughness on the machined surface. It also damages the cutting edge and contributes to tool wear. How the built-up edge forms, grows, and breaks off has been described in this chapter under "Chip Formation" and will not be repeated here. Very often, when the built-up edge is broken off, it will pluck out a piece of the cutting edge to which it has been welded,

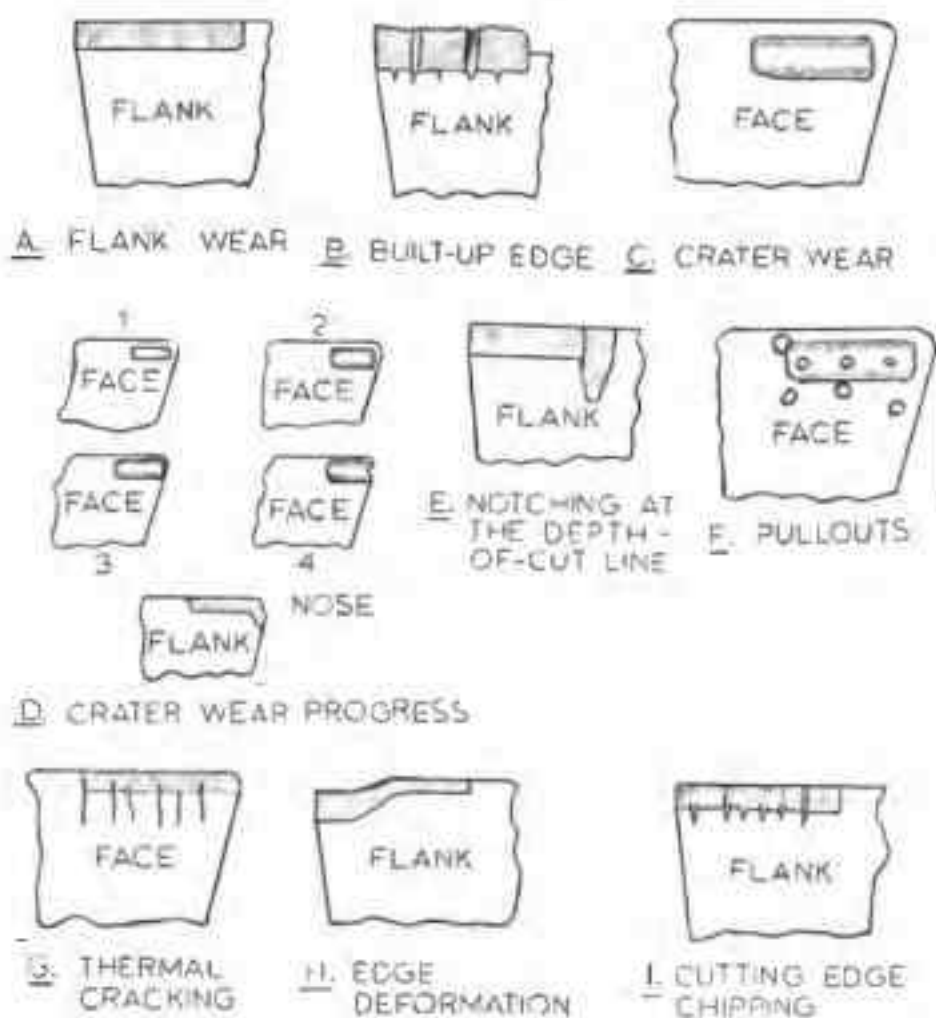


Fig. 7-24. Types of tool wear.

thereby forming a notch on the flank that may extend below the flank wear line giving it a jagged appearance. The voids thus created are quickly filled with work material which is welded to the tool material and work hardened by the pressure of the cut. Surprisingly, controlled tests have shown that on high speed steel tools, the built-up edge may actually prolong the life of the cutting tool by "repairing" the damaged areas with work material that is hardened and welded in place. These tests show that the tool life may be somewhat erratic; however, the tool life is not shortened and may be increased. Studies have shown that the tool *may* cut for periods of time without any increase in the width of the wear land. Apparently the built-up edge protects the flank of the tool from exposure to wear.

The built-up edge is not a problem when rough turning with high speed steel cutting tools, except when turning certain very soft materials which form an excessively large built-up edge. Since it does cause roughness on

the machined surface, on finishing cuts steps should be taken to reduce or eliminate the built-up edge. For high speed steel cutting tools, the most effective means of combating the built-up edge is to use a mineral oil base cutting fluid having good anti-weld properties. The rake angle on the tool should be made as large as possible and the surfaces on the face and flank should be honed smooth. Unless required to add strength, the cutting edge should not be honed round or to a bevel edge; it should remain sharp. When cutting with cemented carbide cutting tools, the most effective and easiest way to eliminate the built-up edge is to increase the cutting speed above the speed where the built-up edge will not form. For many materials this cutting speed is approximately 150 to 250 fpm (45 to 75 m/min.).

Crater Wear. The combined effect of heat, pressure, and abrasion caused by the movement of the chip, and some very complex chemical and metallurgical interactions between the work material and the tool material cause the surface on the face of the cutting tool near the cutting edge to erode. When this erosion becomes severe, a crater will form on the face near the cutting edge. The severity of the erosion is dependent on the work material, tool material, and the cutting speed. A typical crater is shown in Fig. 7-24C, and the progress of crater wear is shown at D. The formation of the crater starts a short distance behind the cutting edges. As the tool continues to cut, the crater will enlarge, in depth as well as spreading out. In time the crater approaches the cutting edges; usually it will approach the end cutting edge first. When this occurs the end cutting edge will be weakened and will break off in the region of the crater. Since this is close to the nose of the tool, the nose will fail shortly afterwards. With the nose having failed, the failure of the cutting tool is complete. This type of tool failure is called crater breakthrough. When machining titanium and its alloys the crater forms very close to the side cutting edge; thus, the crater will usually break through at the side cutting edge to cause tool failure.

When straight tungsten carbides cut steel, a crater will form very rapidly and crater breakthrough on the end cutting edge will occur in a short time. Steel should always be cut with a crater resistant carbide or with a coated carbide. Straight titanium carbide cutting tools have excellent crater resistance, but they tend to be brittle. Crater breakthrough can be retarded by using a tool having a small end cutting edge angle. If all else fails to prevent excessive cratering, the cutting speed should be reduced.

Notching at the Depth of Cut Line. This form of tool wear is characterized by a deep notch that forms at the depth of cut line, as shown in Fig. 7-24E. The notch is caused by cutting through hard and abrasive scale, or by a layer of highly work hardened material. Scale that is formed on the surface of castings, forgings, hot rolled stock, and heat

treated parts is composed of a variety of oxides that are very abrasive and hard. Some materials, such as the austenitic stainless steels, work harden very easily when they are machined or cold worked, forming a layer of very hard material. High temperature alloys also cause excessive notching at the depth of cut line.

Notching at the depth of cut line is very difficult to prevent when it occurs. The most common corrective action is to use a tool having a large lead angle or, when this cannot be done, to reduce the feed rate. Honing a bevel on the cutting edge at the depth of cut line may be helpful, and using a harder grade of carbide may also help.

Pullouts. Pullouts (Fig. 7-24F) are very small craters that form on the face of the tool when machining materials that will readily weld to the tool material, such as high temperature alloys. Usually a larger crater will also form and the pullouts occur around the crater. Pullouts will also occur within the crater; however, they will usually be filled with work material that is welded in place. Pullouts are caused by particles of the chip that are welded firmly to the face of the tool. As the chip continues to sweep by, it will literally knock off the welded-on particle, which will pluck out a small particle of the tool material as it breaks off, leaving a small crater behind. While pullouts must be considered a form of tool wear, they are generally not harmful to tool life, except when they enlarge the crater or when they occur near the cutting edge. When they occur near the depth of cut line, pullouts can contribute to the formation of the notch in this region. Usually no steps are taken to prevent the formation of pullouts. The remedial measures are to decrease the cutting speed and to use a suitable cutting fluid.

Thermal Cracking. Thermal cracks (Fig. 7-24G) occur when the cutting zone of the tool is rapidly heated and cooled while the adjacent tool material is not unduly heated. This causes large differences in temperature, or temperature gradients, between adjacent layers of material on the tool, which in turn cause the cracks to form. Since the large temperature differences do not occur at the cutting edge but a short distance away, the cracks will first form a short distance away from the cutting edge. As the tool continues to cut, the cracks will propagate toward the cutting edge until it is finally reached. When the cracks reach the cutting edge, the tool will ultimately, but not necessarily immediately, fail. Thermal cracking generally occurs when machining at high speeds with cemented carbide tools. They may be caused by an interrupted cut or when milling, when the tool is alternately heated and cooled. A coolant that reaches the hot cutting tool only intermittently can cause thermal cracks to form. In this case a constant coolant supply to the tool must be obtained, or no coolant should be used. Thermal cracking may be pre-

vented by a large reduction in the cutting speed. First, however, a tougher and more shock resistant grade of carbide should be tried.

Another major cause of thermal cracking is improper tool grinding. Any cutting tool can be severely damaged in this manner before it is used. Grinding (thermal) cracks are caused by pressing the tool hard against the grinding wheel, or by using too much infeed. A dull grinding wheel may also cause thermal cracks; always use a sharp grinding wheel when grinding a tool. Cemented carbides may be damaged by using an incorrect type of abrasive; a diamond abrasive is recommended for grinding carbide tools.

Edge Deformation. When taking a very heavy cut with a cemented carbide cutting tool, the carbide in the cutting zone may be heated to the point where it softens and deforms under the cutting load. As shown in Fig. 7-24H, this often occurs in the region of the nose. Deformation will result in the formation of a large wear land after a very short cutting time. Tantalum carbide is added to the tungsten carbide to resist deformation. Whenever deformation is apt to occur, a grade of carbide containing tantalum carbide should be used. Sometimes a coolant may be used to cool the tool, which helps to prevent deformation. If these preventative measures will not help, the feed and depth of cut must be reduced.

Cutting Edge Chipping. This type of tool failure is not always easy to recognize because the chipping may be minute, although more massive chipping may also occur. Often a ragged wear land will be an indication of chipping, as shown in Fig. 7-24I. Chipping is the mechanical failure of the cutting edge caused by overloading or by inherent lack of strength. (Chipping can also be caused by the built-up edge, but this type of chipping will not be considered here.) Anything that will strengthen the cutting edge will help to overcome chipping. Honing the cutting edge will strengthen it and often prevent chipping. This is easy to do on the job, or pre-honed inserts can be purchased when this type of tooling is used. Honing too much off the cutting edge should be avoided as this will reduce the tool life. One method is to hone a flat land on the cutting edge that is .002 to .005 in. (0.05 to 0.13 mm) wide and at an angle of 30 degrees with respect to the face. Other steps to strengthen the cutting edge may also be taken. A softer but tougher grade of carbide, or a high-speed steel that does not contain cobalt may be used as a cutting tool material. The relief angle on the tool should be as small as possible. Using a negative rake angle is often the most effective way to stop chipping. Chipping may not be the fault of the cutting tool. It may be caused by machine tool vibration, chatter, or a lack of rigidity of the setup.

Oxidation: Tool wear is caused by abrasion and by complex chemical-

metallurgical reactions occurring at the tool-chip interface and on the flank of the tool. It is also caused by oxidation of the heated tool material which is exposed to the atmosphere. Oxidation of the tool material may be a significant cause of tool wear whenever the tool material is heated to a very high temperature, such as when operating at very high cutting speeds.

Metal Cutting Fluids

The functions of metal cutting fluids are: 1. to cool the cutting tool as a primary function, and also to cool the workpiece; 2. to reduce the heat generated by friction through lubrication; 3. to provide anti-weld properties in order to prevent welding of the chip to the tool; and 4. to wash away the chips. There are three basic types of cutting fluids: 1. water soluble oils; 2. straight cutting oils; and 3. chemical fluids.

Water soluble oils, also called emulsified oils, are mixed with water in a water-to-oil ratio ranging from 5 to 1 to as much as 50 to 1. The mixture has an appearance ranging from milky to cloudy, or almost clear. Water is the best coolant, but has many disadvantages as a cutting fluid, the primary disadvantage being that it causes rust. Soluble oils prevent rust, are excellent coolants, and have a measure of lubricity. Additives may be added to soluble oils to enhance their performance and to prevent bacterial growth. Soluble oils can be used for almost all light, medium, and heavy-duty machining operations. They can be used over the entire range of cutting speeds, including the higher speeds where cemented carbide tools are used. They are not recommended for certain severe cutting operations requiring greater lubricity and anti-weld properties. Soluble oils are also recommended for almost all precision grinding operations.

Straight cutting oils are made from mineral oil that is usually compounded with additives to increase their effectiveness. Uncompounded mineral oils are sometimes used on very light duty machining operations on aluminum, magnesium, brass, and sulfurized free cutting steels. However, in most instances these materials are machined with mineral oils having fatty oil additives. Fatty oil additives add lubricity to the cutting oil. They are largely composed of lard oil, castor oil, sperm oil, and fatty esters. Active additives will react mildly with metal to form an oxide film. Sulfur in the base mineral oil is usually not active, but when added to the oil it is an active additive. Chlorine is an active additive. Active sulfur will not become active until a higher temperature is reached, while chlorine becomes active at a lower temperature. Singulary or combined, active sulfur and chlorine provide anti-weld properties to the cutting oil to prevent the chip from welding and to prevent or reduce the formation of the built-up edge, thereby improving the finish on the machined sur-

face. Straight oils should be used when the cutting speed is moderate or low and on work where soluble oils do not give a satisfactory performance. They are particularly recommended on jobs where the chip is crowded, such as thread cutting. Straight oils are used on some precision grinding operations; however, provisions must be made to prevent the vapors from exploding.

Chemical fluids, also called synthetic cutting fluids, are proprietary chemical compounds to which water is added. There are two kinds; those having a wetting agent and those that do not. Those that do not have a wetting agent are restricted to rough grinding operations, where they tend to keep the grinding wheel open and free cutting. Those having a wetting agent have some lubricity to allow the machine tool slides and other machine parts to function smoothly. Chemical fluids are excellent coolants and are recommended for machining operations that are in the moderate to higher cutting speed range.

In general, cutting fluids should be applied in a continuous stream that floods the cutting tool. An intermittent supply on a hot cutting tool will likely cause thermal cracks to form on the tool. It is not, however, poor practice to apply a cutting fluid manually with a brush or a squirt can whenever the cutting speed is moderate or low. There are many occasions in the toolroom or in a job shop where this practice is the most practical method of application. At these speeds the tool is not likely to be damaged and the application of the cutting fluid is beneficial.

The only real measure of a cutting fluid is how it performs on the job. When selecting a cutting fluid, the cutting speed should be considered. In general, when the cutting speed exceeds about 75 to 100 fpm (25 to 30 m/min) a water soluble oil or a chemical fluid should be used. Below this speed range, first consider a water soluble oil; then if the job is too difficult for a water soluble oil, select a straight cutting oil.

Estimating the Power Required for Turning

A knowledge of the power required to perform machining operations is useful when planning new machining operations, optimizing existing machining operations, and developing specifications for new machine tools to be acquired. The measure of power in customary inch units is the horsepower. In SI metric units the kilowatt is the measure of both mechanical and electrical power. The power required to cut a material by machining depends on the rate at which the material is being removed and by an experimentally determined power constant, K_p , which is also called the unit horsepower, unit power, or specific power consumption. The power constant is equal to the horsepower required to cut a material at a rate of one cubic inch per minute. In SI metric units, the power constant is equal to the power in kilowatts required to cut a material at a rate of one cubic

centimeter per second, or 1000 cubic millimeters per second ($1 \text{ cm}^3 = 1000 \text{ mm}^3$). Different values of the power constant are required for inch and for metric units, which are related as follows: to obtain the SI metric power constant, multiply the inch power constant by 2.73; to obtain the inch power constant, divide the SI metric power constant by 2.73. Values of the power constant are provided in Tables 7-9, 7-10, and 7-11. These values may also be used for planing, shaping, and broaching, provided they are used with the appropriate formulas.

The value of the power constant is essentially unaffected by the cutting speed, the depth of cut, and the cutting tool material, when operating at normal cutting conditions. There are, however, factors that do influence the value of the power constant and thereby the power required to machine the material. These factors include the hardness and the microstructure of the work material, the feed rate, the rake angle of the cutting tool, and the condition of the cutting edge of the tool, whether it is sharp or dull. Values of different hardness levels for each material are given in the power constant tables, where this information is available. Feed factors for the power constant are given in Table 7-12. In most instances the effect of the rake angle can be disregarded. Most of the values in the power constant tables are for a positive 14 degree rake angle. Only when the deviation from this angle is large is it necessary to make a correction. Using a rake angle that is more positive will cause the power required to decrease approximately one percent per degree. Using a rake angle that is more negative will cause the power required to increase, again approximately one percent per degree. Many disposable insert cutting tools have a pressed-in-chip breaker or other cutting edge modifications, which have the effect of increasing the rake angle and reducing the power required to cut a material. The extent of this effect cannot be predicted without a test of each design. The same uncertainty exists with most ground-in chip breaker designs.

The values in the power constant tables are for sharp cutting tools. However, all cutting tools wear as they are being used and are expected to continue to cut with wear present on the cutting edge. More power is required to cut as the cutting edge becomes worn. Factors to provide for tool wear are given in Table 7-13. The extra-heavy duty turning category in this table occurs only on certain operations, such as roll turning, where the flank wear below the cutting edge is allowed to reach as much as .040 inch (1.00 mm) before the tool is replaced. On most rough turning operations the cutting tool is replaced when the flank wear has reached .030 inch (0.76 mm). Most cutting fluids will reduce the power required, when operating in the lower range of cutting speeds. It is not possible, however, to provide specific power reduction data for cutting fluids in general because each cutting fluid exhibits its own characteristics.

The machine tool transmits the power from the driving motor to the cutting tool, where the power is then used to cut the workpiece. How efficiently this is done is measured by the machine tool efficiency factor E . Average values of this factor are provided in Table 7-14.

The formulas for estimating the power required for turning are given

Table 7-9. Power Constants, K_p , for Wrought Steels, Using Sharp Cutting Tools

Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units
Plain Carbon Steels	80-100	.63	1.72
	100-120	.66	1.80
	120-140	.69	1.88
	140-160	.74	2.02
	160-180	.78	2.13
	180-200	.82	2.24
All Plain Carbon Steels	200-220	.85	2.32
	220-240	.89	2.43
	240-260	.92	2.51
	260-280	.95	2.59
	280-300	1.00	2.73
	300-320	1.03	2.81
	320-340	1.06	2.89
	340-360	1.14	3.11
Free Machining Steels	100-120	.41	1.12
AISI 1108, 1109, 1110, 1115, 1116, 1117, 1118, 1119, 1120, 1125, 1126, 1132	120-140	.42	1.15
	140-160	.44	1.20
	160-180	.48	1.31
	180-200	.50	1.36
AISI 1137, 1138, 1139, 1140, 1141, 1144, 1145, 1146, 1148, 1151	180-200	.51	1.39
	200-220	.55	1.50
	220-240	.57	1.56
	240-260	.62	1.69
Alloy Steels	140-160	.62	1.69
	160-180	.65	1.77
	180-200	.69	1.88
	200-220	.72	1.97
AISI 4023, 4024, 4027, 4028, 4032, 4037, 4042, 4047, 4137, 4140, 4142, 4145, 4147, 4150, 4340, 4640, 4815, 4817, 4820, 5130, 5132, 5135, 5140, 5145, 5150, 6118, 6150, 8637, 8640, 8642, 8645, 8650, 8740	220-240	.76	2.07
	240-260	.80	2.18
	260-280	.84	2.29
	280-300	.87	2.38
	300-320	.91	2.48
	320-340	.96	2.62
	340-360	1.00	2.73
	140-160	.56	1.53
	160-180	.59	1.61
	180-200	.62	1.69
	200-220	.65	1.77
AISI 4130, 4320, 4615, 4620, 4626, 5120, 8615, 8617, 8620, 8622, 8625, 8630, 8720	220-240	.70	1.91
	240-260	.74	2.02
	260-280	.77	2.10
	280-300	.80	2.18
	300-320	.83	2.27
	320-340	.89	2.43
	160-180	.79	2.16
	180-200	.83	2.27
	200-220	.87	2.38
AISI 1330, 1335, 1340, E52100	220-240	.91	2.48
	240-260	.95	2.59
	260-280	1.00	2.73

Table 7-10. Power Constants, K_p , for Ferrous Cast Metals, Using Sharp Cutting Tools

Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units	Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units
Gray Cast Iron	100-120	.28	0.76	Malleable Iron	150-175	.42	1.15
	120-140	.35	0.96		175-200	.57	1.56
	140-160	.38	1.04		200-250	.82	2.24
	160-180	.52	1.42		250-300	1.18	3.22
	180-200	.60	1.64	Cast Steel	150-175	.62	1.69
	200-220	.71	1.91		175-200	.78	2.13
Alloy Cast Iron	220-240	.91	2.48		200-250	.86	2.35
	150-175	.30	0.82	...	...	...	...
	175-200	.63	1.72	...	...	...	...
	200-250	.92	2.51				

Table 7-11. Power Constant, K_p , for High Temperature Alloys, Tool Steel, Stainless Steel, and Nonferrous Metals, Using Sharp Cutting Tools

Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units	Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units
High Temperature Alloys				Stainless Steel	150-175	.60	1.64
A286	165	.82	2.24		175-200	.72	1.97
A286	285	.93	2.54		200-250	.88	2.40
Chromoloy	200	.78	3.22	Zinc Die Cast Alloys	...	.25	0.68
Chromoloy	310	1.18	3.00	Copper (pure)	...	.91	2.48
Inco 700	330	1.12	3.06	Brass			
Inco 702	230	1.10	3.00	Hard	...	.83	2.27
Hastelloy-B	230	1.10	3.00	Medium	...	.50	1.36
M-252	230	1.10	3.00	Soft	...	.25	0.68
M-252	310	1.20	3.28	Leaded	...	.30	0.82
Ti-150A	340	.65	1.77	Bronze			
U-500	375	1.10	3.00	Hard	...	.91	2.48
Monel Metal	...	1.00	2.73	Medium	...	.50	1.36
				Soft	...	.33	0.90
Tool Steel	175-200	.75	2.05	Aluminum			
	200-250	.88	2.40	Cast	...	.25	0.68
	250-300	.98	2.68	Rolled (hard)	...	.33	0.90
	300-350	1.20	3.28	Magnesium Alloys	...	.10	0.27
	350-400	1.30	3.55				

below. Since conditions in different shops may vary and machine tools are not all designed alike, some variation between the results calculated and those experienced on the job is to be expected. However, the calculations do provide a reasonable estimate which will suffice in most practical situations.

For inch units only:

$$Q = 12 V f d \quad (7-6)$$

For SI metric units only:

$$Q = \frac{V}{60} f d \quad (7-7)$$

Table 7-12. Feed Factor, C , for Power Constants

Inch Units				SI Metric Units			
Feed in.*	C	Feed in.*	C	Feed mm†	C	Feed mm†	C
.001	1.60	.014	.97	0.02	1.70	0.35	.97
.002	1.40	.015	.96	0.05	1.40	0.38	.95
.003	1.30	.016	.94	0.07	1.30	0.40	.94
.004	1.25	.018	.92	0.10	1.25	0.45	.92
.005	1.19	.020	.90	0.12	1.20	0.50	.90
.006	1.15	.022	.88	0.15	1.15	0.55	.88
.007	1.11	.025	.86	0.18	1.11	0.60	.87
.008	1.08	.028	.84	0.20	1.08	0.70	.84
.009	1.06	.030	.83	0.22	1.06	0.75	.83
.010	1.04	.032	.82	0.25	1.04	0.80	.82
.011	1.02	.035	.80	0.28	1.01	0.90	.80
.012	1.00	.040	.78	0.30	1.00	1.00	.78
.013	.98	.050	.72	0.33	.98	1.50	.72

* Turning-in./rev; Milling-in./tooth; Planing and Shaping-in./stroke; Borehoning-in./tooth.

† Turning-mm/rev; Milling-mm/tooth; Planing and Shaping-mm/stroke; Borehoning-mm/tooth.

For either inch or SI metric units:

$$P_c = K_p C Q W \quad (7-8)$$

$$P_m = \frac{P_c}{E} = \frac{K_p C Q W}{E} \quad (7-9)$$

where:

 P_c = Power at the cutting tool; hp, or kW P_m = Power at the motor; hp, or kW K_p = Power constant (Tables 7-9, 7-10, and 7-11) Q = Metal removal rate; in.³/min, or cm³/s C = Feed factor for power constant (Table 7-12) W = Tool wear factor (Table 7-13) E = Machine tool efficiency factor (Table 7-14) V = Cutting speed; fpm, or m/min f = Feed rate; in./rev, or mm/rev d = Depth of cut; in., or mm**Example 7-3:**

A 160-180 HB AISI 1040 shaft is to be turned on a geared head lathe. The following cutting conditions are used: cutting speed = 325 fpm, or 100 m/min; feed rate .016 in./rev, or 0.40 mm/rev; and, depth of cut = .100 in., or 2.54 mm. Using both the inch and metric values separately, estimate the power at the cutting tool and at the motor required to take this cut.

For inch units:

K_p = .78 (From Table 7-9); C = .94 (From Table 7-12); W = 1.30 (From Table 7-13); E = .80 (From Table 7-14)

$$Q = 12 V f d = 12 \times 325 \times .016 \times .100 = 6.24 \text{ in.}^3/\text{min}$$

Table 7-13. Tool Wear Factors, W *

Types of Turning Operations	W
All turning operations with sharp cutting tools	1.00
Finish turning (light cuts)	1.10
Normal rough and semi-finish turning	1.30
Extra-heavy-duty rough turning	1.60-2.00

* This table can also be used for planing and shaping.

Table 7-14. Machine Tool Efficiency Factors, E

Type of Drive	E	Type of Drive	E
Direct Belt Drive	.90	Gear Head Drive	.70-.80
Back Gear Drive	.75	—	—

$$P_c = K_p C Q W = .78 \times .94 \times 6.24 \times 1.30 = 5.9 \text{ hp}$$

$$P_m = \frac{P_c}{E} = \frac{5.9}{.80} = 7.4 \text{ hp}$$

For SI metric units:

$$K_p = 2.13 \text{ (From Table 7-9)}; C = .94 \text{ (From Table 7-12)}; W = 1.30 \text{ (From Table 7-13)}; E = .80 \text{ (From Table 7-14)}$$

$$Q = \frac{V}{60} f d = \frac{100}{60} \times 0.40 \times 2.54 = 1.69 \text{ cm}^3/\text{s}$$

$$P_c = K_p C Q W = 2.13 \times .94 \times 1.69 \times 1.30 = 4.4 \text{ kW} \quad (4.4 \times 1.341 = 5.9 \text{ hp})$$

$$P_m = \frac{P_c}{E} = \frac{4.4}{.80} = 5.5 \text{ kW} \quad (5.5 \times 1.341 = 7.4 \text{ hp})$$

Whenever much stock is to be removed from the workpiece, cutting conditions should be selected that will utilize the maximum power available on the machine tool. The cutting conditions should be selected in the following order: 1. select the maximum depth of cut that can be used; 2. select the maximum feed rate that can be used; and, 3. estimate the cutting speed that will utilize the maximum power available on the machine. This order is based on obtaining the maximum possible tool life, while at the same time obtaining as much production as possible from the machine. *The tool life of a cutting tool is most affected by the cutting speed, then by the feed rate, and least of all by the depth of cut.* The maximum metal removal rate that a given machine is capable of machining from a given material is used as the basis for estimating the cutting speed that will utilize all of the power available on the machine.

Example 7-4:

A .125 inch deep cut is to be taken on a 200-210 HB AISI 4350 steel shaft, using a 5 hp lathe which has a machine tool efficiency factor of .80. The maximum feed rate for this job is estimated to be .015 in./rev. Estimate the cutting speed that will utilize the maximum power available on the lathe.

$K_p = .72$ (From Table 7-9); $C = .96$ (From Table 7-12); $W = 1.30$ (From Table 7-13)

$$Q_{\max} = \frac{P_m E}{K_p C W} = \frac{5 \times .80}{.72 \times .96 \times 1.30} \quad \left(P_m = \frac{K_p C Q W}{E} \right)$$

$$= 4.45 \text{ in.}^3/\text{min}$$

$$V = \frac{Q_{\max}}{12 f d} = \frac{4.45}{12 \times .015 \times .125} \quad (Q = 12 V f d)$$

$$= 198 \text{ fpm}$$

Turning on Lathe Centers

The basic operations performed on an engine lathe are illustrated in Fig. 8-1. Those operations performed on external surfaces with a single point cutting tool are called turning. Except for drilling, reaming, and tapping, the operations on internal surfaces are also performed by a single point cutting tool. All of these operations are treated in the next seven chapters of this book. This chapter is confined to the class of lathe work where the workpiece is supported by one or two centers.

All machining operations, including turning and boring, can be classified as roughing, finishing, or semi-finishing. The objective of a roughing operation is to remove the bulk of the material as rapidly and as efficiently as possible, while leaving a small amount of material on the workpiece for the finishing operation. Finishing operations are performed to obtain the final size, shape, and surface finish on the workpiece. Sometimes a semi-finishing operation will precede the finishing operation to leave a small predetermined and uniform amount of stock on the workpiece to be removed by the finishing operation.

Turning on Lathe Centers

Generally, longer workpieces are turned while supported on one or two lathe centers. Cone shaped holes, called center holes, which fit the lathe centers are drilled in the ends of the workpiece—usually along the axis of the cylindrical part. The end of the workpiece adjacent to the tailstock is always supported by a tailstock center, while the end near the headstock may be supported by a headstock center or held in a chuck. In Fig. 8-2 the headstock end of the workpiece is held in a four jaw chuck, while in Fig. 8-3 the workpiece is held in a collet type chuck. This method holds the workpiece firmly and transfers the power to the workpiece smoothly; the additional support to the workpiece provided by the chuck lessens the tendency for chatter to occur when cutting. Precise results can be obtained with this method if care is taken to hold the workpiece accurately in the chuck.

Very precise results can be obtained by supporting the workpiece be-

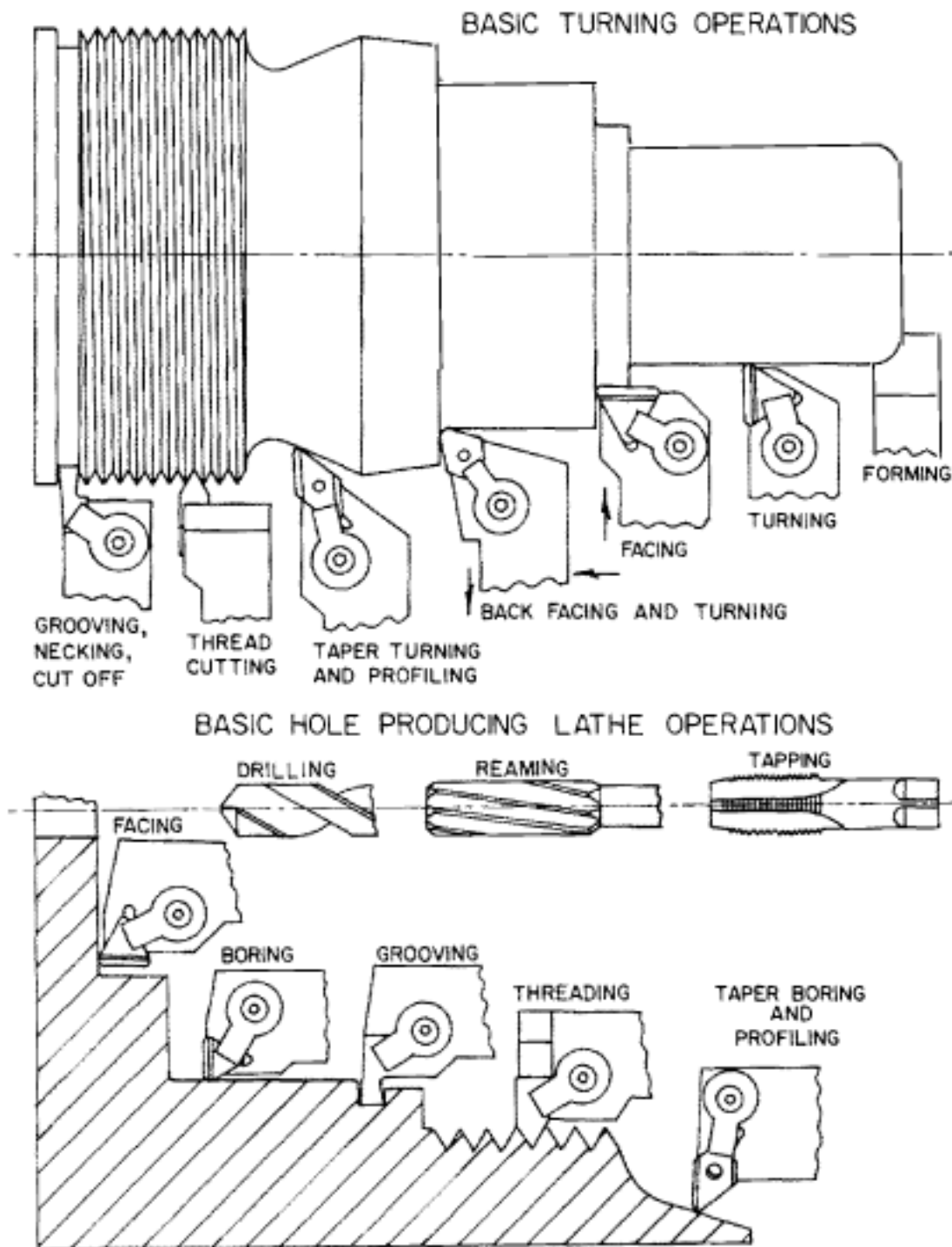
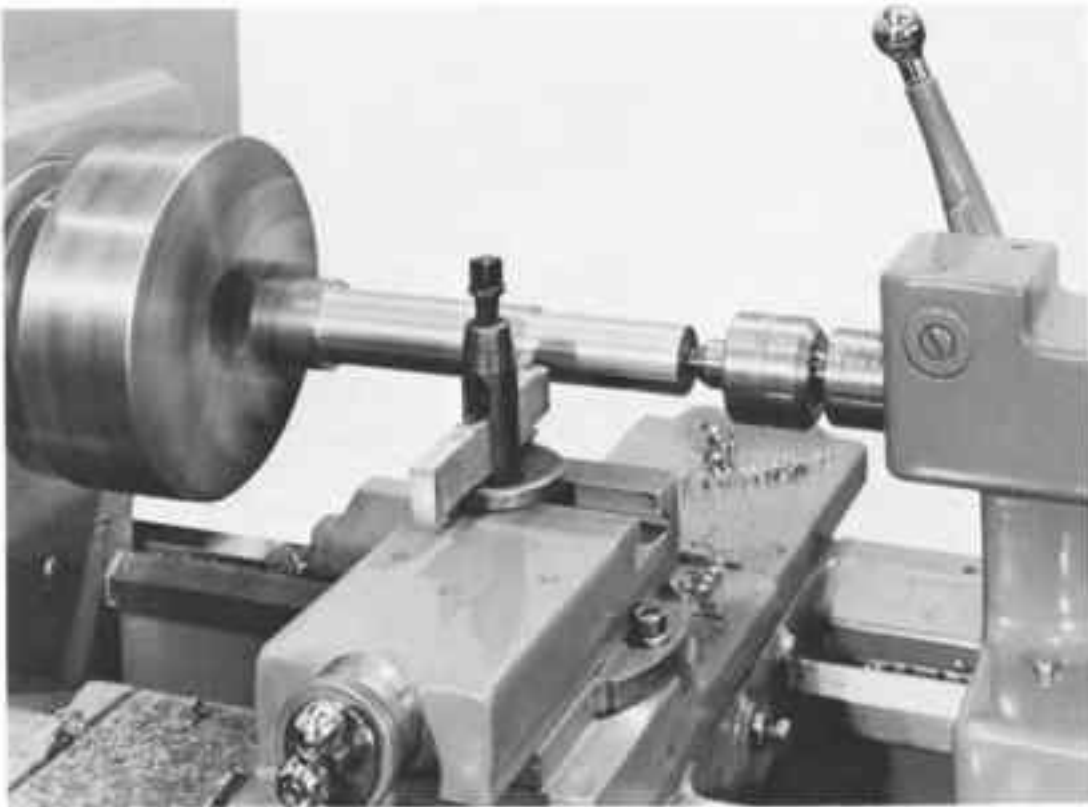


Fig. 8-1. The basic turning operations performed on a lathe. (Top) On outside diameters. (Bottom) In producing holes.

tween two centers, as shown in Fig. 8-4. A lathe dog is clamped to the workpiece; together they are driven by a driver plate mounted on the spindle nose. One end of the workpiece is machined; then the workpiece can be turned around in the lathe to machine the other end. The center

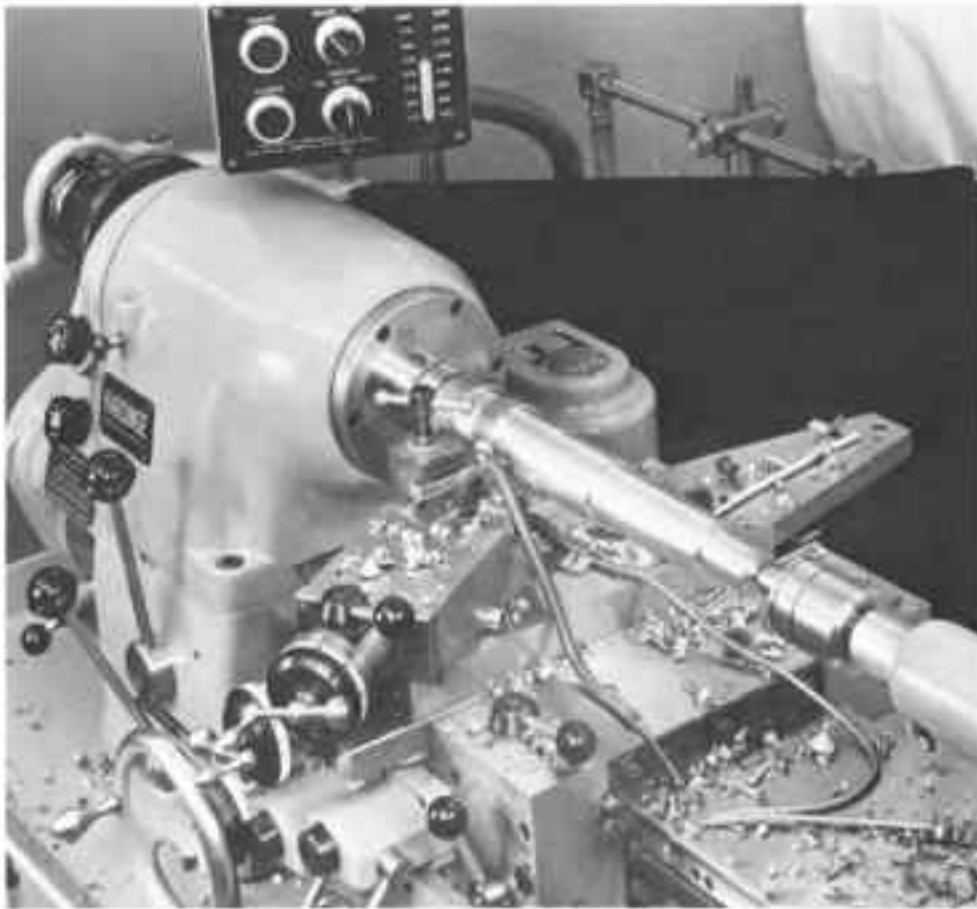


Courtesy of LeBlond Machine Tool Co.

Fig. 8-2. Turning with workpiece mounted on tailstock center and held by a four-jaw chuck.

holes in the workpiece serve as precise locating surfaces as well as bearing surfaces to carry the weight of the workpiece and to resist the cutting forces. After the workpiece has been removed from the lathe for any reason, the center holes will accurately align the workpiece back in the lathe or in another lathe, or in a cylindrical grinding machine. The workpiece must never be held at the headstock end by both a chuck and a lathe center. While at first thought this seems like a quick method of aligning the workpiece in the chuck, this must not be done because it is not possible to press evenly with the jaws against the workpiece while it is also supported by the center. The alignment provided by the center will not be maintained and the pressure of the jaws may damage the center hole, the lathe center, and perhaps even the lathe spindle. Compensating or floating jaw chucks used almost exclusively on high production work provide an exception to the statements made above. These chucks are really work drivers and cannot be used for the same purpose as ordinary three or four-jaw chucks.

While very large diameter workpieces are sometimes mounted on two centers, they are preferably held at the headstock end by faceplate jaws



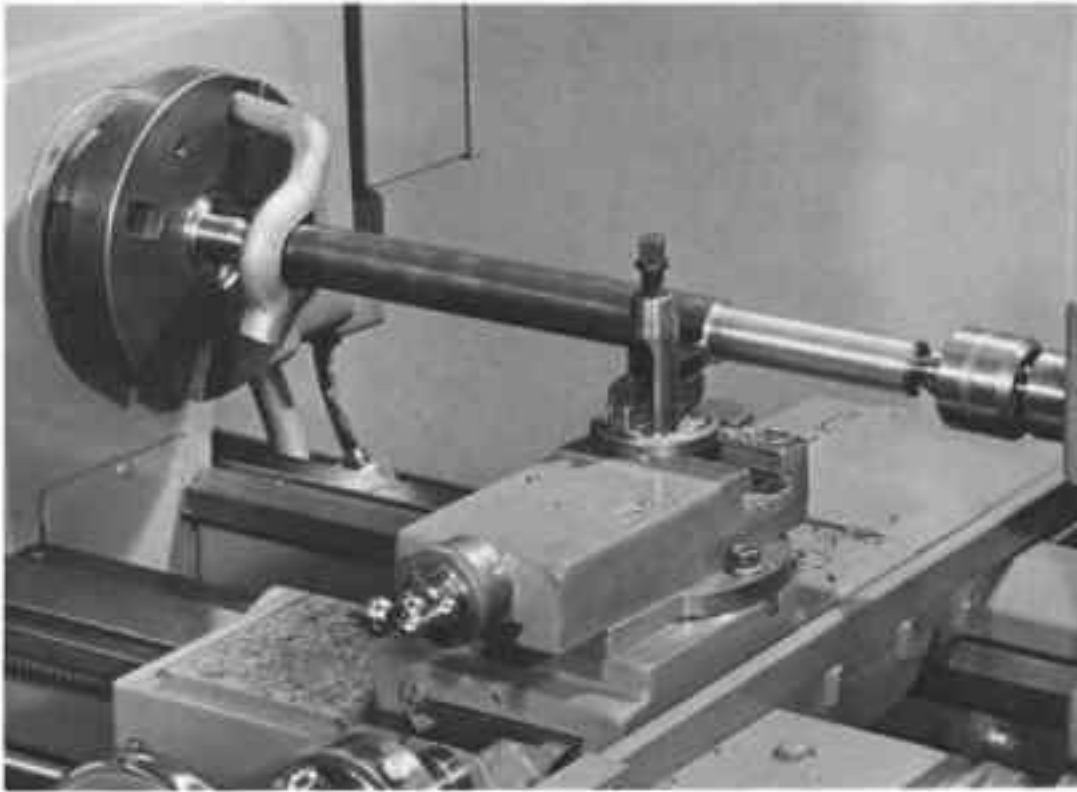
Courtesy of Hardinge Brothers Inc.

Fig. 8-3. Turning with workpiece mounted on the tailstock center and in a collet chuck.

(see Fig. 8-5) to obtain the smooth power transmission; moreover, large lathe dogs that are adequate to transmit the power not generally available, although they can be made as a special. Faceplate jaws are like chuck jaws except that they are mounted on a faceplate, which has less overhang from the spindle bearings than a large chuck would have.

Preparation of the Workpiece

Before the workpiece can be placed between centers in a lathe a center hole must be drilled in each end. This hole is drilled with a combination drill and countersink which is commonly called a center drill (see Fig. 8-6). The conical hole should be drilled deep enough so that a good bearing surface is provided for the lathe centers; it must not, however, be drilled beyond the largest diameter of the center drill. The countersink has a 60-degree included angle and the lead drill provides a clearance for the lathe center points. The axes of the two center holes must coincide with each other so that the lathe centers will seat firmly on the conical surfaces of the holes. There are several methods of drilling the center



Courtesy of LeBlond Machine Tool Co.

Fig. 8-4. Turning between two centers. A lathe dog is clamped to the workpiece, and together they are driven by the driver plate on the lathe spindle nose.

holes in the work. Three different methods for doing this job will be described.

1. The centers of the end faces of the work are determined by laying out, as described in Chapter 3. A deep, center punch-mark is made at the center of each end. A drill chuck is mounted in the headstock spindle of the lathe as shown in Fig. 8-7. The center punched hole of the workpiece is held against the tailstock center, as shown, and the tailstock spindle is advanced in order to feed the other end face into the center drill. One hand is used to turn the tailstock hand-wheel while the other hand is used to hold the work. The center drill is started in the center punch mark and the hole is drilled to the correct depth. The work is then turned around and the other center hole is drilled. This is an accurate method of drilling the center hole when the workpiece is light enough in weight to be easily handled.
2. Short workpieces that can be held in a chuck without protruding too far, can be center drilled, as shown in Fig. 8-8. The workpiece must be accurately held in the chuck, otherwise the axes of the two center holes will not coincide. If the three-jaw universal chuck



Courtesy of LeBlond Machine Tool Co.

Fig. 8-5. Measuring the diameter of a workpiece in a lathe. The part is held by faceplate jaws at the headstock end.

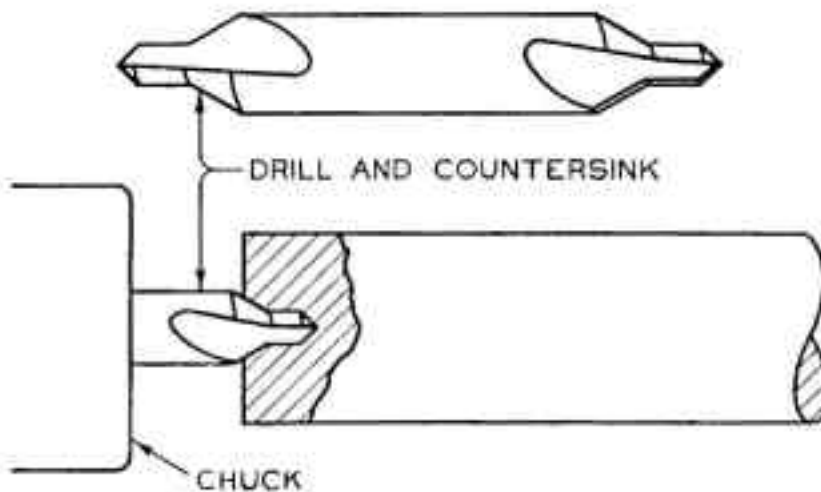


Fig. 8-6. Combination drill and countersink, or center drill, and its application in centering parts for turning.

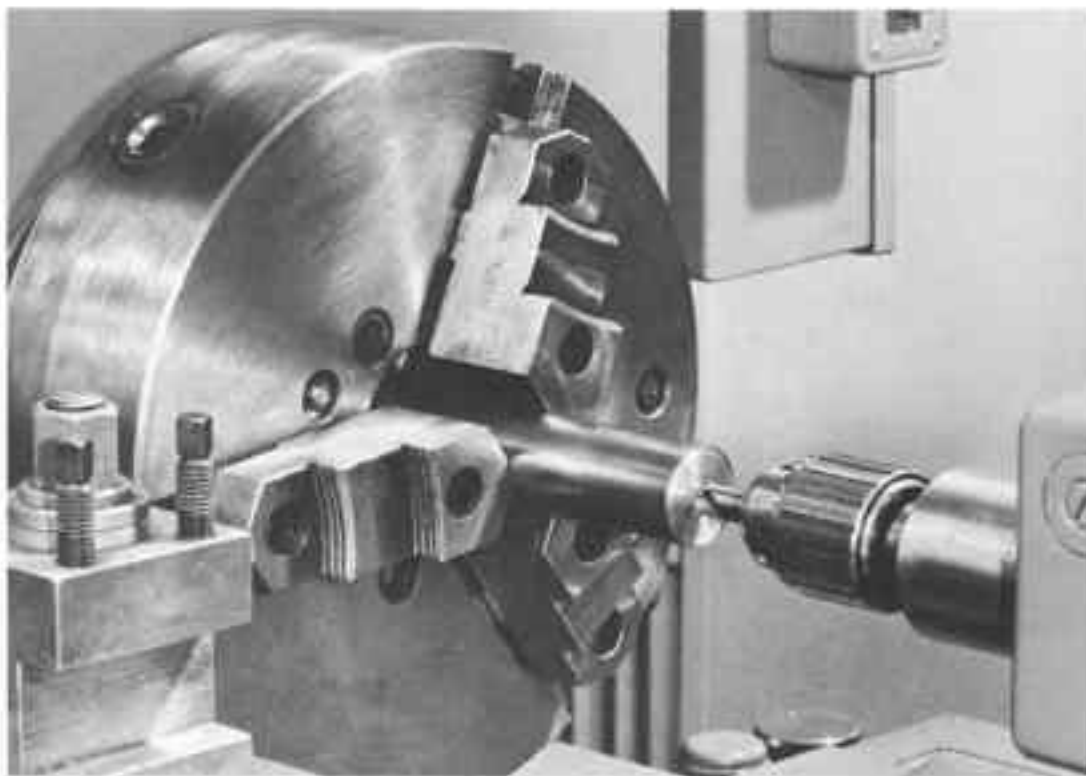
does not run true, it is best to use a four-jaw, independent chuck that can be adjusted to hold the workpiece concentric within one-thousandth of an inch, if desired. This is accomplished by placing a dial test indicator against the workpiece and adjusting the chuck until the hand of the indicator does not move when the lathe spindle is slowly turned by hand. A center drill is placed in



Fig. 8-7. Drilling center holes in the ends of a cylindrical workpiece.

a drill chuck which is held in the tailstock spindle. The tailstock spindle is advanced manually to drill the hole to the correct depth. This method has the advantage of not requiring a layout to be made on the workpiece prior to drilling.

3. Long workpieces, and those that are too heavy to be held by hand, are center drilled as shown in Fig. 8-9. The work is held by a three-jaw, universal chuck or by a four-jaw, independent chuck. It must be held concentric with the rotation of the lathe. The other end of the workpiece is supported by a steady rest. The steady rest must be adjusted to support the work so that its axis coincides with the axis of the lathe. One method by which this can be done is to find the center of the end face of the work by making a layout as described in Chapter 3. A tailstock center is then positioned opposite the layout mark which determines this center and the steady rest is adjusted until this layout mark is in alignment with the point of the tailstock center. The work can also be aligned by using a dial test indicator that is attached to a surface gage. The contact point of the indicator is moved over the top of the work, and along the side of the work, at each end. When indicating the side of the work, parallelism between the work and the axis of the lathe is obtained by using the finish machined vertical sides of the lathe bed as a reference surface for the surface gage. Accurate results can be obtained by using a surface gage without an indicator. The point of the surface gage scriber is adjusted to touch the top and side of the work at each end. This method is preferred when the work has a rough surface scale.



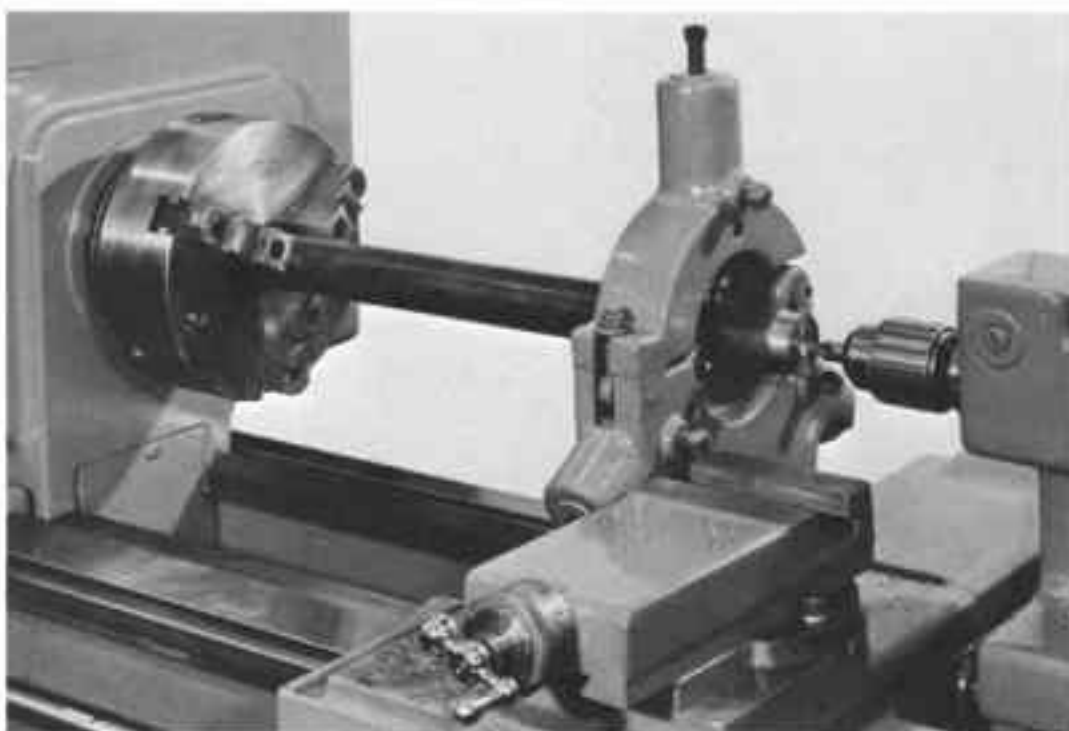
Courtesy of LeBlond Machine Tool Co.

Fig. 8-8. Drilling the center hole in the end of a short workpiece by holding it in a chuck.

Before the center holes are drilled, a facing cut should be taken across the end of the work. This is done as shown at I, Fig. 7-14. The work can be machined to the correct length by facing both ends. A short spot should be turned on the outside diameter adjacent to the end face as shown in Fig. 8-9. The spot is turned by cutting just deeply enough to produce a cylindrical surface that is concentric with the center hole in the end. The purpose of the spot is to provide a reference surface which can be gripped by the chuck and against which an indicator can be placed in order to test the concentricity with which that end of the work is held by the chuck. The center hole is drilled in each end using the tailstock, as shown in Fig. 8-9.

The Machine Setting

A part of any machine tool setup is to set the machine to operate at the correct cutting speed and feed rate. Most machine tool builders provide instruction plates attached to the machine which provide the necessary information for the operator to adjust the handles and levers so that the machine will operate at the desired speed and feed. The procedure for selecting the correct cutting speed and for calculating the spindle speed has been treated in detail in Chapter 7. While the feed rate was



Courtesy of LeBlond Machine Tool Co.

Fig. 8-9 Drilling center holes in the ends of a longer workpiece.

also treated in this chapter, a further treatment of this subject will be helpful.

The feed rate cannot be calculated; it is largely a matter of exercising sound judgment based upon experience. All of the following factors must be considered when selecting the feed: 1. the depth of cut to be used; 2. the surface finish to be produced on the workpiece and the nose radius of the cutting tool; 3. the type of cut, finishing or roughing; 4. the size of the lathe and the power available; 5. the rigidity of the workpiece and the cutting tool; and, 6. how securely and rigidly the workpiece is held in the lathe. Table 8-1 provides some representative feed rates. It can be used as a guide in selecting feeds, based on the size of the lathe.

Table 8-1. Guide for Selecting Feed Rates

Size of Lathe	Finishing Cuts		Medium-Roughing Cuts		Heavy-Duty Roughing Cuts	
	in./rev	mm/rev	in./rev	mm/rev	in./rev	mm/rev
Light Bench Lathes	.002 to .005	0.05 to 0.12	.007 to .010	0.18 to 0.25	.010 to .015	0.25 to 0.40
12" to 16" Lathes	.002 to .010	0.05 to 0.25	.010 to .015	0.25 to 0.40	.020 to .040	0.50 to 1.02
24" to 32" Lathes	.003 to .012	0.07 to 0.30	.010 to 0.20	0.25 to 0.50	.020 to .060	0.50 to 1.52
36" and Larger Lathes	.005 to .015	0.12 to 0.40	.012 to .025	0.30 to 0.75	.030 to .105	0.75 to 2.54

Preparation of the Engine Lathe

The engine lathe must be level in two directions in order to produce accurate work. If it is not level the bed will have a twist which will cause the headstock and the tailstock to be misaligned with the movement of the carriage and the cross slide. A precision level that is accurate to .0005 inch per foot is used to level the lathe. It must be level, parallel and perpendicular to the lathe axis at both extremes of the lathe bed. Leveling screws or wedges are provided on the legs of the lathe to level the bed. The lathe should be leveled when installed, and a periodic check should be made to make certain that it remains level. It is not, however, necessary to check this before doing every job.

If the workpiece is to be held between the headstock and the tailstock center the following preparations must be made. The taper in the headstock must be thoroughly cleaned before the center adapter is installed. The cleanliness of the headstock taper, as well as the taper of the adapter should be checked by using the bare hand or fingers. The presence of nicks or small particles of dirt or metal could not be detected through a rag or cloth placed on the tapers; however, they can readily be felt by the fingers or the bare hand. All taper fits on any machine tool should be checked in this manner. If the taper is dirty, or badly nicked, it will not hold and it will not provide the desired accurate alignment of the mating parts. After the adapter has been inserted in the lathe spindle, the headstock center is placed in the adapter using the same care (see Fig. 8-10).

The headstock center of the lathe always rotates with the lathe spindle; therefore it is called a live center. Tailstock centers may be of the type that also rotate with the workpiece; however, frequently they are stationary or dead centers. The cone point of the headstock center must rotate without any eccentricity otherwise the two ends of the finish-turned workpiece will not be concentric to each other. Such eccentricity cannot be tolerated on precision parts.

To show how this eccentricity can occur, assume that the workpiece is held in a lathe between a live headstock center and a dead tailstock center. Furthermore, assume that the tailstock center is accurately aligned with the axis of the lathe but that the headstock center is not rotating concentrically with respect to the lathe axis. The longitudinal feed, guided by the ways of the lathe, will cause the path of the cutting tool to be parallel to the lathe axis. As this tool cuts the rotating workpiece it will produce a cylindrical surface which is concentric to the tailstock center and to the center hole in the tailstock end of the workpiece. When the workpiece is turned end for end in the lathe, the center hole, which is concentric with the first-turned surface, will now be mounted on the eccentrically rotating headstock center which will cause the first turned surface to rotate eccentrically. While the first-turned surface is turning eccentrically, the cutting tool will turn the other end of the workpiece concentric with the axis of the lathe as before. Thus, the surfaces turned at each end of the work cannot be concentric with each other as shown

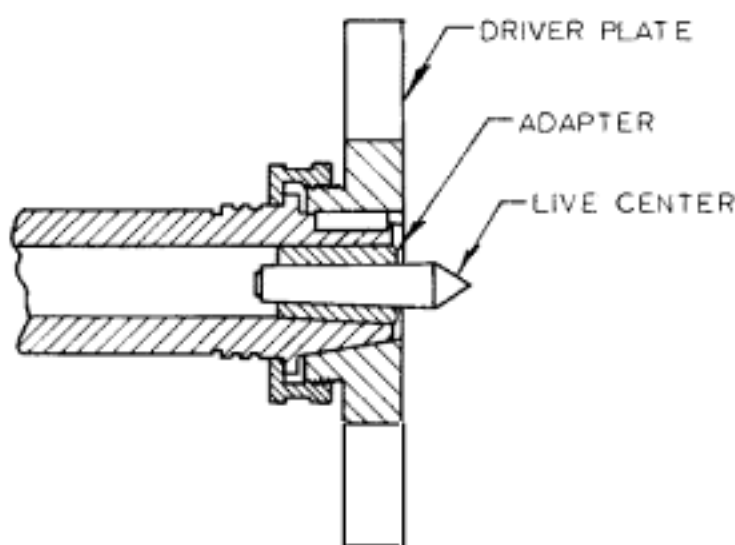


Fig. 8-10. Type L, long taper spindle-nose showing driver plate, adapter, and live headstock center mounted on spindle. Adapter allows for larger spindle bore. Live center and adapter can be removed by tapping with a bar placed through other end of spindle.

in Fig. 8-11. This can only be corrected by truing the headstock center so that it rotates concentrically, with respect to the lathe axis.

Headstock centers may be made from soft steel or from hardened steel. Soft centers have the advantage of being easily trued by simply turning a 60 degree cone point, while mounted in the headstock of the lathe as shown at A in Fig. 8-12. The compound rest is positioned at 30 degrees, with respect to the lathe axis, and a cut is taken by feeding the tool with the compound rest slide. Only very shallow cuts should be taken until the cone point of the center runs true. With the compound rest in this position, the cone point of the center will have an included angle of 60 degrees. This angle must be accurate and should be checked with a center gage or with a precision bevel protractor.

Hardened headstock centers usually retain their accuracy for a longer period. They are trued in a manner that is similar to truing a soft center except that the cone point is ground with a grinding wheel instead of a single-point tool as shown at B, in Fig. 8-12. The grinding wheel is driven by a tool post grinding attachment which is mounted on the compound rest. The depth of cut of the grinding wheel is adjusted with the cross feed and the wheel is fed across the cone point of the center by the compound rest slide. The cone point is ground until it runs true and the included angle is correct. Hardened tailstock centers can be ground in the same manner by mounting them in the headstock spindle of the lathe.

The tailstock may be a live center that rotates with the workpiece or it may be a dead center that does not rotate. The live tailstock center, shown in Fig. 8-13, is mounted in precision anti-friction bearings. Live

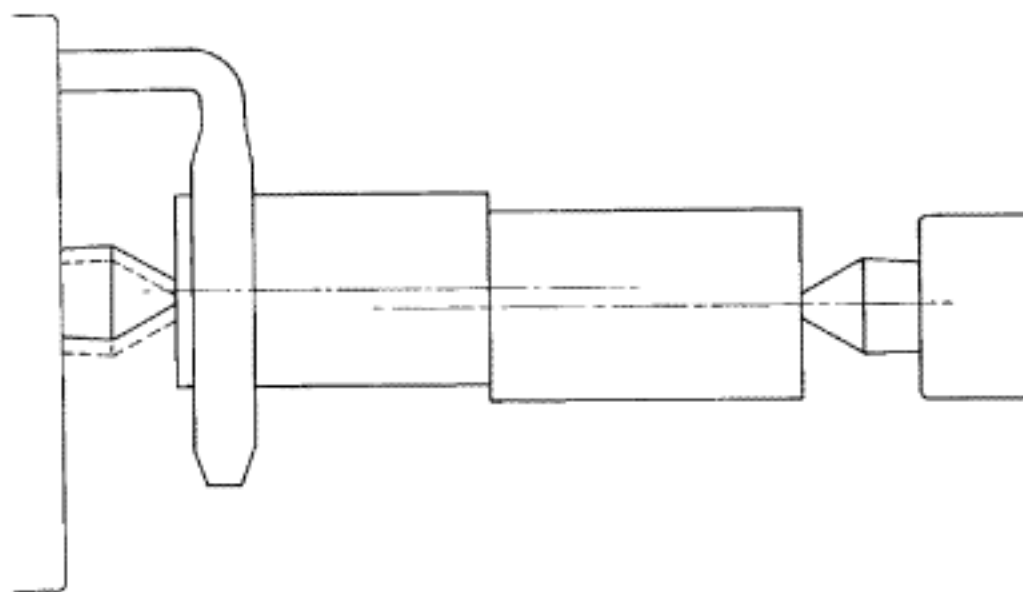


Fig. 8-11. Finish-turned surfaces not concentric; caused by turning each end with an eccentrically rotating live center.

tailstock centers are a necessity when turning at very high cutting speeds such as those normally used with cemented carbide cutting tools. The cone point of the live tailstock center must run true for the same reasons that the headstock center must run true. This is checked with a dial test indicator. The center is slowly turned, manually, and any movement of the indicator hand is noted. If the center does not run true it must be replaced. It must be disassembled and reground in order to be trued. Dead tailstock centers are always hardened because the cone point acts as a bearing surface for the rotating workpiece. All the surfaces of the dead center must be smooth; they are finished by grinding.

Another fundamental requirement for turning a cylindrical workpiece that is held on the tailstock center is that this center must be aligned to coincide with the axis of the lathe. If the work is also held by a center at the headstock end, the headstock and the tailstock centers must be aligned with each other. If the tailstock center is misaligned, the turned surface on the work will be tapered instead of cylindrical. For every .001 inch that the tailstock center is misaligned in a direction crosswise to the lathe bed, the workpiece will be tapered .002 inch in its length. It can thus be seen that the tailstock center must be very accurately aligned in a crosswise direction. An error in the vertical alignment of the tailstock center is not nearly as critical when turning. For example, to cause a taper of .001 inch to be turned in the length of a workpiece that is one inch in diameter, the vertical misalignment must be .022 inch; for a 4-inch diameter, the vertical misalignment must be .045 inch. The vertical alignment built into precision lathes is within a few thousandths of an inch

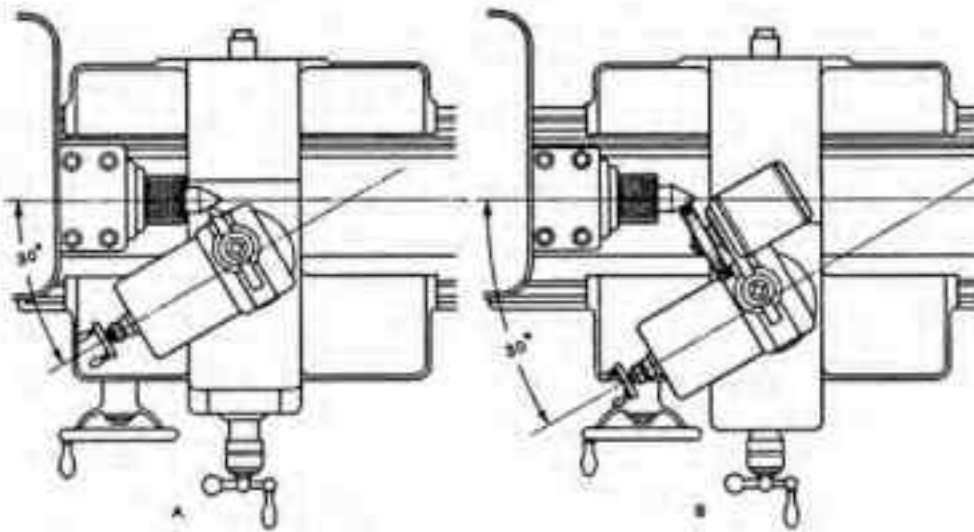
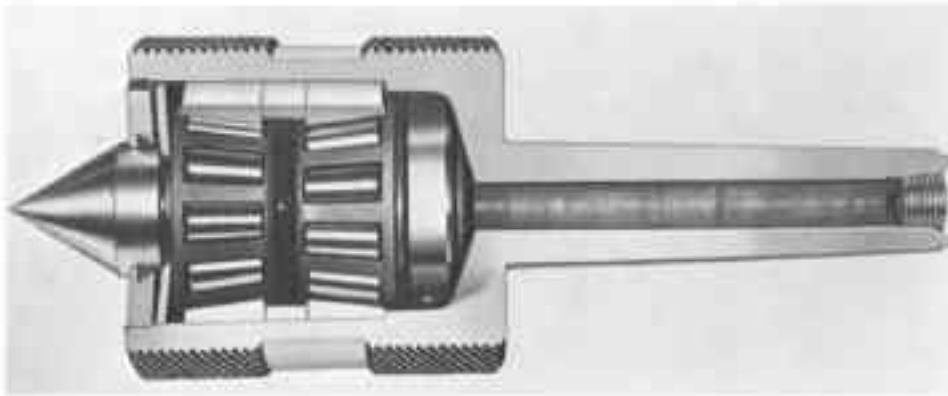


Fig. 8-42. A. Truing a soft headstock center with a lathe tool. B. Truing a hardened headstock center by grinding with a tool post grinder.



Courtesy of the Ideal Industries Inc.

Fig. 8-13 Live tailstock center mounted on precision antifriction bearings.

and the vertical misalignment of the tailstock center is almost never a cause of taper unless the lathe is very badly worn. However, a small vertical misalignment can cause a straight reamer, which is held in the tailstock spindle, to ream a slightly tapered hole.

A check should be made to determine the accuracy of the alignment of the tailstock center before any attempt is made to turn a cylindrical surface unless it is known to be in alignment. If the tailstock center is not in alignment, steps must be taken to bring it into alignment. There are several very precise methods that can be used; these are described later in this section. First, however, as a rough check, the tailstock can be moved up to the headstock until the two centers are opposite each other and their

alignment can be checked visually. If a headstock center is not being used, another method of making a rough check is to check the alignment scale at the end of the tailstock (see Fig. 6-20). If a misalignment is indicated, the clamp bolt should be loosened, and the two adjusting screws turned, until the proper alignment is obtained.

A more precise check for the alignment of the tailstock should be made before attempting to turn an accurate cylindrical surface. The recommended procedure for *checking this alignment* is to cut two spots on the workpiece as shown at A and B in Fig. 8-14. The two spots should be cut as far apart as possible on the workpiece. In the case of a piece of cylindrical bar stock, as shown in Fig. 8-14, one of the spots should be cut as close to the lathe dog as possible while the other spot is cut at the tailstock end. If a casting or forging is being turned, the spots should be cut at each end of the longest cylindrical surface to be turned. The procedure for making a precision check of the alignment is detailed step-by-step below:

1. Set up the lathe and the cutting tool as shown at A and B, Fig. 8-14. Check to make sure that the lathe dog will not hit the carriage when the cutting tool is close to the dog. Also, check to make sure that the cutting tool can be positioned clear of the tailstock end of the workpiece in order to be able to cut the spot at that end. Adjust the cutting speed and the feed of the lathe. Use a feed of approximately 0.005 inch per revolution.
2. Position the cutting tool approximately one inch to the right of the lathe dog.
3. Start the lathe spindle and slowly feed the cutting tool into the workpiece, by hand, until a complete cylindrical surface is being turned. Then, engage the longitudinal feed and allow the tool to cut as close to the dog as possible, before disengaging the feed.
4. Withdraw the cutting tool from the workpiece with the cross slide and move the carriage until the cutting tool is clear of the tailstock end of the workpiece.
5. Stop the lathe and measure the diameter of the first spot with a micrometer caliper.
6. Cut the second spot at the tailstock end so that its length is about $\frac{3}{4}$ inch and its diameter is equal to the diameter of the first spot. Use the trial cut procedure to turn the diameter to size. Briefly, take a short cut and with the spindle stopped, measure the diameter with a micrometer caliper. Move the cutting tool longitudinally to clear the end of the workpiece again and then move the cross slide inward the distance required to cut this spot to size, as determined by the measurement. Start the spindle and cut the spot to size.
7. Attach a dial test indicator to the cross slide as shown in Fig.

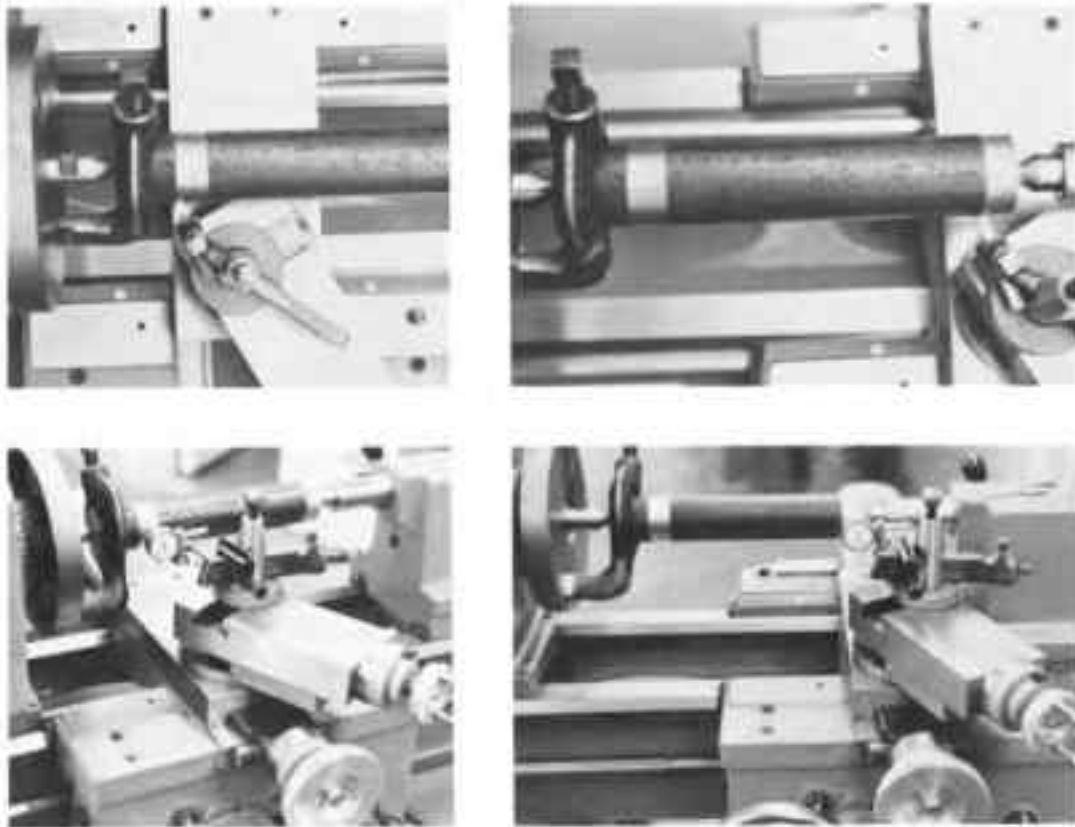


Fig. 8-14. Procedure for aligning the tailstock center. (Top) First Step—cutting spot at headstock end of work. Second step—cutting spot at tailstock end of work. Third step—Indicating headstock spot. Final step—Indicating tailstock spot.

8-14C and D. The indicator should be graduated to read .001 inch, or less.

8. Position the indicator against the first spot at the headstock end and move the cross slide until the indicator reads zero.
9. With the indicator reading zero, note the reading of the cross feed micrometer dial. It is convenient, if possible, to set the micrometer dial to read zero at this position since this reading can easily be remembered. This position of the cross slide is called the zero reference position. All of the indicator readings on both spots are made with the cross slide at the zero reference position, as determined by the cross feed micrometer dial reading.
10. Withdraw the indicator from the first spot with the cross slide and move the carriage until the indicator is opposite the second spot at the tailstock end.
11. Move the cross slide inward to the zero reference position, as determined by the cross feed micrometer dial reading. The indicator will now touch the second spot.

12. Read the reading of the dial test indicator. If the reading is zero, the tailstock center is aligned and no further work is required.
13. If the indicator reading is not zero, adjust the tailstock crosswise until the indicator reads zero. To adjust the tailstock, loosen the clamp bolt (Fig. 6-20) and turn the adjusting screws until the indicator reads zero. Then tighten the clamp bolt and check the indicator reading again. The indicator must read zero with the clamp bolt tightened. If necessary, repeat this procedure until the indicator reads zero with the clamp bolt tight.
14. Check the alignment again by taking indicator readings on both spots with the cross slide at the zero reference position. If necessary, adjust the tailstock again until the indicator readings on the two spots are zero.

After the spots have both been cut to the same size, it will not require much time to adjust the tailstock into alignment. The spot at the headstock end should always be cut first because it is easier to use the trial cut procedure at the tailstock end where the tool can clear the end of the workpiece.

In the procedure just given, the two spots act as gaging or reference surfaces. A special test bar can be made having two equal diameter reference surfaces, one at each end. When this bar is placed between centers in the lathe, the procedure for aligning the tailstock center is similar to the procedure just described. Since the two spots do not have to be cut, the test bar method is fast. However, on some older worn lathes it is not always precise. If the test bar is not approximately the same length as the workpiece, the tailstock will have to be moved along the lathe bed after it has been aligned with the test bar. Moving the tailstock from one position to another on some older lathes will cause the alignment to change slightly, but only a slight misalignment will prevent the lathe from doing precise work. Remember that for each .001 inch misalignment of the tailstock center, the taper along the length of the workpiece will be twice this amount.

Another method is to cut two spots as before, except that both spots are cut with the cross slide in the same position, as determined by the cross feed micrometer dial reading. The two spots are measured with a micrometer caliper and the tailstock is moved a distance equal to one-half of the difference in the measurement. A second cut over the two spots must then be taken and both spots measured again to check the previous tailstock adjustment. This procedure is repeated until the measurements over both spots are equal. Since this procedure requires repeated cuts to be taken over the spots, it is slower than the others.

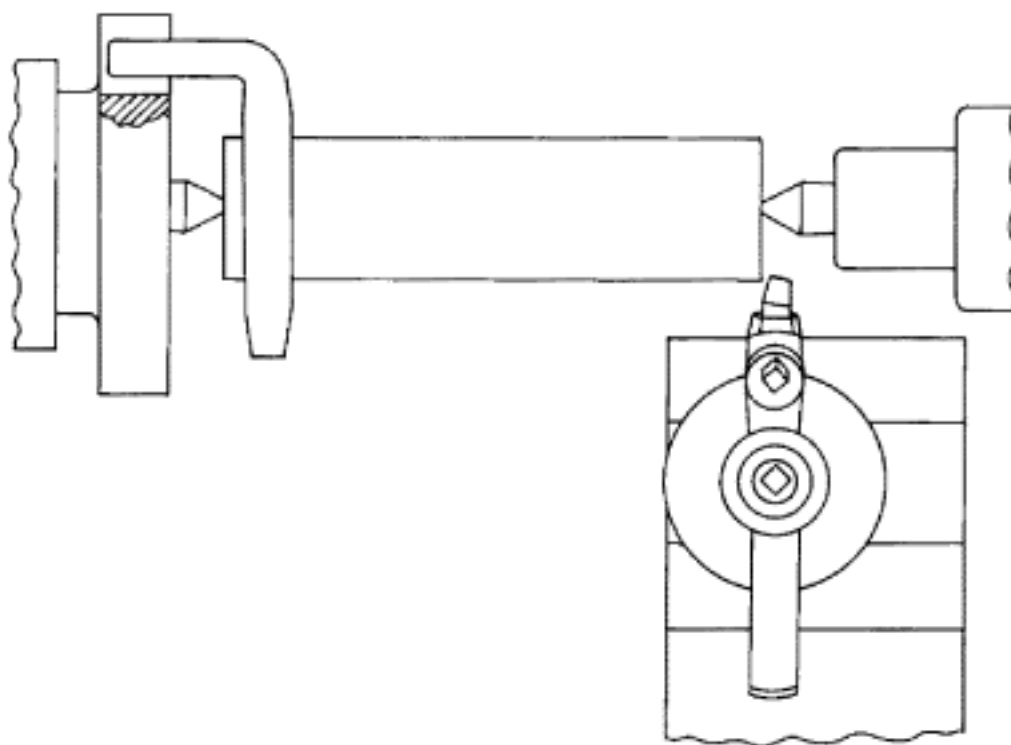


Fig. 8-15. The correct setup for turning between lathe centers.

Setting-Up the Work for Turning Between Centers

The correct setup for turning between centers is shown in Fig. 8-15. A driver plate is mounted on the spindle nose of the lathe. The driver plate is a small face plate that is used to drive the lathe dog.

Lathe dogs are used to transmit the drive from the driver plate of the lathe to the workpiece when turning between centers. Several different types of lathe dogs used, are shown in Fig. 8-16. The lathe dog is placed over the workpiece but is not tightened until later.

When a nonrotating, or dead, tailstock center is used, a lubricant must be used between this center and the workpiece. Although proprietary lubricants are available, a good tailstock lubricant can be made by mixing a small amount of lubricating oil with painter's white lead. No lubricant is required if a live tailstock center is used nor is a lubricant used on the headstock center. The workpiece is then placed between the centers. The lathe dog is positioned with the bent end, or tail, between one of the slots on the faceplate and is clamped to the work. The tail of the dog must not bind in the bottom of the driver plate slot, as shown in Fig. 8-17, as this will cause the headstock center to become unseated. When this occurs a larger lathe dog must be used. The tailstock spindle is adjusted until the work seats on both centers, without binding, and the spindle is clamped in place. It is important to check this adjustment frequently, during the

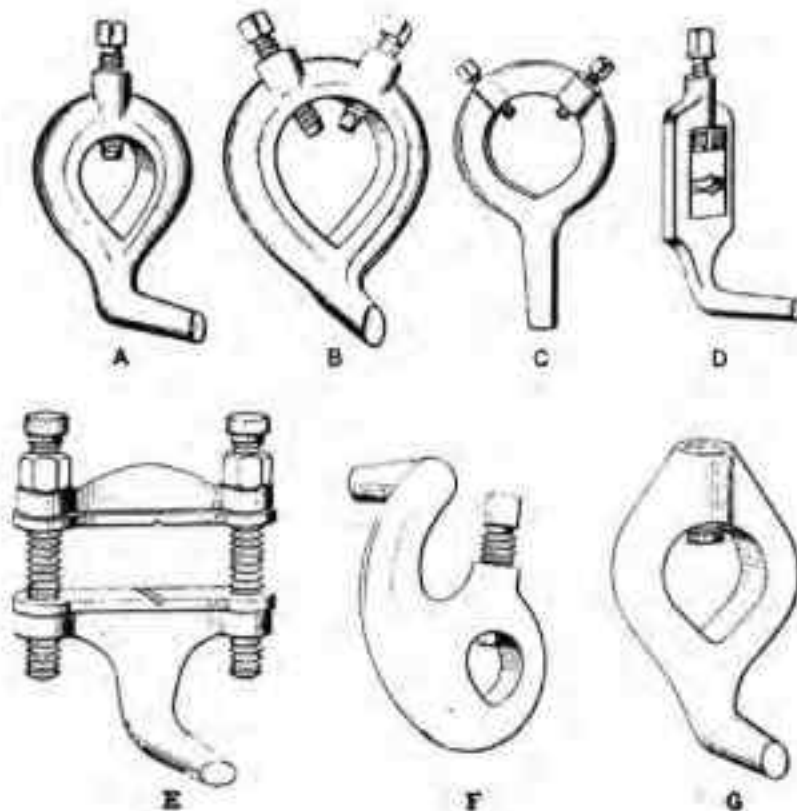


Fig. 8-16. Types of dogs, or drivers, commonly used in connection with lathe work.

course of the turning operation. When a heavy cut is taken, some of the heat generated by the cutting action is absorbed by the workpiece which will cause it to expand in length enough to make the centers bind. When this occurs, the tailstock center will overheat and, consequently, be ruined. When a heavy roughing cut is followed by a light finishing cut, the work will lose heat and contract, causing the adjustment of the centers to be too loose. A turning tool (Figs. 7-14 and 7-15) is selected and clamped in the tool post. The overhang of the cutting edge beyond the tool post should be kept to a minimum in order to make the setup as rigid as possible. The nose of the cutting tool should be positioned "on center" or at the height of the lathe centers, as shown in Fig. 8-18. The turning tool is usually, but not always, held about square with the workpiece as in Fig. 8-15. When turning close to the lathe dog or to a large shoulder (see Figs. 8-14 and 8-23), or for other reasons, it may be held at an angle with respect to the workpiece axis.

End Facing with the Work Held Between Centers

The first operation that is generally performed after the work has been set up is to face the ends of the work to length. If the ends have been

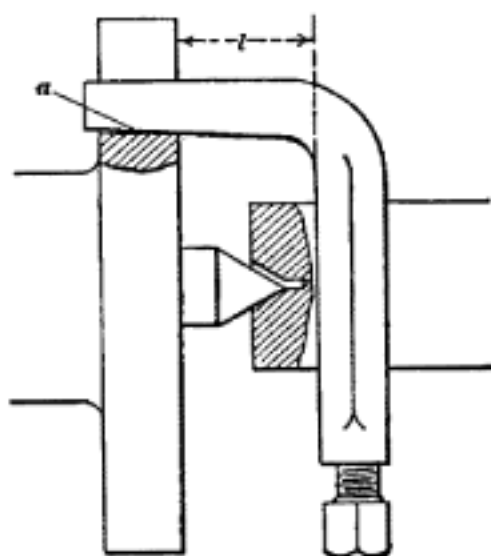


Fig. 8-17. Dog that is too short for driver plate.

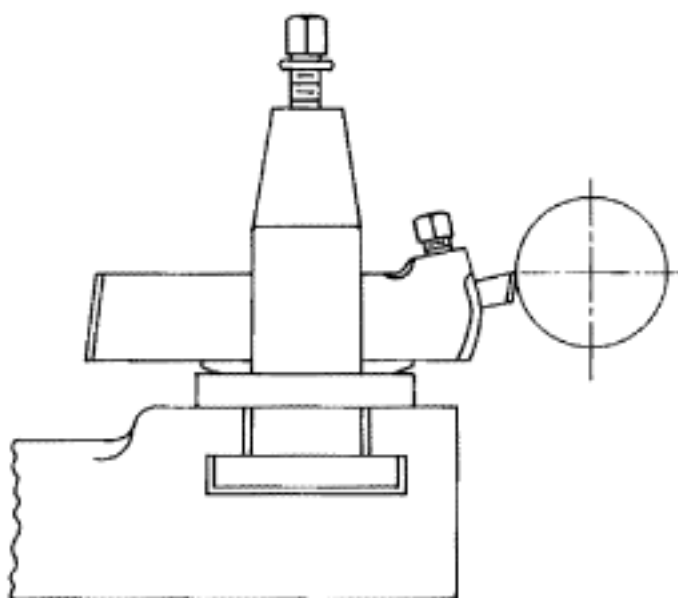


Fig. 8-18. Correct vertical position of cutting tool for turning. The nose of the cutting tool should always be "on center," or at the height of the lathe centers.

faced while drilling the center holes, this operation is, of course, unnecessary. An end facing tool, Fig. 8-19, and G, Fig. 7-14, is placed in the lathe tool holder. It is positioned with the nose leading slightly, as shown in Fig. 8-19. The end face of the workpiece is rough faced, using the longitudinal feed handwheel on the apron to establish the depth of cut, and the tool is fed across the end face with the cross slide. The direction

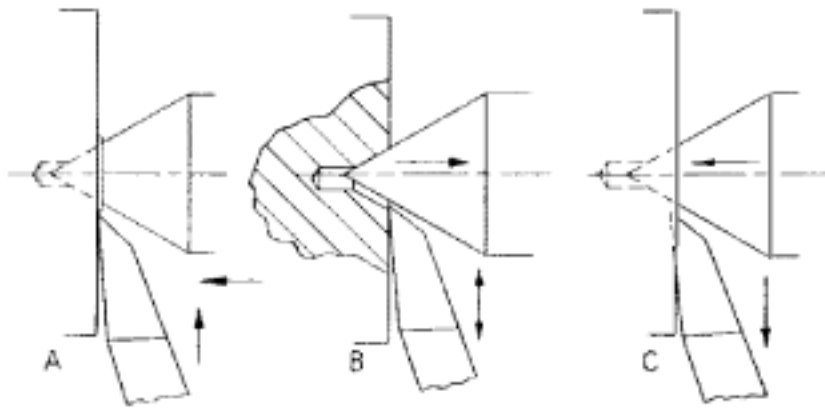


Fig. 8-19. Procedure for cutting off burr surrounding center hole, using an end facing tool.

of the feed should be outward; i.e., from the center toward the outside diameter of the workpiece. This operation leaves a sharp, unsightly burr surrounding the center hole on the work, which must be removed by the finish facing operation.

The procedure for removing this burr is illustrated in Fig. 8-19. The end-facing tool is positioned near the tailstock center, as shown at A. The tool is then moved into the face of the work using the longitudinal feed handwheel on the apron in order to obtain the depth of cut. The following steps must then be taken in rapid order:

1. The tailstock spindle is unclamped.
2. The tailstock center is moved out a very short distance by moving the spindle with the tailstock handwheel.
3. Simultaneously with Step 2, the facing tool is fed toward the center by manually turning the cross-feed handle. See B, Fig. 8-19. This procedure will cause the burr surrounding the center hole to be cut off.
4. Immediately after the burr is cut off the facing tool is backed out, again using the manual cross feed, and the center is repositioned in the center hole.
5. The finish-facing cut is then completed by facing from the center toward the outside, as shown at C. The power cross feed may be used to take this cut.

The work is turned around in the lathe in order to face the second end. A simple layout is made before this operation is begun. A line is laid out from the finished face of the work to some distance less than the required length of the work, as shown in Fig. 8-20. It is convenient to make this distance one inch less than the required length of the work. It would be

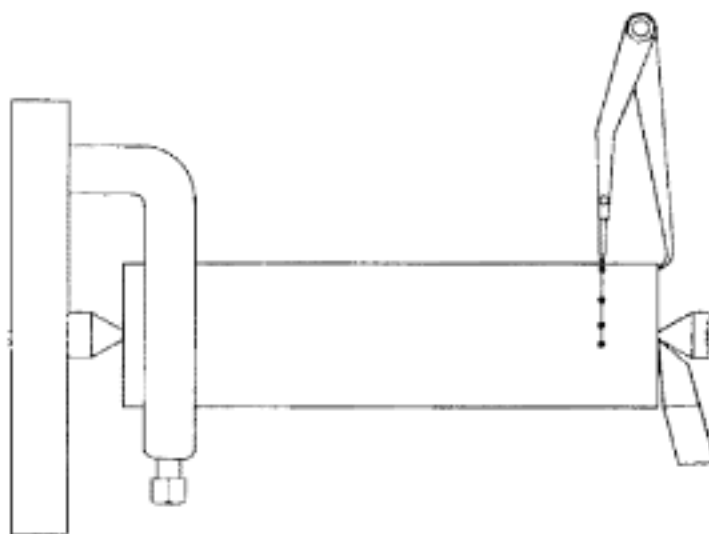


Fig. 8-20. Measuring length of workpiece by measuring to layout line with hermaphrodite calipers. These calipers can also be used to scribe layout lines on cylindrical surfaces, as shown.

difficult to scribe a line at the exact length of the work as it would fall too close to the existing face. Furthermore, the chips resulting from the facing cut might possibly destroy the layout before the facing operation is completed. After the layout has been made, the facing operation is performed, as described above, until the distance from the second face to the layout line is one inch, as measured by a rule or by hermaphrodite calipers.

Cylindrical Turning

As already explained, two types of cuts are normally taken when turning; namely, roughing cuts and finishing cuts. In either case, the cutting tool is positioned to cut the required diameter by using the trial-cut procedure. Since it is so frequently used this method will be described again, in greater detail.

The cutting tool is first adjusted to take a shallow cut using the cross slide to position the tool. Care must be taken to make sure that this cut will leave the work oversize. The automatic, longitudinal power feed is engaged and a cut, approximately one-half inch in length, is taken. With the lathe spindle stopped, the diameter of the work is measured with micrometer calipers. The carriage is then moved back toward the tailstock in order to position the tool for taking a second cut. Before this cut is taken, the cutting tool is adjusted to cut the workpiece to the desired diameter by moving the cross slide the exact distance required. This distance is one-half the difference between the measured diameter of the trial cut and the desired diameter. The distance that the cross slide is moved can be accurately determined by reading the micrometer cross-feed dial. As explained in Chapter 5, some cross-feed dials are direct reading,

while others must be moved one-half of the difference between the trial-cut diameter and the desired diameter. Since direct reading dials have graduations that are equal to a movement of one-half of a thousandth of an inch, this subtraction is not necessary. On direct-reading dials the cross slide is moved the required distance by reading the number of graduations corresponding to the difference in the diameter. It must be understood that in either case, the cross slide moves the same distance, and that the difference is in how the micrometer cross-feed dials are graduated. After the cutting tool has been set to take the second cut, the longitudinal feed is engaged and another one-half-inch-long cut is taken. The resulting diameter is measured. If the diameter of the work is to size, the longitudinal feed is engaged and the cut is taken to the required length. It is sometimes necessary to take several trial cuts before the desired diameter on the work is obtained.

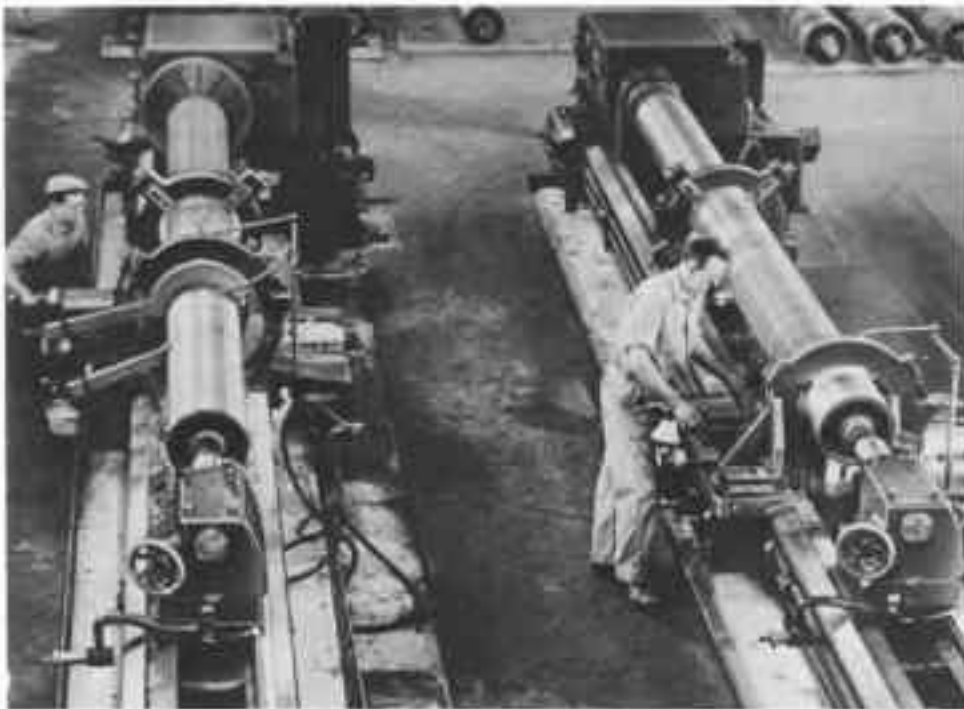
The entire workpiece should be "roughed out" before taking any finish cuts by taking one or more roughing cuts on each diameter that is to be turned. The roughing cuts should leave the diameter of the workpiece oversize a predetermined amount, which in general practice, may vary from approximately .020 to .060 inch. Shoulder lengths should be left short, or "undersize," in order to provide metal for the finish-turning operation. This amount is usually, approximately, .015 to .030 inch.

The finishing cut is often taken with the same cutting tool as was used to take the roughing cut. This procedure requires fewer changes of cutting tools. It is sometimes best, however, to use a separate tool for finish turning since less tool wear then occurs. The separate finish-turning tool, having less wear than the roughing tool, will frequently produce a better surface finish. Sometimes special tools, such as shown in Fig. 7-16, are used for the finish-turning operation. The trial-cut procedure is used to obtain the finished diameter on the workpiece. When finish turning, great care must be exercised in the performance of this procedure so that the work diameter is not cut undersize. The final finishing cut should not be taken until the lathe operator is certain that the cutting tool is set to cut the work to the desired diameter.

The cutting speed for finish turning can usually be faster than for rough turning because of the decrease in the depth of cut; however, not all lathes can be adjusted to obtain this speed differential. Cemented-carbide cutting tools should be operated at a cutting speed that is fast enough to produce a good surface finish. When these cutting tools are operated above a certain critical cutting-speed range, an excellent surface finish can easily be obtained. This critical cutting-speed range is dependent on the grade of carbide used and the nature of the work material. In general, it is in the range of 150 to 200 fpm. When slower cutting speeds must be used, the surface finish obtained can be improved by using a good cutting fluid and by using a cutting edge that is in good condition. This usually means that the cutting edge must be replaced before excessive wear has taken place.

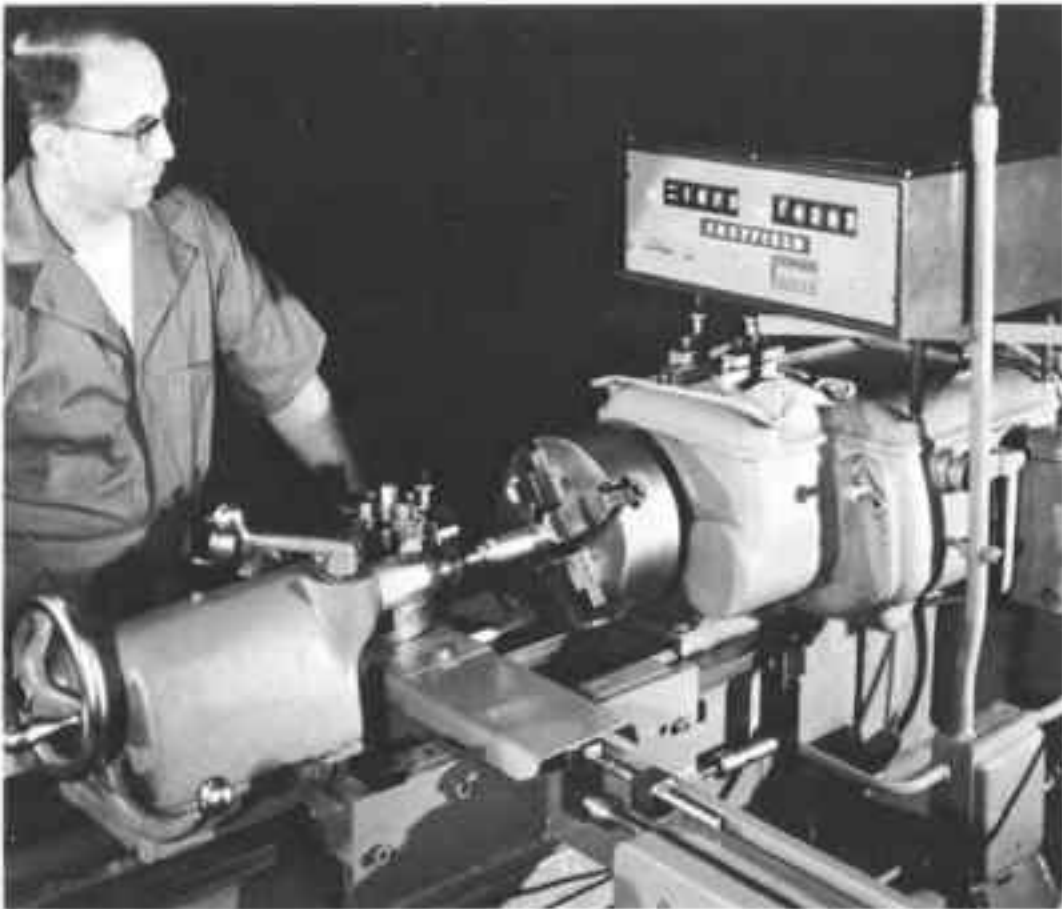
Sometimes, filing and polishing are used to obtain a smooth finish on parts that have been turned in a lathe. The file that is generally used for lathe work is a single-cut, bastard file of "mill" section. The amount of metal that is left for filing should be kept to a minimum. It is difficult to file a part round, in a lathe. In general, filing should be avoided except to "break" sharp corners of shoulders. An abrasive cloth may be used to obtain a smooth surface by pressing it against the rotating workpiece. A rather smooth finish can be obtained by using the finer grade of abrasive cloth. Polishing straps are sometimes used when polishing large diameters. A polishing strap is shown in Fig. 8-21. The two ends of the polishing strap are held together with a piece of leather belting.

Figure 8-22 illustrates a digital readout system that can be adapted to engine lathes. This system reduces the time and effort required to set the cutting tool in order to turn the work to the required diameter and length. The cutting tool is set against the end face, and the digital display for the longitudinal feed is set to zero. A short trial cut is taken and the diameter is measured with a micrometer caliper. The digital display for the cross slide is then set to read the diameter of the trial cut. From then on, the position of the point of the cutting tool—with respect to length and diameter—will appear on the digital display. Thus, the lathe operator can set the cutting tool to cut the length and diameter wanted, by observ-



Courtesy of LeBlond Machine Tool Co.

Fig. 8-21. Turning and polishing large-diameter workpieces. Note the polishing straps used to polish large diameters on the lathe at the left. The abrasive cloth is enclosed in the device that looks like a large nutcracker.



Courtesy of The Bendix Corporation, Automation and Measurement Division
Fig. 8-22. Engine lathe equipped with a digital readout.

ing the dimensions displayed on the digital display without taking any further trial cuts. For accurate work, of course, roughing cuts should be followed by a light finishing cut to minimize the effect of deflections caused by the heavy roughing cuts.

Shoulder Turning

There are three steps in shoulder turning:

1. Locating the shoulder on the work
2. Turning the smaller diameter to size
3. Facing the shoulder to length.

The length of the shoulder can be laid out using a hermaphrodite caliper as shown in Fig. 8-20. A scratch, or mark, made with the point of the cutting tool can also be used to mark the location of the shoulder. Often, the location of the shoulder is measured as the cut progresses, using a steel rule or a hermaphrodite caliper. A groove cut with a necking tool, A, Fig. 8-23, can serve to locate a shoulder.

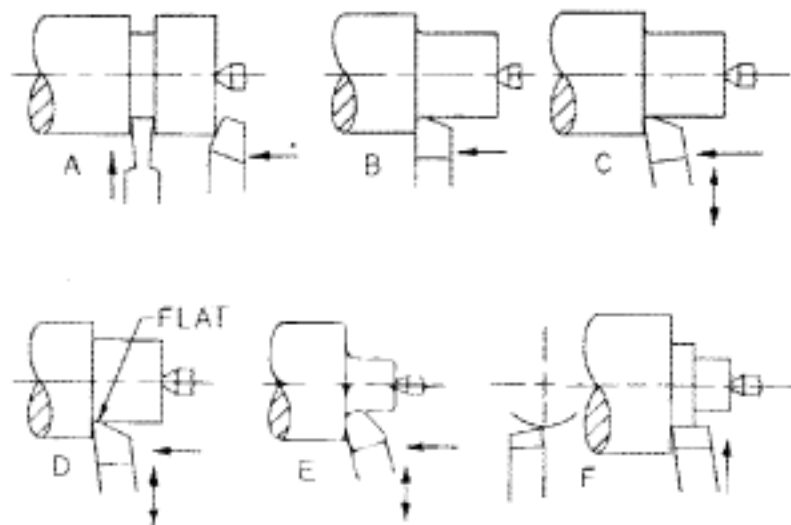


Fig. 8-23. Procedures for turning shoulders.

Several methods of turning shoulders are illustrated in Fig. 8-23. At A, a groove is first cut with a necking tool in order to establish the length of the shoulder. The small diameter is then cut to size by one or more cuts with the turning tool. When this method is used, all of the required shoulders are "blocked out," and the adjacent cylindrical surfaces are cut to size. This method does have a disadvantage, in that two tools need to be used. On slender workpieces a very slow feed is necessary to cut the grooves in order to avoid the occurrence of chatter. A faster, and even simpler method of cutting a shoulder is shown at B. The cutting tool is positioned to have a zero-degree lead angle. The shoulder is formed by the cutting edge when turning the smaller diameter. Shoulders turned in this manner are not necessarily perfectly square. When a shoulder must be square the method illustrated at C, is recommended. Usually the shoulder is first roughed out using either of the two previous procedures. The tool is then reset with the nose leading, as shown at C. It is set to cut the small diameter to size, using the trial-cut procedure. The micrometer cross feed dial reading used to take this cut, is noted. When the tool approaches the shoulder the automatic-power longitudinal feed is disengaged and the nose of the tool is fed by hand, longitudinally, into the shoulder. When it has penetrated the shoulder enough to take a light cut, it is fed out using the cross slide. The length of the shoulder is then measured, and the tool is moved longitudinally, just enough to remove the excess metal from the shoulder. It is then fed into the work with the cross slide. When it approaches the corner of the shoulder it is carefully fed by hand, until the reading of the micrometer cross-feed dial is the same as was previously noted. In this manner the tool will not undercut the corner and a smooth blend can be obtained between the shoulder and the previously turned diameter. It is sometimes an advantage to turn the compound rest 90 degrees, for this operation. In this position, the com-

pound-rest slide will move parallel to the longitudinal feed, and it can be used to set the depth of cut for the final finishing cut on the shoulder. Very accurate shoulder lengths can be obtained using this method of setting the tool. A micrometer carriage stop, Fig. 6-23, if available, can also be used to obtain accurate shoulder lengths. This same procedure can be used to cut a shoulder having a sharp corner, as shown at D, Fig. 8-23, or to cut a radius in the corner, as shown at E. The tool, at D, must be ground with a flat on the end-cutting edge in order to finish turn the small diameter, otherwise it will cut threadlike grooves on this surface. Very large radii are cut using a large forming tool that is similar to the tool at E. The corner is first blocked out using a turning tool, such as shown at B. It is sometimes faster to block out the corner with a facing tool, as shown at F. The corner radius is finished by touching the finished surface at the small diameter without penetrating this diameter with the radius-forming tool. This can be done if care is exercised. The reading of the micrometer cross-feed dial is noted when the radius forming tool is in this position. The radius is then cut, as shown at E, with care being taken at all times not to move the micrometer cross-feed dial beyond the setting noted above. A heavy-duty cutting oil, or white lead mixed with oil to a thin consistency, should be used as a lubricant when performing this operation in order to obtain a good surface finish in the radius. The cutting speed should also be greatly reduced when using the large radius form tool.

Mandrels

A mandrel is a work-holding device that is pressed into a finished machined hole of a workpiece in order to rotate the workpiece accurately about the axis of the hole, while the mandrel is held between the lathe centers. Standard mandrels are made from hardened and ground steel. They have a taper of .006 inch per foot. The size of standard mandrels is always marked on the large end. A thin film of oil should be placed on the mandrel before it is inserted into the hole. The small end of the mandrel should be inserted into the hole and the mandrel pressed in until it is tight. If hammer blows are used for this purpose, the end of the mandrel should be protected with a piece of soft metal or hard wood. The best practice is to insert the mandrel into the hole with an arbor press.

Two examples of the application of mandrels are shown in Fig. 8-24. In the upper view, the outside diameter of the bushing is turned to size and is concentric with the bore that has previously been finished to size. In the lower view, the small cast-iron wheel, having a finished hole through the hub, is having the outside diameter turned concentric and the sides faced perpendicular to the axis of the hole. If a comparatively large pulley is mounted on the mandrel, it must be driven directly by pins attached to a faceplate and engaging the pulley arms in order to prevent the pulley from turning on the mandrel when taking a cut.

Some special mandrels are shown in Fig. 8-25. A nut mandrel is shown

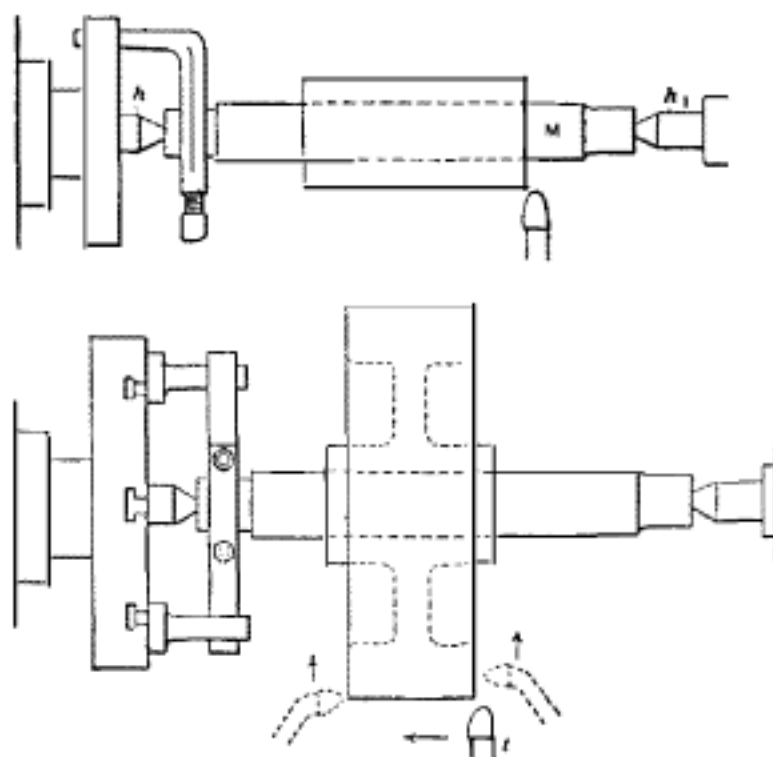


Fig. 8-24. (Upper Diagram) Bushing mounted on a mandrel for turning. (Lower Diagram) Turning a pulley held on a mandrel.

at A. Often, this type of mandrel is made by chucking a piece of scrap stock and turning it to the size required to fit the threads on the work. Thus, the threads are concentric with the rotation of the lathe spindle and any thread mounted on this mandrel will likewise rotate concentrically with the lathe. Nuts and other threaded workpieces can have surfaces turned concentrically, and faces turned perpendicularly, with respect to the threads that are held by the nut mandrel. Gang mandrels, shown at B, can be used to hold two or more parts between centers in a lathe. Cone mandrels can be used to hold large diameters. A cone mandrel can be seen at C. Expanding mandrels, D, consist of a slotted sleeve that has a tapered hole which fits onto a tapered shaft. When the shaft is forced onto the sleeve, the taper forces the sleeve to expand and to tighten inside the hole of the workpiece. An expanding mandrel is shown in Fig. 8-26.

Steady Rests and Follower Rests

One of the most useful lathe accessories is the steady rest shown in Fig. 8-27. The purpose of the steady rest is to support long, slender parts, in a lathe. The additional support given by the steady rest reduces the deflection of the workpiece caused by cutting forces; thereby, the accuracy of the work is improved and the tendency for chatter to occur is reduced. On very long and slender workpieces a steady rest is sometimes required just to keep the workpiece from sagging under its own weight. The steady rest may be used to support workpieces that are held between

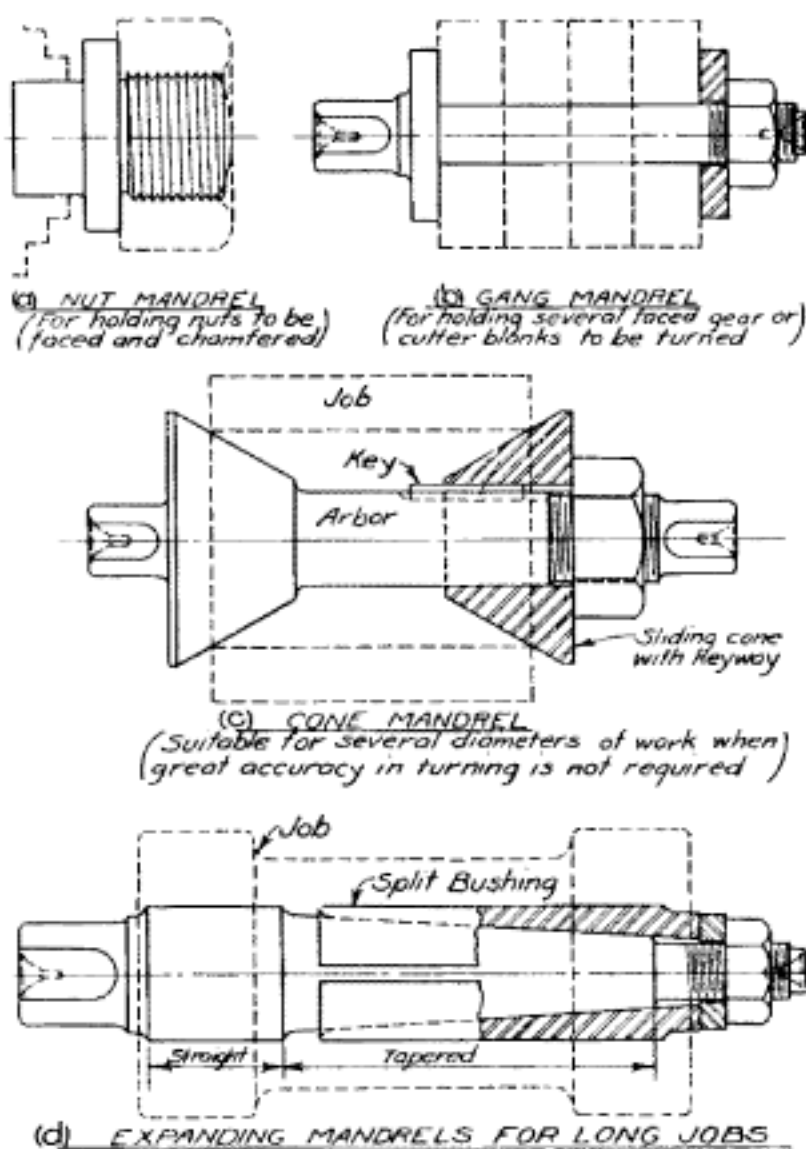


Fig. 8-25. (a.) Nut mandrel; (b.) Gang mandrel; (c.) Cone mandrel; (d.) Expanding mandrel.

centers (Fig. 8-27), with one end held in a chuck and the other on the tailstock center (Fig. 8-28). In order to leave the end face free for machining, one end is held in a chuck while the other end is supported only by the steady rest (see Figs. 8-9 and 6-24).

The steady rest is composed of a frame containing three movable jaws which form a three-point bearing. The frame is hinged on one side to allow the upper half to swing back for inserting or removing the work. The base of the frame fits on the ways of the lathe bed and is secured to the bed with a clamp. Before applying the steady rest, it may be necessary to turn a true surface, or spot, on the workpiece in order to provide a bearing surface for the jaws of the steady rest. The spot is turned by a tool having a small nose radius and a 0-degree lead angle. The spot is



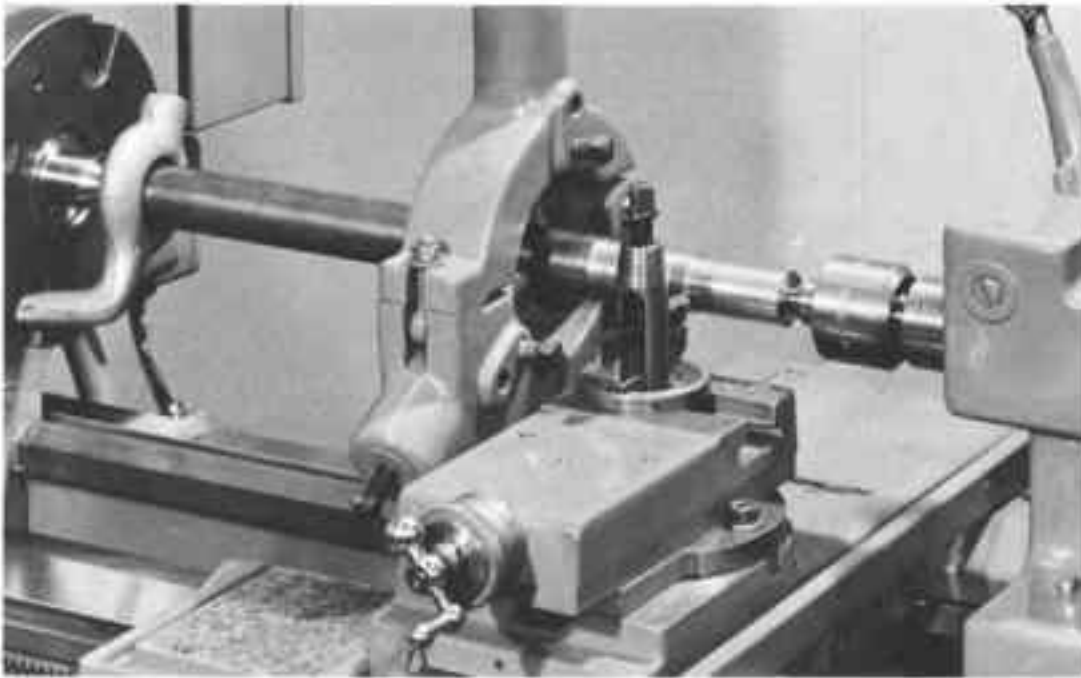
Courtesy of The Western Tool & Mfg. Co.

Fig. 8-26. An expanding mandrel used to hold a bushing while turning a short taper using the compound-rest.

sometimes polished with an abrasive cloth. The jaws should be carefully adjusted against the workpiece. For very accurate work, a dial test indicator is positioned on the workpiece 180 degrees opposite one of the lower jaws. This jaw is brought against the work until the indicator shows a deflection of a few-thousandths-of-an-inch. The jaw is then carefully backed off until the indicator returns to its original reading. The second lower jaw is adjusted by the same procedure. The top jaw is then brought down on the work and adjusted so that the work will turn freely, but without play.

The surface against which the jaws are brought to bear should be kept well oiled. If the jaws are required to bear against a finished surface, this surface should be protected by placing a strip of leather belting, or a strip of abrasive cloth (with the abrasive side out) between the jaws and the work. The steady rest must be located where it will not interfere with the turning tool. In some cases this may require the steady rest to be moved to another position after turning one section.

Sometimes, a long cylindrical surface must be turned on a workpiece so that there is no room to place the steady rest without interfering with the cut. In such instances a follower rest, Fig. 8-29, must be used to support long, slender workpieces against deflections. The follower rest is similar to the steady rest except that it has only two jaws and it is



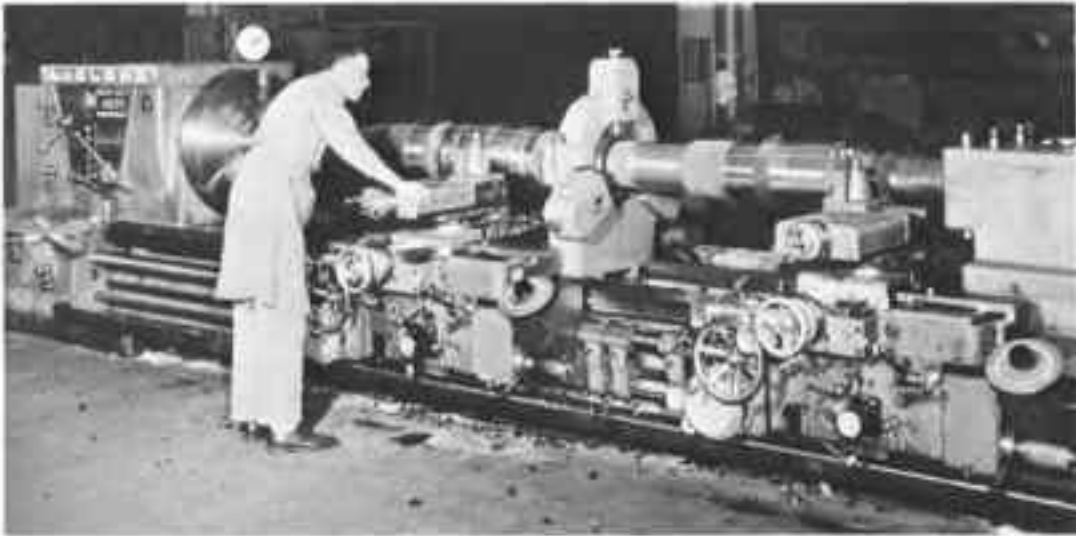
Courtesy of LeBlond Machine Tool Co.

Fig 8-27. Steadyrest used to support a slender part that is held between centers.

attached to the carriage. The jaws are adjusted to the finished surface after a short cut has been taken. The follower rest then moves along behind the cutting tool for the remaining length of the cut.

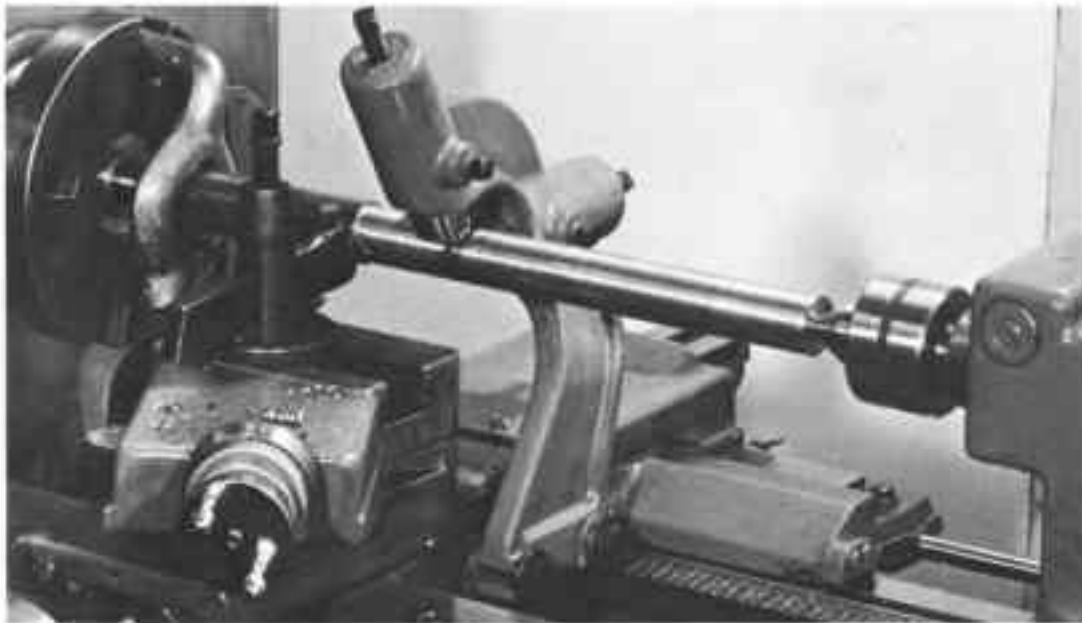
Knurling

A knurled surface is a raised surface, usually with a diamond-shape pattern, which is formed to produce an ornamental effect or to provide a nonslip or gripping surface. This pattern is formed on the surface by a knurling tool such as is shown in Fig. 8-30. This tool has three pairs of rolls which can be made to produce a fine, medium, or a coarse-pitch knurled surface. The rolls, or knurls, are made from hardened steel and each knurl has a series of teeth or ridges that incline to the right on one knurl and to the left on the other. When the two knurls are pressed into the work as it slowly rotates, one knurl forms a series of left-hand ridges and the other a series of right-hand ridges which cross, to form the diamond-shape pattern. The knurled surface is formed by the high local pressures produced by the knurls which cause the metal on the surface of the work to flow, or extrude, in an action which is similar to center punching. Knurled surfaces with a single groove are sometimes made, although the diamond pattern is the most common on machine parts. Several different kinds of knurling tools are also used. Some knurling tools are designed to reduce the load, which can be very high, on the cross-feed screw and nut.



Courtesy of LeBlond Machine Tool Co.

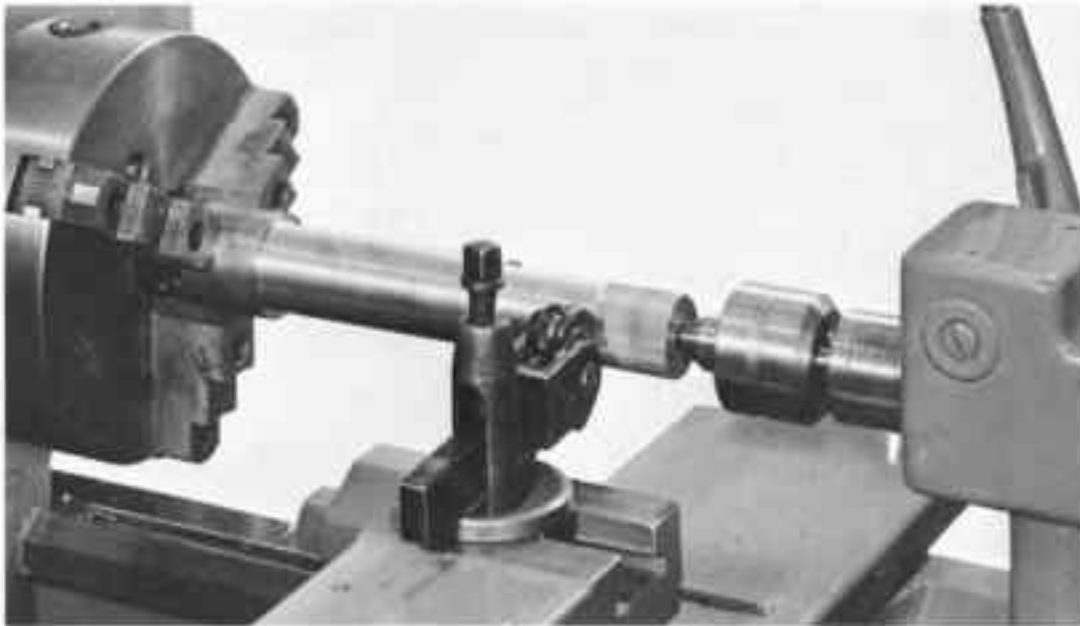
Fig. 8-28. Large steadyrest used to support long workpiece held in a chuck and between centers on a lathe having two carriages.



Courtesy of LeBlond Machine Tool Co.

Fig. 8-29. Follower rest used to support slender workpieces provides support for cutting entire length without stopping.

The knurling operation is performed by clamping a knurling tool in the toolholder and aligning it so that the knurls are parallel to the surface of the workpiece. The lathe should be set to operate at a very slow speed and a plentiful supply of lubricating oil should be placed on the surface



Courtesy of LeBlond Machine Tool Co.

Fig 8-30. Knurling to provide a slip-free surface.

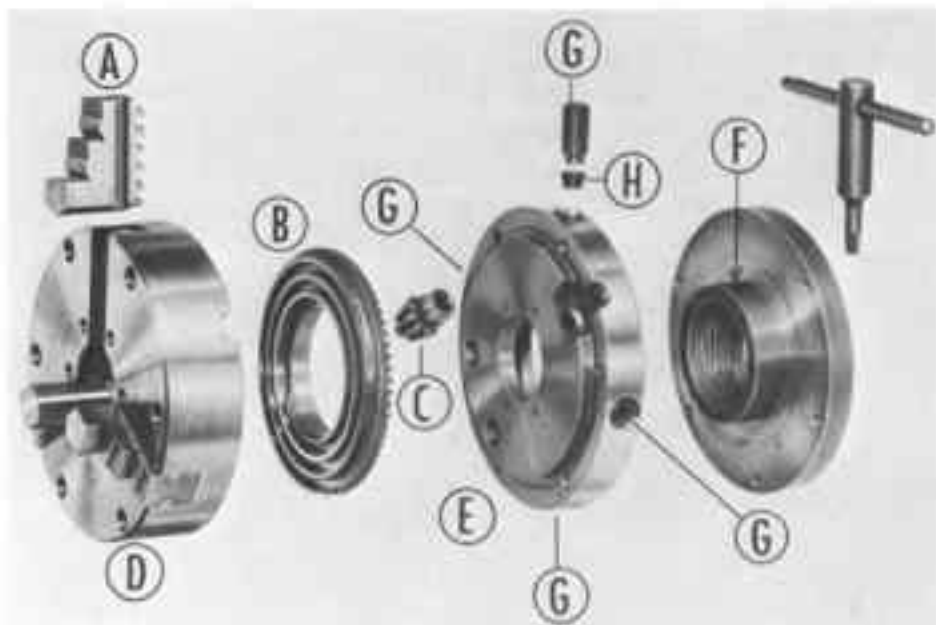
to be knurled. The knurling operation should be started, if possible, with about one-third or one-half of the knurl in contact with the surface to be knurled. The knurl is then fed inward, using the cross feed, until a well-formed knurled surface is visible on the surface of the work. If a fine knurl is being made it may be desirable, at this point, to feed the knurl into the full depth. On harder materials and coarse knurl patterns, the initial feed should not be to the full depth unless the width of the knurled surface on the work is less than the knurl rolls, in order to prevent an excessive load from occurring on the cross-feed screw and nut. If the surface to be knurled is wider than the knurls, the power longitudinal feed of the lathe should be engaged and the knurling tool is traversed back and forth, feeding it inward before each passage, until the full depth is reached. The knurls should never be allowed to lose contact with the work until the job is finished. The correct depth is obtained by observing the pattern of the knurl on the part.

Chucking Work

There are many different designs and types of chucks used in a machine shop. For example, there are drill chucks for holding straight-shank twist drills, magnetic chucks, vacuum chucks, etc. The four-jaw independent chuck, the three-jaw universal chuck, and collet chucks are commonly used in the engine lathe to clamp workpieces and hold them in alignment for the performance of chucking operations which includes turning, facing, thread cutting, drilling, boring, reaming, and cut off. Short workpieces are held only by the chuck, while longer ones are given additional support by the tailstock center or by a steady rest.

Mounting the Workpiece in a Three-Jaw Universal Chuck

A detailed view of a three-jaw chuck is shown in Fig. 9-1. The workpiece is placed in the chuck jaws and the jaws are closed by turning bevel pinion *C*, with the chuck key. The bevel pinion engages the gear teeth on the back of the scroll *B*, which turns inside of the chuck body *D*. The



Courtesy of the Buck Tool Company

Fig. 9-1. Details of a 3-jaw, Adjust-Tru, universal scroll chuck.

chuck jaws *A*, engage the threads on the scroll. When it rotates, the jaws move in the slides that are machined onto the face of the chuck body. Each slide and jaw is numbered, for each jaw must be assembled into its corresponding slide by always starting with jaw number 1, then number 2, and finally, number 3. The back plate, or mounting plate, *F*, contains the mounting used to hold the chuck on the lathe spindle nose. In this illustration a threaded mounting is shown.

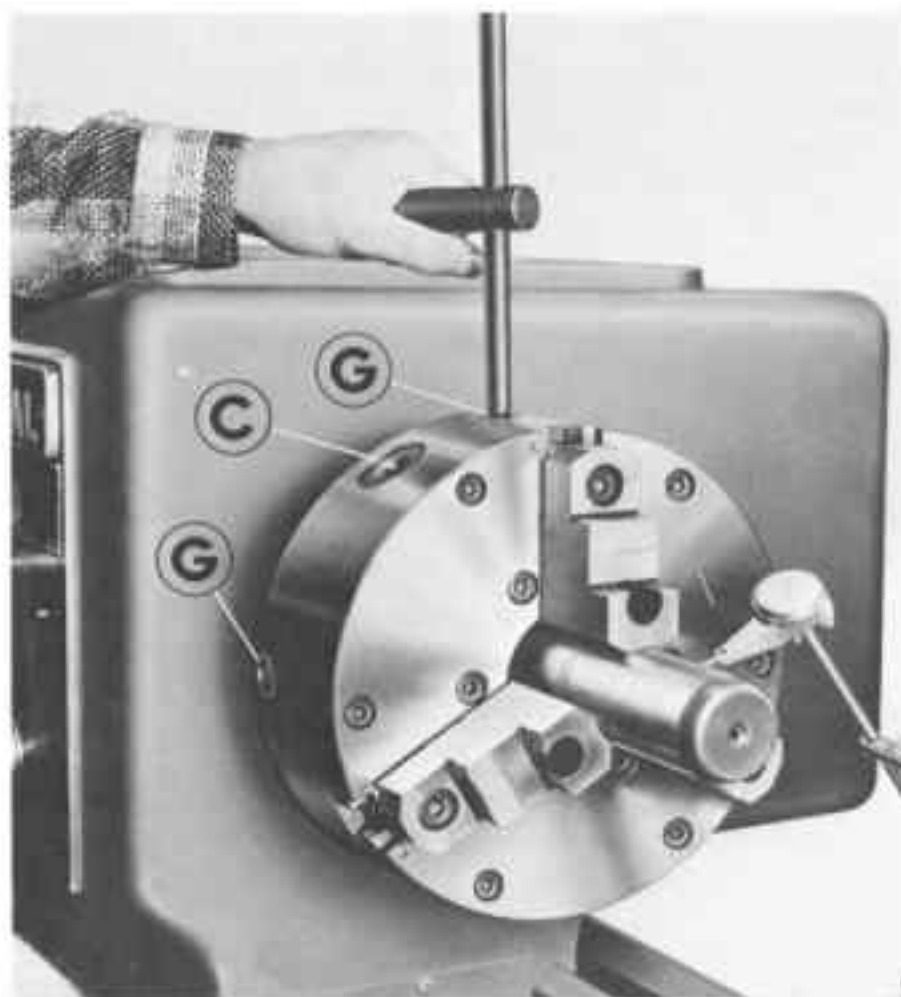
Because of the clearances required to allow this mechanism to operate without binding and because of wear, three-jaw chucks do not always hold the workpiece accurately in terms of a few thousandths of an inch. In many cases this is good enough; however, when greater accuracy is required the jaws must be ground true while in the chuck body; a difficult procedure to perform. This can be avoided with the chuck shown in Fig. 9-1. A fine adjustment of the chuck is provided that is independent of the chuck jaws. Four opposite socket-head screws, *G*, work on pressure plugs, *H*, that contact surface, *F*, of the back plate in order to shift the complete chuck and work into alignment. The work is placed in the chuck, Fig. 9-2, and tightened with the chuck key by turning pinion *C*. A dial test indicator is used to check the concentricity of the work. Any eccentricity can be rapidly removed by adjusting opposite pairs of socket-head screws, *G*. No further adjustment of the socket screws is required when chucking duplicate parts.

The basic methods of holding the workpiece in both three- and four-jaw chucks are illustrated in Fig. 9-3. Larger workpieces are held on the outside diameter by reversing the chuck jaws, as shown in view B. To reverse the jaws on a three-jaw universal scroll-type chuck requires a separate set of jaws, or the two-piece type jaws which are keyed and bolted together. Two-piece type jaws are shown on the chuck in Fig. 9-2. The jaws on four-jaw independent chucks can be inserted into the chuck body in either of the two positions shown in Fig. 9-3.

Mounting the Work in a Four-Jaw Independent Chuck

The four-jaw independent chuck is a very versatile chuck with a wide range of applications on engine lathe work. Each jaw works independently by means of a separate screw turned by the chuck key. Parts can be held firmly and, if required, very accurately, in this chuck. The accuracy with which a part can be held is dependent entirely on the condition of the surface on the work that is used to test the concentricity. A rough-cast surface that is not round, obviously cannot be chucked to "thousandths," because no reference surface exists to prove the truth of the position of the part. Cold-drawn and machined surfaces, on the other hand, can be indicated into position within "thousandths," using a dial test indicator as shown in Fig. 9-5. When the procedure is understood, four jaw chucks can usually be adjusted accurately within minutes.

The chuck jaws should be opened just far enough to admit the workpiece by observing the position of each jaw in relation to concentric



Courtesy of the Buck Tool Company

Fig. 9-2. Making a fine adjustment to remove eccentricity on workpiece on a 3-jaw, Adjust-Tru, universal scroll chuck.

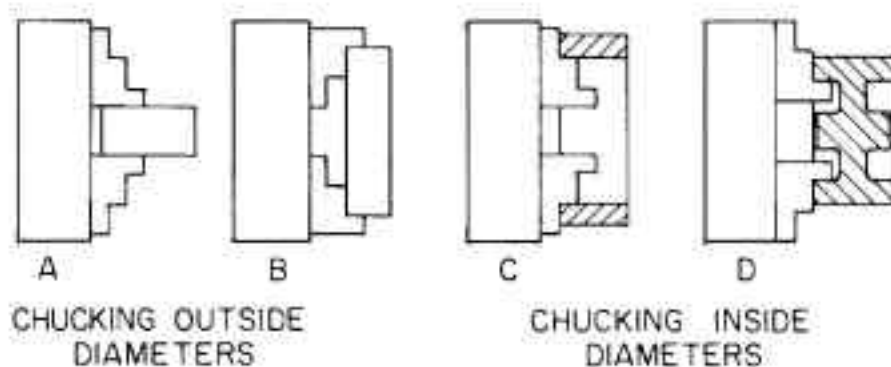


Fig. 9-3. Schematic drawing showing the basic methods of holding the workpiece in both three-jaw independent chucks and four-jaw universal chucks. Only two chuck jaws are shown for illustration purposes. Any set of steps on the jaws can be used to hold parts.

grooves that are machined into the face of the body. The jaws are then closed to hold the work lightly. The shank end of a tool holder, or a piece of metal that is resting in the tool post, is placed against the work—opposite one of the chuck jaws—as shown at A, Fig. 9-4. The tool holder must not be clamped tightly in the tool post as it must be able to be pushed aside with little effort. The chuck is then turned manually. The high spot on the work pushes the shank of the tool away resulting in an opening, D (shown in B, Fig. 9-4). This occurs when the low spot is opposite the shank. All effort should first be made to adjust the jaw with

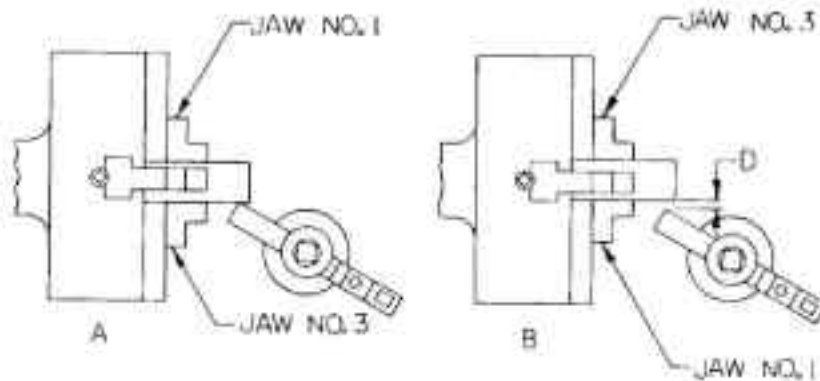


Fig. 9-4. Procedure for adjusting workpiece in a 4-jaw independent chuck.

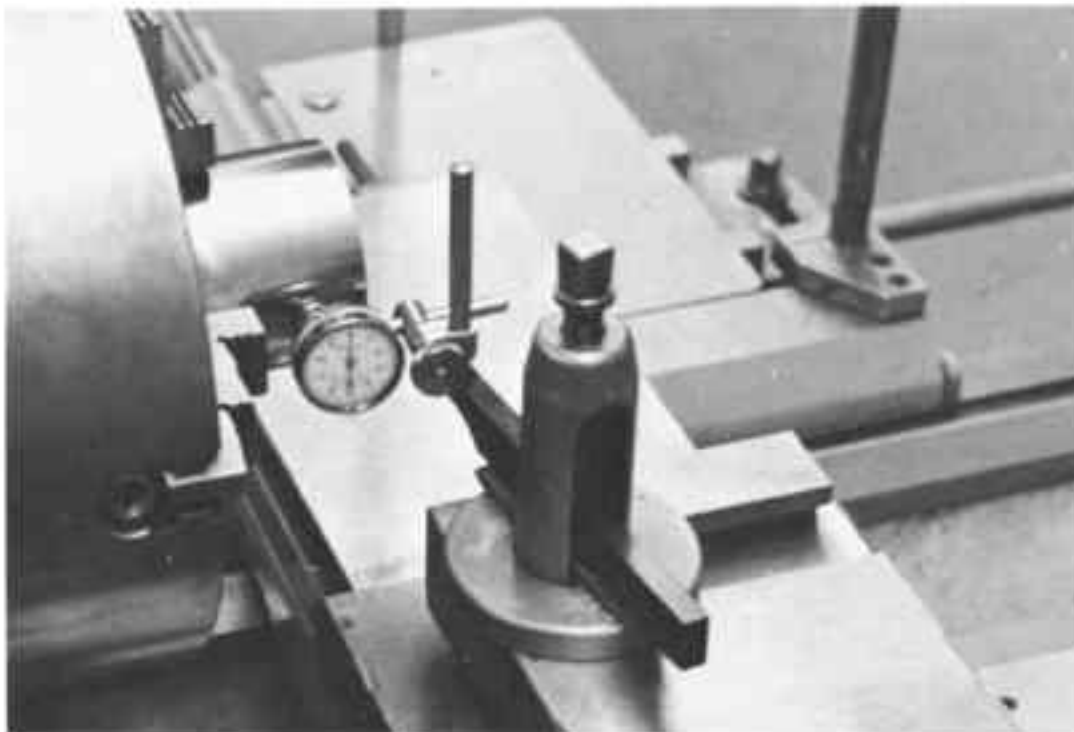
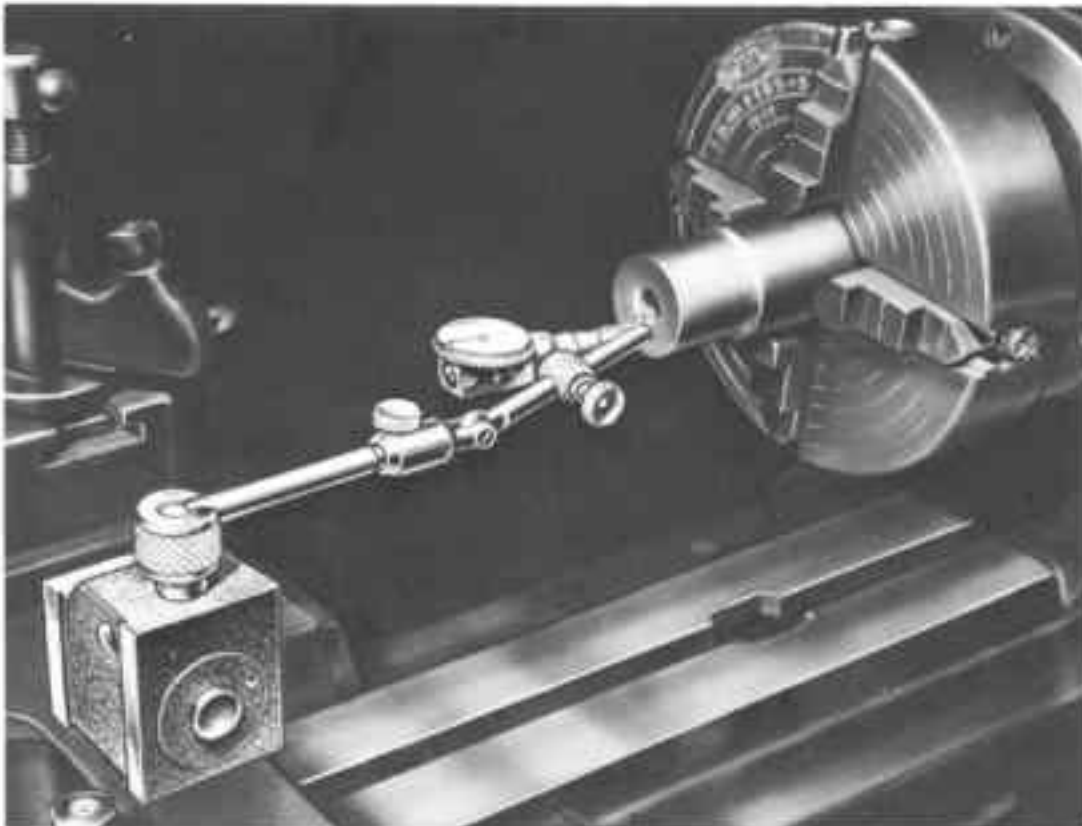


Fig. 9-5. Using a dial test indicator to test the concentricity of a workpiece held in a 4-jaw independent chuck.

the largest opening between the shank and the work, and its opposing jaw, until the openings opposite these two jaws are equal. The other two jaws then are adjusted likewise. Finally, the fine adjustment is made again, first by adjusting two opposing jaws, then the other pair of opposing jaws. When the reference surface used to check the concentricity is a finished surface, it should finally result in work that can be rotated without any light showing between this surface and the shank. For very accurate work a dial test indicator can be used in place of the tool shank in making the fine adjustment. (See Fig. 9-5.) The dial test indicator should not be used to check a rough surface. The contact point of the indicator should be used only against a finished surface. Sometimes a hole in a workpiece must be made to run true in a chuck. This is done by indicating inside the hole, as shown in Fig. 9-6. In this case the indicator is held by a magnetic base.

Some parts held in a chuck will tend to show endwise eccentricity or wobble in addition to radial eccentricity. In some cases this can be corrected by placing the work against the face of the chuck jaws or chuck body. When this cannot be done, the shank of the tool holder, or a piece of steel, is used against the face in a similar manner as for centering the



Courtesy of The L. S. Starrett Company

Fig. 9-6. Using a dial test indicator to test the concentricity of a hole located in a workpiece held in a 4-jaw chuck.

work radially. In this case the high spots are found which are then tapped back with light blows from a soft hammer. The fine adjustment can be checked by using a dial test indicator. The contact point of the indicator should not be in contact with the workpiece when it is being tapped into adjustment with the hammer, in order to avoid damage to the indicator mechanism. Very large four-jaw chucks have radial T-slots spaced between the chuck jaws. Studs which extend an equal distance from the face of the jaws can be fastened in the T-slots, and the parts being chucked can be positioned against the end of these studs, thereby obtaining endwise alignment.

The workpiece is sometimes deliberately mounted eccentrically in the four-jaw chuck in order to turn a surface which is eccentric with respect to another surface. (See Fig. 9-7.) The flexibility of the four-jaw chuck is clearly demonstrated in this operation. It is, perhaps, the most versatile and useful work-holding device for engine-lathe work.

Mounting Work in Collet Chucks

Spring collet chucks are highly accurate chucks. They should be used to clamp on finished surfaces that are accurate to within a few thou-



Courtesy of the Clamping Division of the Atlas Press Company

Fig. 9-7. Work held eccentrically in a 4-jaw independent chuck.

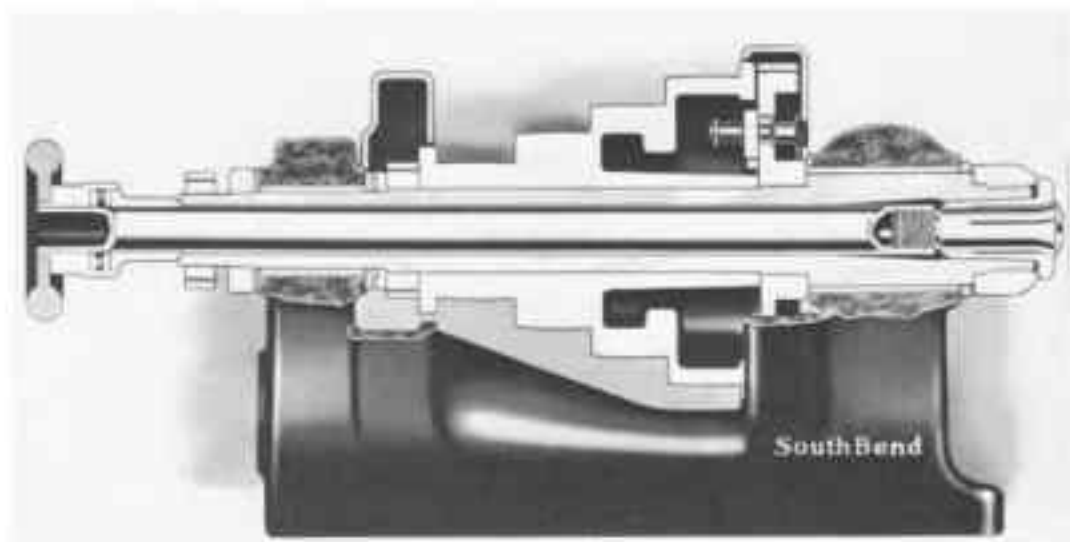
sandths of an inch of the size of the collet used. The spring collets exert an even and uniform pressure around the workpiece, which minimizes the distortion of thin-walled parts. Figure 9-8 shows a group of spring collets used to hold round, square, and hexagon-shaped workpieces. A draw-in, bar-type collet chuck is shown in Fig. 9-9. A special collet adapter is inserted in the spindle nose and the collet is inserted in the adapter. The draw-in bar is inserted in the rear of the spindle. When the draw-in bar is turned, it draws the collet into the spindle by the action of the threads. The taper on the adapter (corresponding to the taper on the collet) causes the collet to close as it is pulled into the spindle by the draw-in bar. All of the surfaces of the spindle nose, the adapter, and the collet must be clean since even the smallest particle of dirt or foreign substance will cause the workpiece positioned inside the collet to be held inaccurately. A spindle nose type collet chuck is shown in Fig. 9-10. This chuck can be obtained to fit onto either a threaded cam lock or a long-taper spindle nose. The collet is inserted in the front of the chuck and pulled back against the taper to permit closing the jaws by turning the handwheel. As it is not necessary to reach behind the spindle to close the chuck, a saving of time and effort results.

The diameter of the workpiece held in a spring type collet should be within approximately $\pm .005$ in. (± 0.13 mm) of the diameter for which the collet is made. Figure 9-11 illustrates what happens when the workpiece is too large or too small. The collet will not grip the workpiece firmly, except at the ends of the collet holding surfaces which, as a result,



Courtesy of Hardinge Brothers Inc.

Fig. 9-8. Spring collets for holding different work shapes.



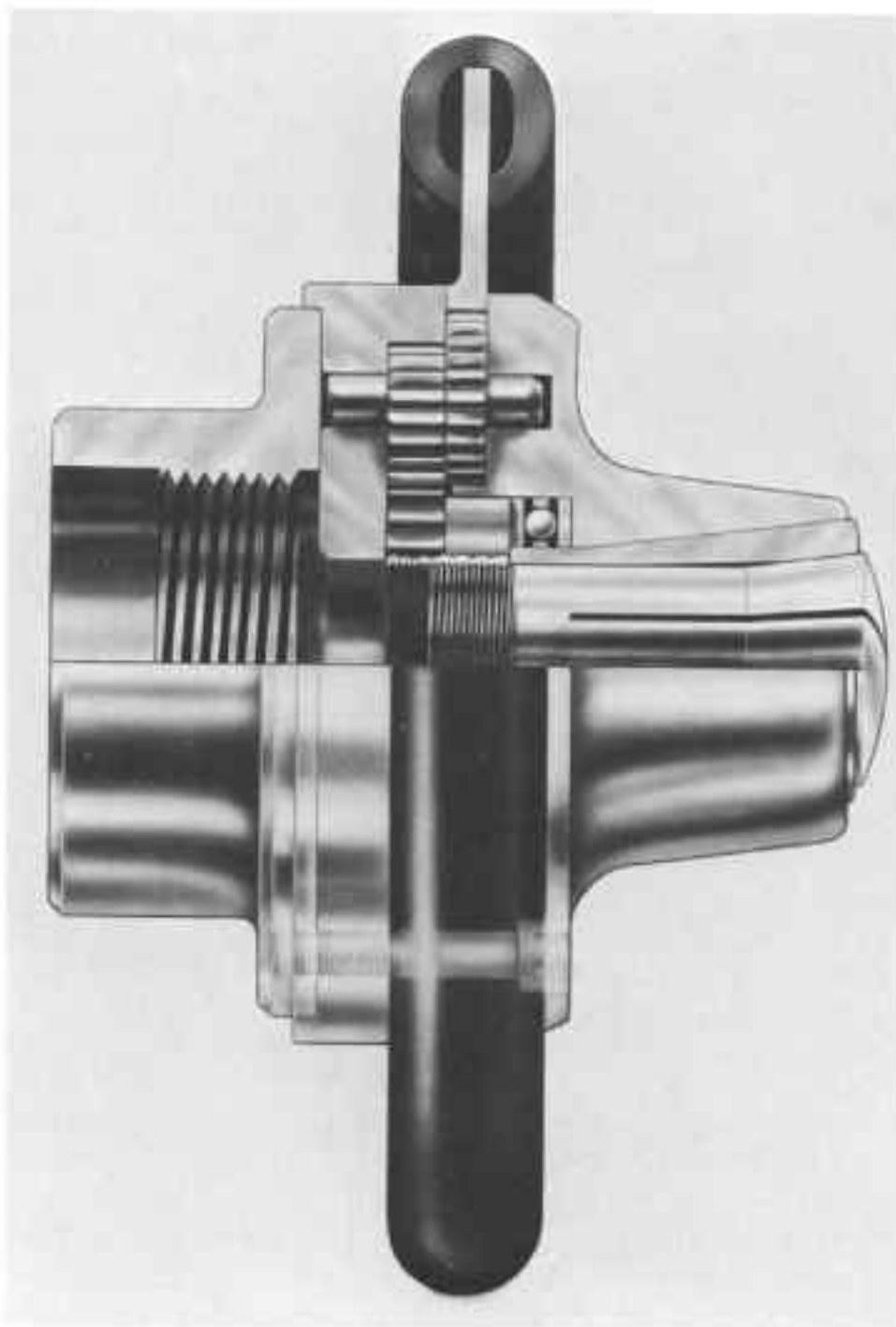
Courtesy of South Bend Lathe

Fig. 9-9. Draw-in bar type collet chuck.

may cause the machined surfaces to be inaccurate. The workpiece surface on which the collet grips should be a finished surface to obtain the accuracy and the maximum holding power for which the collet has been designed.

There are other types of collets which are used on engine lathes, although these collets are used primarily when a number of like parts are to be machined. One such collet is the step chuck, shown in Fig. 9-12. The step chuck consists of a closing ring, which fits on the lathe spindle, and a chuck or collet, which closes to hold the outside diameter of the workpiece. The chuck or collet is made of unhardened steel, which is bored out on the job to the diameter and depth required to fit the workpiece. Before boring the chuck, pins (see Fig. 9-12) are inserted in holes located in the slits. The pins limit the amount that the chuck will close when drawn tight in the closing ring. The chuck is then bored to the required diameter and depth; the diameter should be .0005 in. (0.013 mm) less than the size of the part to be held. After the chuck has been bored to size, the pins are removed and the workpiece can be held, as shown in Fig. 9-13. The part in this illustration is being counterbored using an ordinary turning tool; however, the side relief angle of the tool must be large enough to prevent the flank of the tool from rubbing against the bore. Some regular collets are made from a soft metal so that they can be bored to a size for which there is no standard collet. A soft collet is illustrated in the upper right hand view of Fig. 9-8, which shows the pins used when the collet is bored to size.

Another type of collet is the expanding collet, which is designed to hold



Courtesy of Hardinge Brothers Inc.

Fig. 9-10. Spindle-nose type collet chuck.

by expanding inside of a hole in the workpiece. A taper pin or a tapered expanding bolt is used to expand the collet. While these collets are generally round, they can also be made to other shapes such as square, hexagon, or threaded.

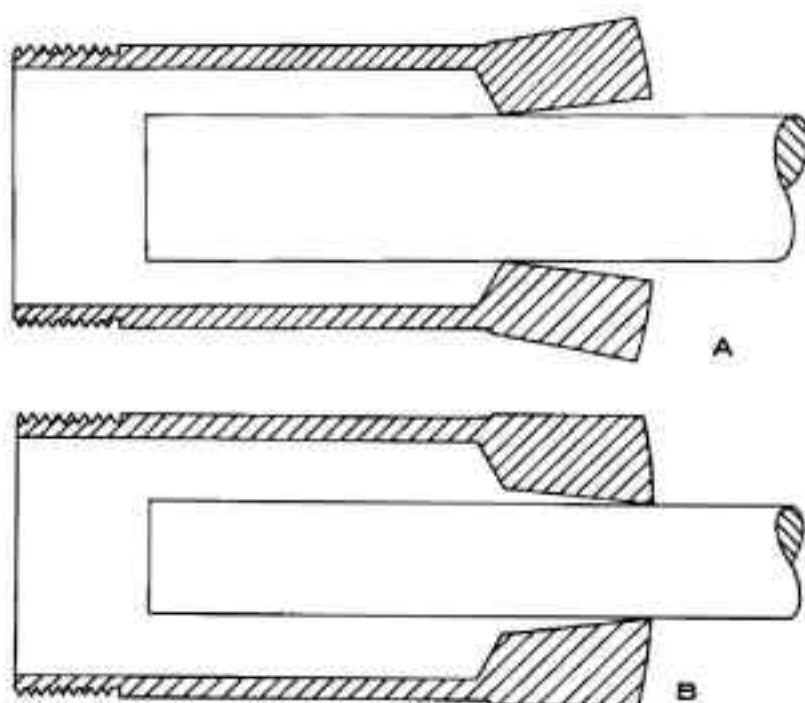


Fig. 9-11. Effect of holding oversize and undersize workpieces in a spring collet. A. Workpiece oversize; B. Workpiece undersize.



Courtesy of Hardinge Brothers Inc.

Fig. 9-12. Step chuck, showing closing ring and collet, which is usually called "the chuck."

Collet-Chuck Work

Collet chucks are used to hold finished surfaces only. This, however, can include raw cold-drawn or centerless ground bar stock. The bar stock

can be placed inside of the headstock spindle with enough material protruding outside of the collet to make the required part. The finished part is then cut off with a cut-off tool. Collet chucks are frequently used for second chucking operations. The part is first finish-turned on one end. It is then turned around and the finished end is held in the collet chuck while the other end is turned to size. A turning operation with the part held in a spindle-nose type collet chuck is shown in Fig. 9-14. Note that the compound rest is set at 90 degrees so that the compound-rest slide moves parallel to the axis of the lathe. This permits the micrometer dial of the compound-rest feed screw to be used to obtain an accurate distance between the shoulder and the end face of the workpiece.

In Fig. 9-15 a very long, small-diameter workpiece is being turned while held in a collet chuck. Turning this between centers would be difficult because the action of the cutting forces would tend to bend the part and the lathe dog would interfere with the cut. The first operation is to hold the workpiece inside of the spindle with just enough protruding from the collet to take a cut of about one or two inches in length. It is then drawn out of the collet and one end is held by a steady rest as shown. This end could be held in a tailstock center; however, on very small diameter work the center hole in the workpiece would be very small affording only a small bearing surface for the tailstock center. A follower rest is placed on the carriage which, in this case, leads the cutting tool. The follower



Courtesy of Hardinge Brothers Inc.

Fig. 9-13. Step chuck holding workpiece to perform turning, facing, boring, and counterboring operations.



Courtesy of Hardinge Brothers Inc.

Fig. 9-14. Turning and facing a part held in a spindle nose type collet.

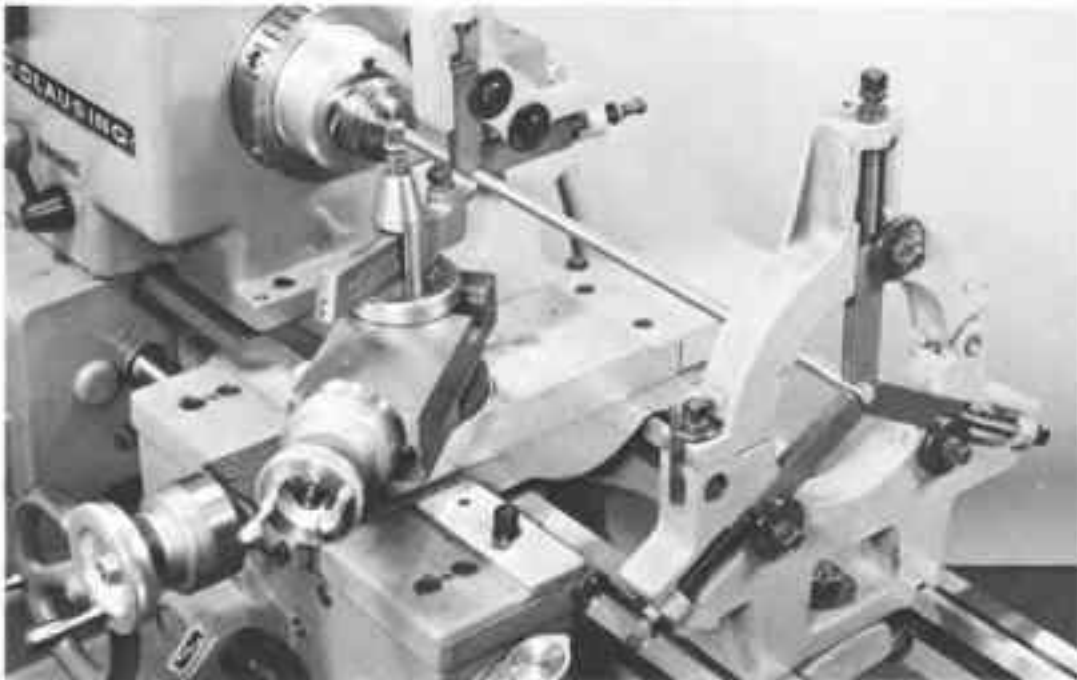
rest prevents the work from deflecting as the tool cuts the length of the part to size. This job could also be done by turning the length between the collet chuck and the steady rest to size, as shown, after which the two ends are cut off.

Chuckling for Work-Turning and Facing

Cylindrical surfaces can be turned with the work held in the chuck as shown in Fig. 9-16. The longitudinal feed causes the cutting tool to travel parallel to the axis of the lathe and as the work revolves, the tool will cut a true cylindrical surface. The trial-cut procedure is used to obtain the desired diameter on the workpiece. Long parts held in a chuck may deflect as a result of the cutting forces which cause the part to be tapered instead of cylindrical. This can be prevented by using a cutting tool having a small-nose radius and setting this tool to have a zero-degree lead angle.

Facing is the operation of turning a plane surface by feeding the cutting tool with the cross slide. A facing operation is illustrated in Fig. 9-17. The depth of cut for this operation can be obtained by moving the carriage longitudinally.

Greater accuracy in setting the depth of cut can be obtained by setting the compound-rest slide so that it will move parallel to the lathe axis as



Courtesy of the Clausung Division of the Atlas Press Company

Fig. 9-15. Turning a slender workpiece held in a collet chuck by a steady rest. Additional support is provided by a follower rest.

shown in Fig. 9-17. The depth of cut is set by moving the compound rest slide, using the micrometer dial on the feed screw to make accurate settings.

Drilling

The first step in drilling a hole on a lathe is to start the hole with a combination drill and countersink, or center drill, as seen in Fig. 9-18. The center drill is held in a drill chuck which is mounted in the tailstock spindle. The work is rotated by the headstock spindle and a high spindle-speed should be used to drill this starting hole. The hole is drilled by clamping the tailstock to the bed of the lathe and feeding the drill into the work manually by turning the tailstock handwheel. The center drill, being short and inflexible, will not deflect; but will start the hole true, without any eccentricity. The twist drill will be guided by the hole made by the center drill and, therefore, will also start true and without eccentricity.

After the starting hole has been made with the center drill, the hole is then drilled to the required size and depth with a twist drill. The spindle speed used to drill the hole should be appropriate both to the size of the drill and the material being cut. As a guide, the Table of Cutting Speeds (in Chapter 5) can be used. The spindle speed of the lathe is calculated from the cutting speed using Formula 5-1. The drill is fed into the work manually, by turning the tailstock handwheel.



Courtesy of the Clausing Division of the Atlas Press Company

Fig. 9-16. Turning a part that is held in a 3-jaw universal chuck.

Small twist drills (up to approximately one-half inch in diameter) can be held in a drill chuck which is mounted in the tailstock spindle, as shown in Fig. 9-19. The taper locates the drill on the axis of the lathe and it will hold the drill sufficiently firm to resist the drilling torque. The torque, encountered when larger drills are used, may be sufficiently large to cause the taper shank to slip in the taper hole of the tailstock spindle. This will result in scoring the inside of the taper hole of the spindle. This damage is frequently so severe that the tailstock spindle is ruined. It is not advisable, therefore, to place large, taper shank drills directly into the tailstock spindle unless some means are provided to prevent the drill from slipping or the step drill method is used to reduce the drill torque.

The tailstock spindle on some lathes is provided with a keyway, or slot, into which the tang of the twist drill fits. On lathes which are so equipped large-size taper-shank drills can be safely driven by the tang. Most engine lathes do not have a driving slot in the tailstock spindle, thus other means must be used to drive the larger-size twist drills. One method of doing this is shown in Fig. 9-20. A lathe dog is clamped onto the twist drill which is inserted in the tailstock spindle so that the tail of the dog rests on the compound rest. The drill should first be inserted in



Fig. 9-17. Taking a facing cut on a gear blank.

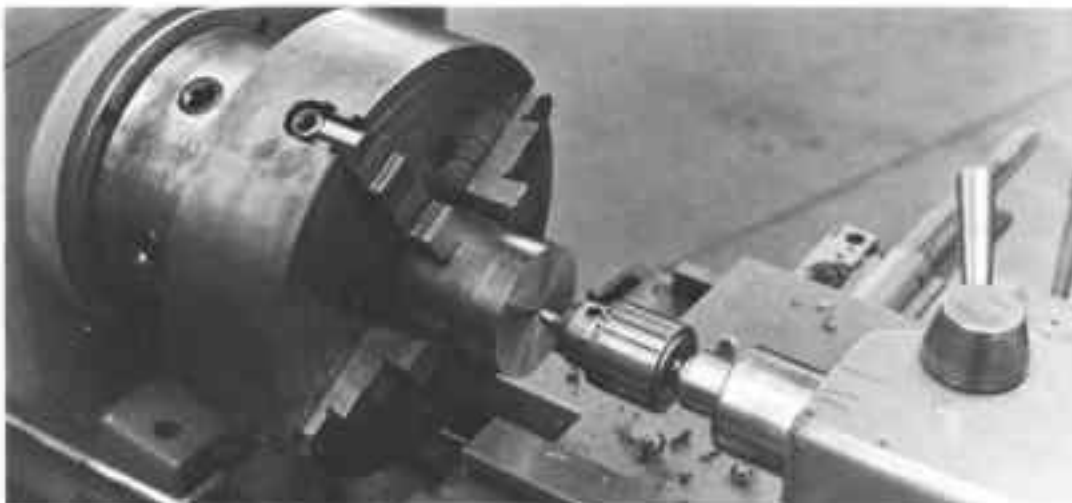


Fig. 9-18. Starting a drilled hole with a center drill.

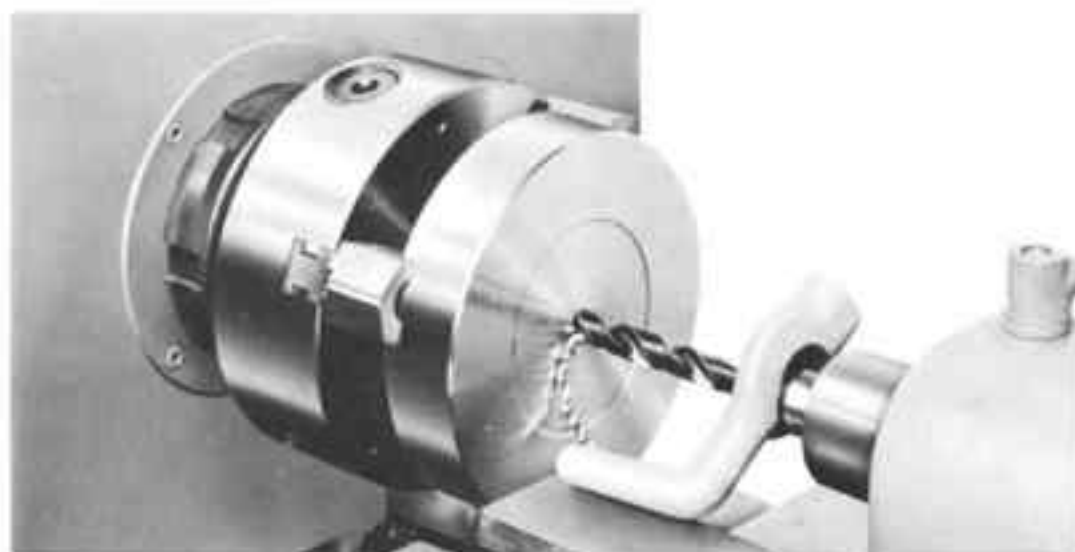
the spindle after which the dog is clamped on the drill so that the tail is against the compound rest. The tail of the dog prevents the drill from turning. Care must be exercised to make certain that the tail of the dog is against the compound rest at all times as the hole is being drilled.

Another device used in driving a large twist drill is called a drill holder,



Courtesy of LeBlond Machine Tool Co.

Fig. 9-19. Drilling a small-diameter hole with a drill held in a drill chuck which is mounted in the tailstock spindle.



Courtesy of LeBlond Machine Tool Co.

Fig. 9-20. Drilling a large-diameter hole with a taper shank twist drill. The lathe dog prevents the drill from slipping.

which holds the drill outside of the tailstock spindle (see Fig. 9-21). The taper shank of the twist drill fits into a matching taper inside the drill holder, which is supported by a tailstock center and the workpiece.

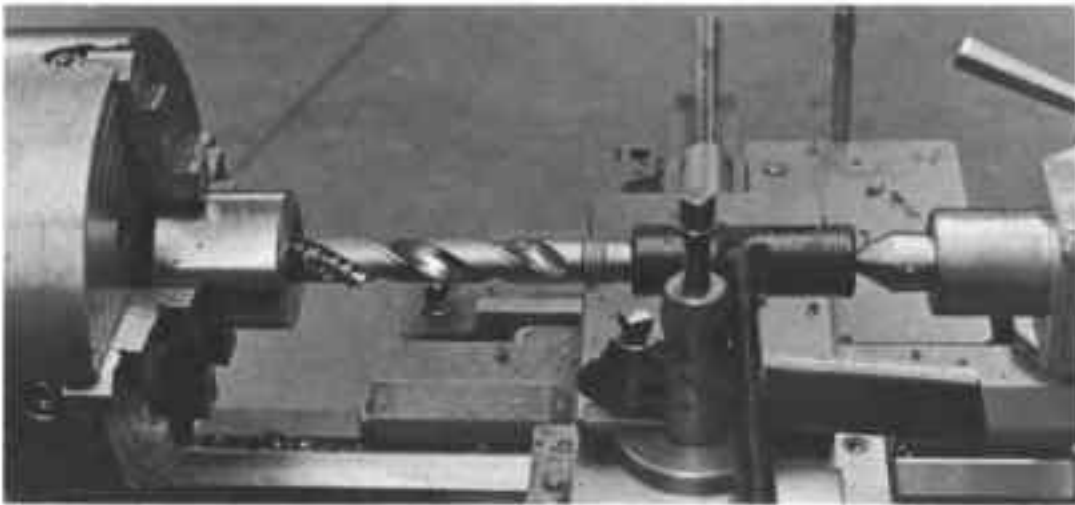
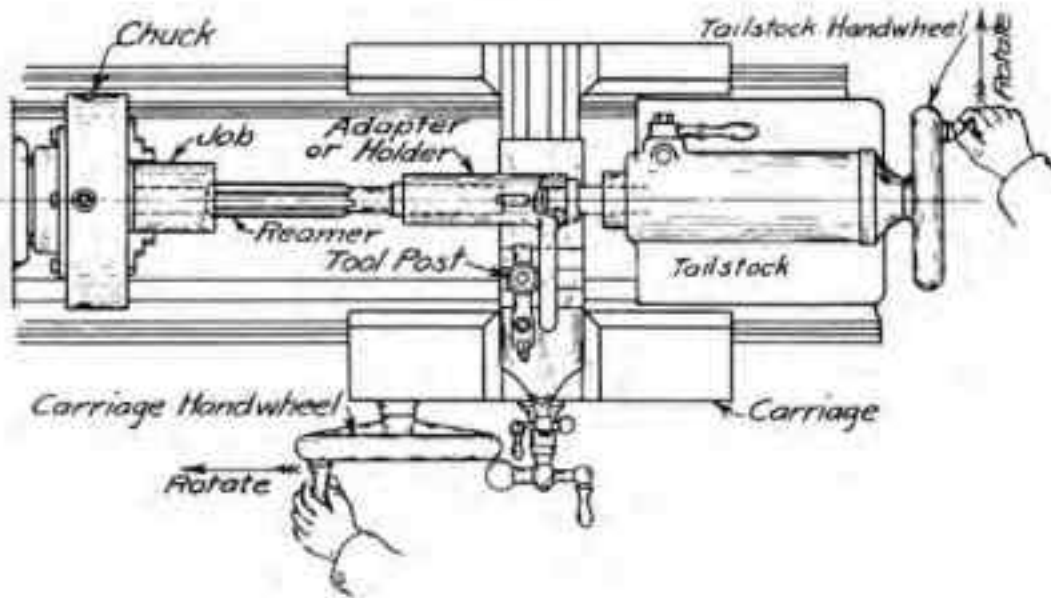


Fig. 9-21. Drilling a large-diameter hole in a lathe. The twist drill is held in a drill holder.

The arm of the drill holder is placed on the shank of the tool holder and against the back of the tool post. As the drill is fed through the workpiece by the tailstock spindle the arm of the drill holder pushes the carriage toward the headstock. The carriage thus provides a drag, or a constraint to the movement of the drill holder, thereby keeping it seated on the tailstock spindle at all times. This constraint is especially important in overcoming the tendency of the drill to pull into the work as it breaks through the far side of the workpiece.

Large holes can be safely drilled with taper-shank drills mounted directly in the tailstock spindle by using the "step drilling" procedure. A small hole, say, one-half inch in diameter, is drilled first. This is followed by enlarging the hole with successively larger-diameter drills until the required size is reached. In this way the torque load encountered by any one drill is not excessive, and the drill will not slip in the tailstock spindle. Another advantage of this method is that the thrust load is reduced which, in turn, reduces the load on the tailstock spindle nut and screw. This may be significant when drilling on a small lathe. The single disadvantage of employing this method is that more time will be required to drill the hole.

All of the methods of drilling a hole thus far described require the drill to be fed into the work manually by turning the tailstock handwheel. Although this is not objectionable on engine lathe work, the drill may be fed with the longitudinal power feed of the carriage using the setup shown in Fig. 9-22. The drill is held in a drill holder that is clamped to the face of a square turret type tool post having a "quick change" feature on one face. This feature permits tool holders for different types of tools to be quickly clamped or released. In making the setup, the drill holder is adjusted to the position where the drill is at the height of the lathe centers;



Courtesy of the Midwest Industrial Management Association

Fig. 9-23. Reaming a hole on a lathe. The reamer is held in a floating reamer holder.

fed through the hole slowly. On engine lathes the reamer is fed manually through the hole by using the tailstock spindle. Cast iron is usually reamed dry; however, a good grade of cutting oil should be used for reaming most other metals including all of the steels.

Boring

The objective of boring a hole in a lathe is:

1. To enlarge the hole
2. To machine the hole to the desired diameter
3. To accurately locate the position of the hole
4. To obtain a smooth surface finish in the hole.

The motion of the boring tool is parallel to the axis of the lathe when the carriage is moved in the longitudinal direction and the workpiece revolves about the axis of the lathe. When these two motions are combined to bore a hole, it will be concentric with the axis of rotation of the lathe. The position of the hole can be accurately located by holding the workpiece in the lathe so that the axis about which the hole is to be machined coincides with the axis of rotation of the lathe. When the boring operation is done in the same setup of the work that is used to turn and face it, practically perfect concentricity and perpendicularity can be achieved.

The setup for boring a hole on an engine lathe is shown in Fig. 9-24. The boring tool is held in a boring bar which is fed through the hole by the carriage. A typical boring tool is shown in Fig. 9-25. Variations of this



Courtesy of the Clausing Division of the Atlas Press Company

Fig. 9-24. Boring a hole on an engine lathe.

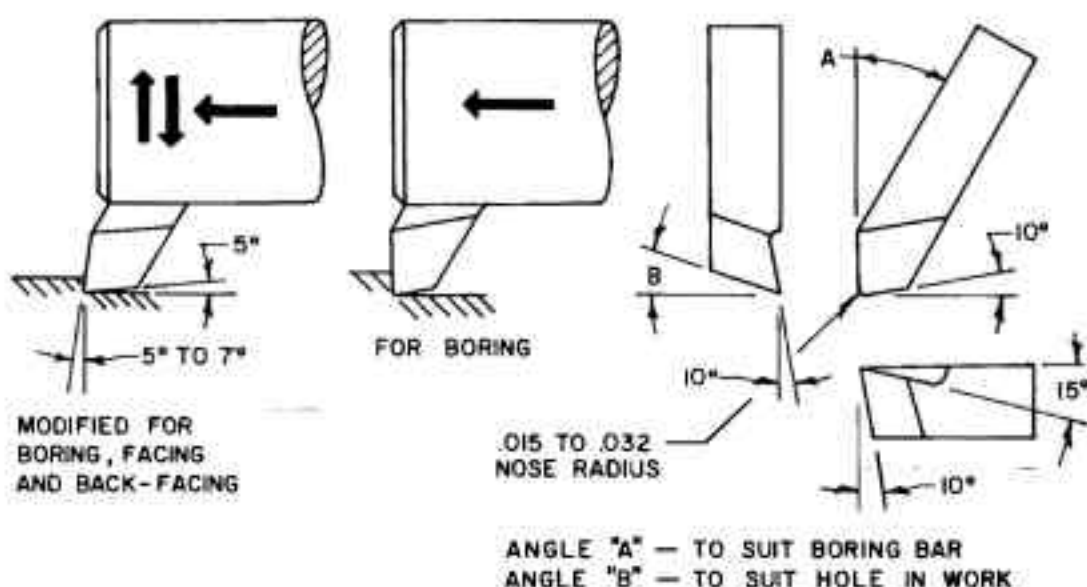


Fig. 9-25. Design of a typical high-speed steel boring tool. The design is typical, but variations are used to suit different conditions.

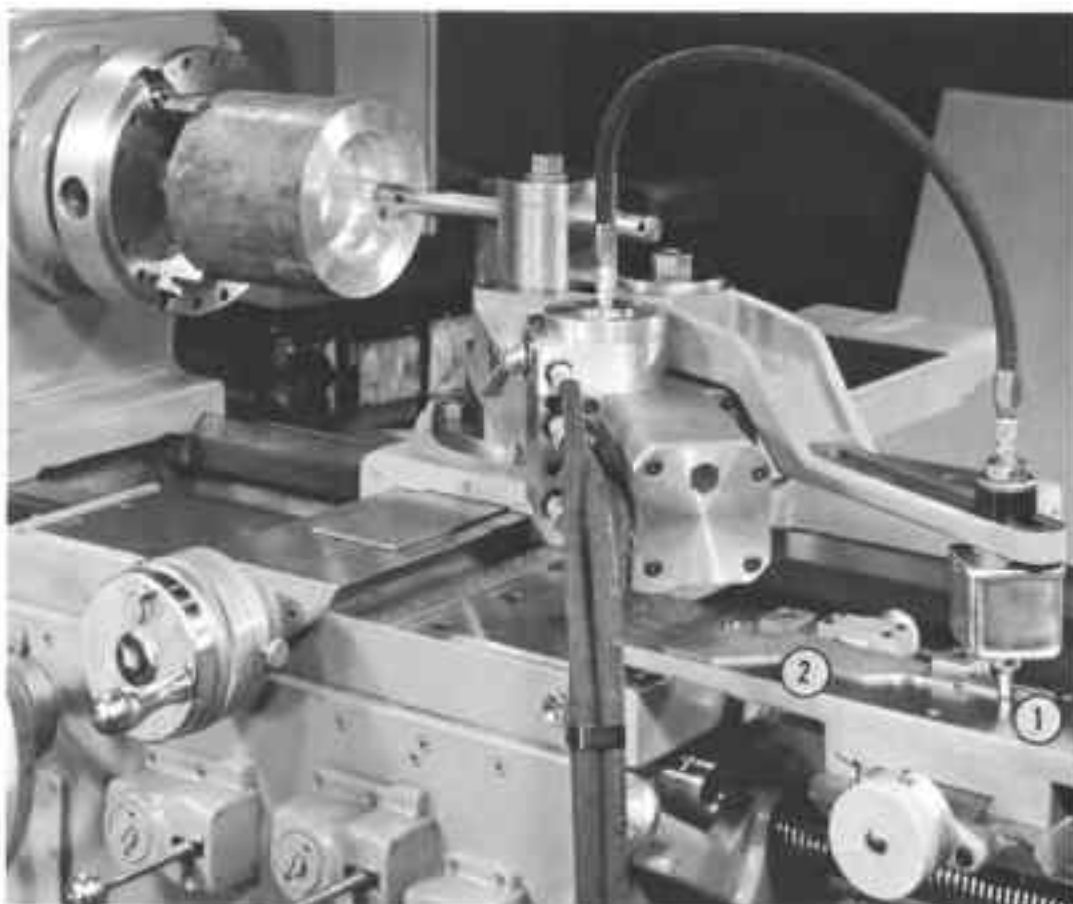
design are used, depending on the job to be done. The lead angle used, if any, should always be small. Also, the nose radius of the boring tool must not be too large. The cutting speed used for boring can be equal to the

speed for turning. However, when the spindle speed of the lathe is calculated, the finished, or largest, bore diameter should be used. The feed rate for boring is usually somewhat less than for turning to compensate for the lack of rigidity of the boring bar.

The boring operation is generally performed in two steps; namely, rough boring and finish boring. The objective of the rough-boring operation is to remove the excess metal rapidly and efficiently, and the objective of the finish-boring operation is to obtain the desired size, surface finish, and location of the hole. The size of the hole is obtained by using the trial-cut procedure. The diameter of the hole can be measured with inside calipers and outside micrometer calipers as described in Chapter 2, Basic Measuring Instruments, or inside micrometer calipers can be used to measure the diameter directly.

Cored holes and drilled holes are sometimes eccentric with respect to the rotation of the lathe. When the boring tool enters the work, the boring bar will take a deeper cut on one side of the hole than on the other, and will deflect more when taking this deeper cut, with the result that the bored hole will not be concentric with the rotation of the work. This effect is corrected by taking several cuts through the hole using a shallow depth of cut. Each succeeding shallow cut causes the resulting hole to be more concentric than it was with the previous cut. Before the final, finish cut is taken, the hole should be concentric with the rotation of the work in order to make certain that the finished hole will be accurately located.

Shoulders, grooves, contours, tapers, and threads are also bored inside of holes. The procedure for boring tapers and threads will be described in subsequent chapters. Internal grooves are cut using a tool that is similar to an external grooving tool (see Fig. 8-1). The procedure for boring internal shoulders is very similar to the procedure for turning shoulders that was previously described in Chapter 8. Large shoulders are faced with the boring tool positioned with the nose leading, and using the cross slide to feed the tool. Internal contours can be machined using a tracing attachment on a lathe as shown in Fig. 9-26. The tracing attachment is mounted on the cross slide and the stylus, 1, in Fig. 9-26 follows the outline of 2, the master profile plate. This causes the cutting tool to move in a path corresponding to the profile of the master profile plate in a manner explained previously in Chapter 6, Engine-Lathe Construction. Thus, the profile on the master profile plate is reproduced inside the bore. The master profile plate is accurately mounted on a special slide which can be precisely adjusted in two directions, in order to align the cutting tool in the correct relationship to the work. This lathe has a cam-lock type of spindle nose which permits it to take a cut when rotating in either direction. Normal turning cuts are taken with the spindle rotating counterclockwise. In Fig. 9-26, the boring cut is taken with the spindle revolving in a clockwise direction, or "backwards." This permits the boring cut to be taken on the "back side" of the bore which is easier to see from the operator's position in front of the lathe. This should not be done on lathes



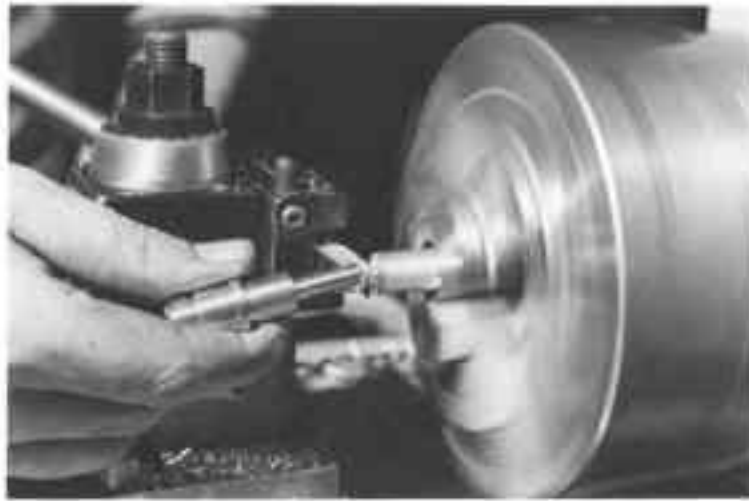
Courtesy of The Monarch Machine Tool Company

Fig. 9-26. Boring an internal contour on an engine lathe equipped with a tracing attachment.

having a threaded spindle nose because the cutting force will tend to unscrew the chuck.

Cutting Off

The objective of the cut-off operation is to sever the end of the workpiece which is protruding from the chuck. Often, this is a finished or semi-finished workpiece that has been machined from a bar of stock which is held in a chuck such as shown in Fig. 9-27. A cut-off tool, Fig. 9-28, is used to perform the cut-off operation. The end cutting edge is ground at a 3° to 15° angle in order to cut off the part without a burr. The burr remains on the stock that is held in the chuck and can be removed by feeding the tool a short distance beyond the point where the stock is separated. The width of the cut-off blade is dependent upon the size of the cut-off tool. It should be kept as small as possible. Although both larger and smaller widths are used, the average width of the cut-off blade is from $\frac{1}{8}$ to $\frac{1}{4}$ inch. The blade has a back taper of one degree on each side in



Courtesy of Cincinnati Milacron

Fig. 9-27. Cutting off a finished part in an engine lathe.

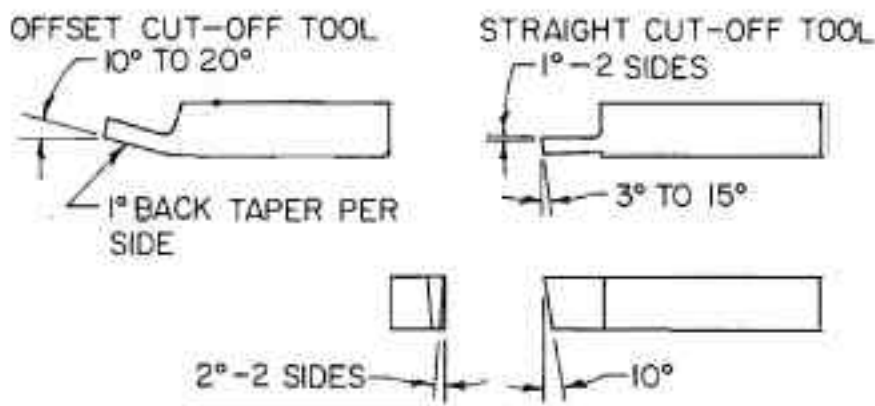


Fig. 9-28. Design of a cut-off tool.

order to provide clearance for the tool with respect to the sides of the groove made by this tool. The side relief angles are two degrees and the end relief angle is ten degrees. The face of the tool is made flat, although a very large radius (equal to the radius of the grinding wheel used to prepare this surface) is sometimes ground on the face. This provides a very small back-rake angle. Two frequently used commercial cut-off tools are shown in Fig. 9-29.

The cutting speed used to cut off stock can usually be as fast as recommended for turning. Occasionally, a very deep cut must be taken to cut off a part, in which case the cutting speed should be reduced. This is done more in the nature of a safety measure than to decrease the cutting load on the tool. The cutting load, or cutting force, on the tool will not be reduced by decreasing the cutting speed, but by reducing the feed. For this reason, a relatively light feed should always be used to perform the cut-off operation. For very small cut-off tools a feed rate as low as



Courtesy of the Armstrong Brothers Tool Co.

Fig. 9-29. A. Straight-shank cut-off tool. B. Offset-shank cut-off tool.

.0005 inch per revolution is sometimes used. Large cut-off tools can operate satisfactorily using a feed of .006 to .010 inch per revolution. For average conditions a feed rate of .001 to .005 inch per revolution is recommended. It is best to use the automatic-power across feed to perform this operation since this feed is more uniform than the hand feed. It should be pointed out that the *actual* feed rate of the cross slide of most engine lathes is less than the *longitudinal* feed rate corresponding to the same setting of the feed-thread gear box. The setting of a quick-change gear box will correspond to the actual longitudinal feed; however, the cross-slide feed will be less. Since the ratio of the longitudinal and cross feeds depends upon the design of the lathe, no general rule for calculating the actual cross feed rate can be given. This must be determined experimentally for each lathe.

The correct setup for an offset cut-off tool is shown in Fig. 9-30. For both offset and straight cut-off tools, the blade must always be positioned perpendicular to the workpiece axis and the end cutting edge must be on center.

Lathe Fixtures

Fixtures are special tools used to locate the workpiece in a desired position for machining and to clamp it firmly in place. Usually, fixtures are used on production work, where a number of like parts are to be machined. There are occasions, however, when the configuration of a part is such that it cannot be located and held by any of the standard work holding methods. On such occasions, a fixture is made to machine **only** one or two parts. Such fixtures are made as simple as possible without sacrificing the required accuracy.

Lathe fixtures are usually mounted on a faceplate or in a four-jaw

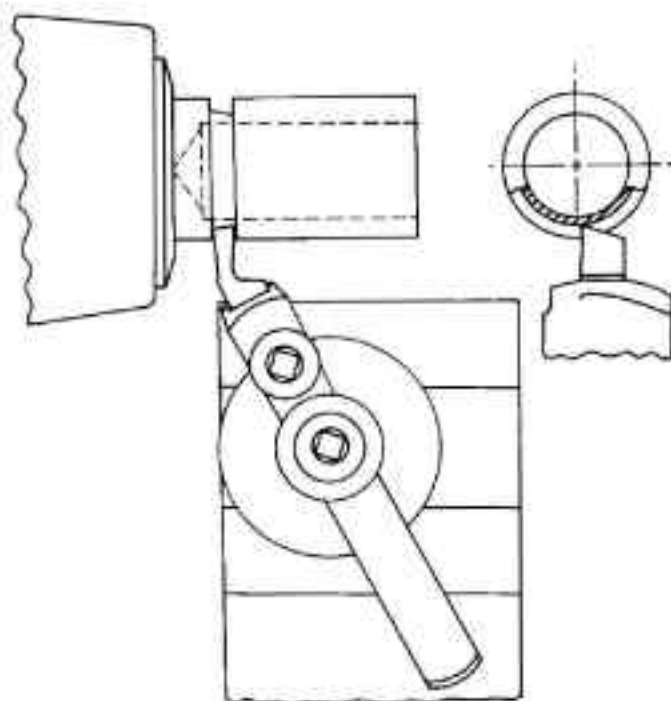
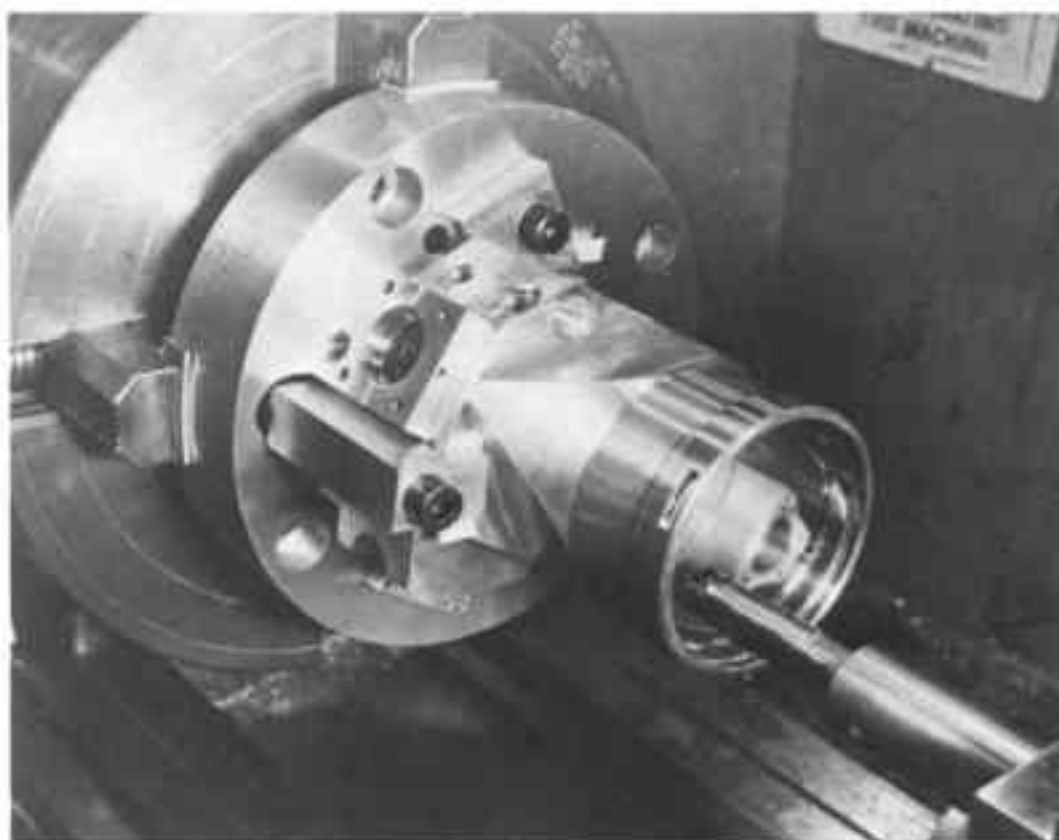


Fig. 9-30. Correct setup for an offset cut-off tool. Blade must be perpendicular to work and on center.



Courtesy of "Model Engineer" magazine

Fig. 9-31. Stub mandrel used to align bore of workpiece with axis of lathe to insure concentricity and perpendicularity of turned surface with bore.



Courtesy of the Accurate Diamond Tool Corp.

Fig. 9-32. Diamond boring a complex part held in a lathe fixture mounted on a four-jaw chuck. Diamond boring tool produces mirror-like surface finish on aluminum part.

chuck, although some very simple fixtures may be held in a three-jaw universal chuck or in a collet chuck. A very simple fixture for turning a pulley is shown in Fig. 9-31. Actually this is a stub mandrel used to insure concentricity between the outside diameter and the bore, and also to insure perpendicularity between the sides and the bore. The mandrel is made from a bar of steel which is turned to fit the bore and a thread is cut on the end to hold a retaining nut. It is turned and then used without removing it from the lathe. It may be removed and used again by indicating one of the cylindrical surfaces or by always holding it in a collet chuck. However, a collet chuck should not be used when turning a large diameter.

Most fixtures not mounted on a faceplate are preferably held in a four-jaw chuck. Since each pair of jaws can be accurately adjusted independently of the other pair, the fixture can be accurately located in the four-jaw chuck. A lathe fixture held in a four-jaw chuck that is mounted on an engine lathe is shown in Fig. 9-32. The fixture is made to run true

by indicating the diameter and the face with a dial test indicator. The workpiece is a complicated forged aluminum alloy valve body that must be machined to close tolerances and to a mirror smooth surface finish. To attain the required accuracy and the mirror smooth surface finish a natural diamond cutting tool held on a boring bar is used.

Taper and Angle Turning

Twist drills, end mills, arbors, lathe centers, other small tools, and machine parts have tapered shanks which fit into spindles, sockets, or adapters having a matching taper. Heavy-duty power generating and marine components are often designed to have tapered fits. The taper provides an accurate means of location for the mating parts. Many tapers are also designed to hold the two members together. They are called self-holding tapers in contrast to self-releasing tapers which are designed to locate, but not to hold. Tapers have been standardized in order to be interchangeable. Most machine shops are frequently called upon to measure or gage, and to machine tapers. External tapers are machined either by turning in a lathe or by grinding. Internal tapers are machined by boring in a lathe, reaming, and by internal grinding. This chapter will concern itself only with the measurement of tapers, and with cutting tapers and other conical surfaces on a lathe.

Standard Tapers

The exact dimensions and specifications for standard tapers are tabulated in engineering handbooks such as "Machinery's Handbook." The objective here is to introduce the standard tapers in common usage in the United States.

Morse Taper. This is a self-holding taper. Taper-shank twist drills are made exclusively with the Morse taper. It is also used extensively on the shanks of reamers, end mills, countersinks, counter bores, spot facers, and lathe centers. It is used on drill-press spindles and lathe spindles. Each size of Morse taper is different, although in most cases it is approximately $\frac{5}{8}$ inch per foot.

Brown & Sharpe Taper. This taper is approximately $\frac{1}{2}$ inch per foot for all sizes except No. 10 which is .5161 inch per foot. Brown & Sharpe tapers are used on the shanks of end mills and reamers, collets, milling machine spindles, and grinding-machine spindles.

Jarno Taper. The Jarno taper has been used on some machine tools. This system is based on some very simple formulas that make all the dimensions very easy to calculate. The taper per foot for all Jarno tapers is .600 inch. The following formulas are used to calculate the other dimensions:

$$\text{Diameter at large end} = \frac{\text{Jarno taper number}}{8}$$

$$\text{Diameter at small end} = \frac{\text{Jarno taper number}}{10}$$

$$\text{Length of taper} = \frac{\text{Jarno taper number}}{2}$$

Thus, a No. 7 Jarno taper has the large diameter equal to $\frac{7}{8}$ inch, the small diameter equal to $\frac{7}{10}$ or .700 inch, and the length is equal to $3\frac{1}{2}$ inches.

Milling-Machine Taper. This taper is $3\frac{1}{2}$ inches per foot in all sizes, in order to assure easy release from the milling-machine spindles of arbors, adapters, and other tools. It was adopted as a taper by the milling-machine manufacturers of the National Machine Tool Builders Association, in 1927. It has now become an American standard and is used on almost all of the milling-machine spindles built in the United States.

American Standard Machine Tapers. This standard includes a self-holding series, and a steep taper, or self-releasing, series. The self-holding tapers are, to a large extent, an adaption of the Morse and Brown & Sharpe tapers. The steep taper series corresponds closely to the Milling-Machine Taper.

There are other standard tapers, such as for American Standard Taper Pins and for the Jacobs Taper that is used for chucks and on spindles of portable air and electric tools. The wide application of all of these standards has contributed greatly to the growth of metal manufacturing.

Taper Calculations

The following definitions are used in dealing with tapered surfaces:

T = Taper; the difference in size between the large and the small diameter.

T_{pi} = Taper per inch; the change in size of the taper in a one-inch length.

T_{pf} = Taper per foot; the change in size of the taper in a one-foot length.

D = Large diameter of the taper

d = Small diameter of the taper

L = The length of the taper in inches.

The formulas used to calculate the components of a taper are given below. Their applications can best be illustrated by examples:

$$T = D - d \quad (10-1)$$

$$T_{pi} = \frac{D - d}{L} \quad (10-2)$$

$$T_{pi} = \frac{T_{pf}}{12} \quad (10-3)$$

$$T_{pf} = 12 \times T_{pi} \quad (10-4)$$

$$D = (T_{pi} \times L) + d \quad (10-5)$$

$$d = D - (L \times T_{pi}) \quad (10-6)$$

$$L = \frac{D - d}{T_{pi}} \quad (10-7)$$

Example 10-1:

A taper is 4 inches long. The large diameter is 1 inch and the small diameter is $\frac{3}{4}$ inch. Calculate the taper per foot.

$$L = 4 \text{ in.}, \quad D = 1 \text{ in.}, \quad d = \frac{3}{4} \text{ in.}$$

$$T_{pi} = \frac{D - d}{L} = \frac{1 - \frac{3}{4}}{4} = .0625 \text{ in.} \quad (10-2)$$

$$\begin{aligned} T_{pf} &= 12 \times T_{pi} = 12 \times .0625 \\ &= .750 \text{ in.} \end{aligned} \quad (10-4)$$

Example 10-2:

Calculate the diameter of the large end of a taper when the small diameter is .778 inch, the taper per foot is .60235 inch, and the length of the taper is 3.250 inch.

$$L = 3.250 \text{ in.}, \quad d = .778 \text{ in.}, \quad T_{pf} = .60235 \text{ in.}$$

$$T_{pi} = \frac{T_{pf}}{12} = \frac{.60235}{12} = .05019 \text{ in.} \quad (10-3)$$

$$\begin{aligned} D &= (T_{pi} \times L) + d = (.05019 \times 3.250) + .778 \\ &= .941 \text{ in.} \end{aligned} \quad (10-6)$$

Measuring and Gaging Tapers

The method of gaging tapers is illustrated in Fig. 10-1. Taper plug and taper ring gages are simple and dependable for gaging tapered holes and tapered shafts, respectively. They simultaneously gage the diameter and the angularity of the taper.

To gage the angularity of a tapered shaft, place a light coating of Prussian blue along the entire length of the taper. If Prussian blue is not available, a light chalk can act as a substitute. The taper is inserted in the ring gage and twisted. If, after it has been removed, the Prussian blue coating has been uniformly rubbed off, the taper is cut at the correct angle. If a portion of the Prussian blue is untouched, the angle is not correct. The direction of the error can be determined by observing the

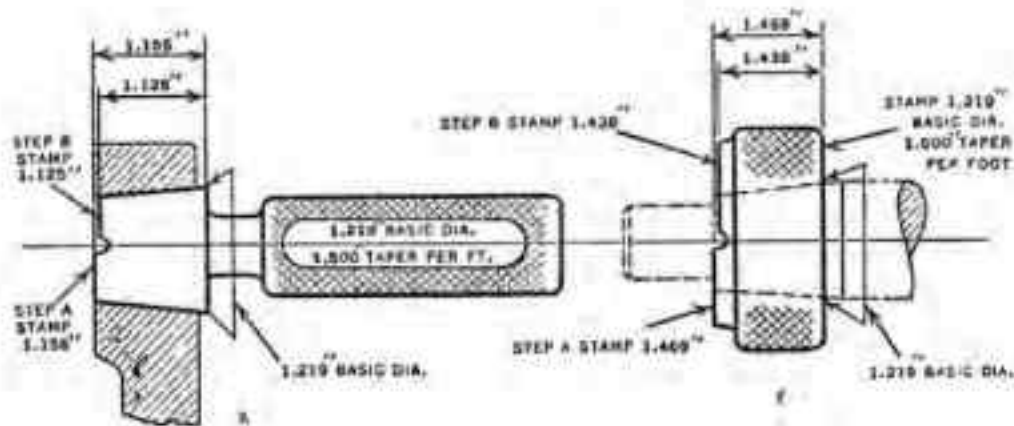


Fig. 10-1. Step type plug and ring Go-Not Go taper gages.

area where the Prussian blue has been rubbed off. If it has been rubbed off toward the large end, the taper is too steep and if it has been rubbed off toward the small end, it is too gradual. Tapered holes are gaged for angularity in a similar manner, using a taper plug gage. In this case the Prussian blue, or chalk, is placed on the plug gage instead of on the workpiece. When gaging tapered holes with a taper plug gage these conditions, previously indicated for a shaft, are reversed. A self-holding taper that has the correct angle can be made to hold together without any difficulty.

The taper plug and ring gages are designed on the GO-NOT GO principle. For example, at *Y*, Fig. 10-1, the small end of the ring gage has two steps, or surfaces, at different levels which are designated Step *A* and Step *B*. When Step *A* is flush with the small end of the shaft, the basic diameter of 1.219 inches is, in this case, 1.469 inches from the small end. When Step *B* is flush, the basic diameter is 1.438 inches from the small end. Hence, this is a limit type of taper ring-gage and the tolerance for the position of the gage is represented by the height of the step which, in this case, equals $1.469 - 1.438 = .031$ inch. When the gage is flush with either step, or when the end of the work falls within the two steps, the taper is within the specified limits. If the end of the work is inside Step *B*, the taper is too large; if the end is outside Step *A*, the taper is too small. The plug gage at *X*, Fig. 10-1, also has a stepped surface and the hole is gaged in a manner similar to gaging the shaft with the taper ring-gage. The plug and ring gages determine if the taper being gaged meets the dimensional specifications; however, it does not actually measure the taper.

The angularity of the taper is measured most readily by using a sine bar or a sine plate. A sine bar is shown in Fig. 10-2. The arbor, or shaft, that is to be measured is held on two bench centers. The sine bar has two buttons which are similar to toolmakers buttons except that they are not adjustable. These buttons are usually either 10 inches or 5 inches apart. Their location on the sine bar is accurate to "tenths." The sine bar is positioned over the taper, as shown, and the difference in the height over the buttons, H_1 and H_2 , is measured using a vernier height-gage and a dial test indi-

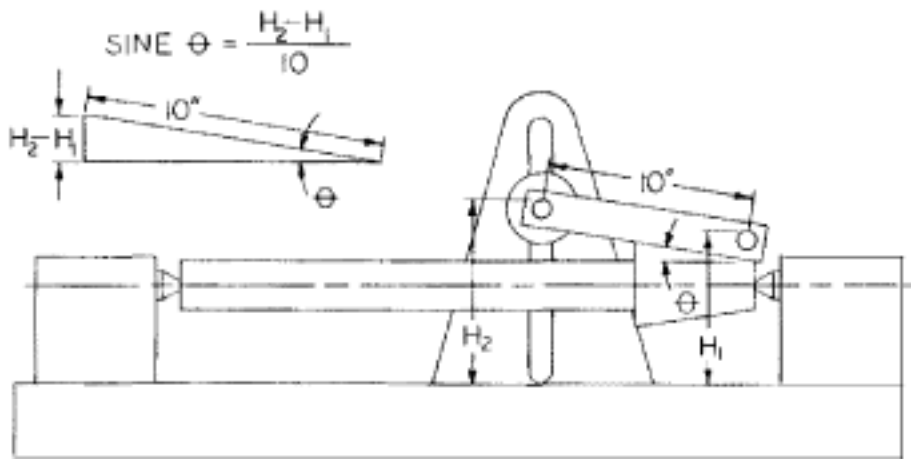


Fig. 10-2. Measuring the angle of the taper using a sine bar and precision bench-centers.

cator. The half angle of the taper is obtained from this measurement by applying the following formula:

$$\sin \theta = \frac{H_2 - H_1}{10} \quad (\text{For 10-inch sine bar}) \quad (10-8a)$$

$$\sin \theta = \frac{H_2 - H_1}{5} \quad (\text{For 5-inch sine bar}) \quad (10-8b)$$

The angle, θ , is found directly from a table of trigonometric functions by obtaining the angle corresponding to the calculated value of $\sin \theta$ in Formula 10-8. The included angle of the taper is two times θ . The taper per inch can be found by using the following formula:

$$T_{pi} = 2 \times \tan \theta \quad (10-9)$$

Example 10-8:

Referring to Fig. 10-2, the vernier height-gage reading obtained for H_1 is 4.678 inches and for H_2 it is 6.122 inches. Calculate the included angle of the taper and the taper per foot of the arbor.

$$\sin \theta = \frac{H_2 - H_1}{10} = \frac{6.122 - 4.678}{10} = .1444 \text{ inch} \quad (10-8a)$$

$$\theta = 8^\circ 18' \quad (\text{From a table of trigonometric functions})$$

$$\begin{aligned} \text{The included angle of the taper} &= 2 \times \theta = 2 \times 8^\circ 18' \\ &= 16^\circ 36' \end{aligned}$$

$$\begin{aligned} T_{pi} &= 2 \times \tan \theta = 2 \times \tan 8^\circ 18' \\ &= 2 \times .14588 = .29176 \text{ inch} \end{aligned} \quad (10-9)$$

$$\begin{aligned} T_{pf} &= 12 \times T_{pi} = 12 \times .29176 \\ &= 3.501 \text{ inch} \end{aligned} \quad (10-4)$$

The half angle of the taper can also be determined by measuring over precision rolls as shown in Fig. 10-3. Two measurements, M and m , must be taken that are a distance H , apart. The distance, or height, above the precision-surface plate H , is obtained by two stacks of gage blocks of equal length. The half angle, θ , is calculated by the following formula:

$$\tan \theta = \frac{M - m}{2H} \quad (10-10)$$

The diameter D , at any given distance H , from the small end of the taper, can be determined by measuring the distance M over precision-gage rolls having a known diameter W by the following formula:

$$D = M - W [1 + \cotan \frac{1}{2} (90 - \theta)] \quad (10-11)$$

The diameter of the small end of the taper can be measured by substituting the measurement m , for M in Formula 10-11.

Example 10-4:

Referring to Fig. 10-3, the measurement across the rolls M is 2.2558 inches, and m is 2.0333 inches. The distance between the rolls H is 4.250 inches and the diameter of the rolls is .500 inch. Calculate the half angle θ .

$$\tan \theta = \frac{M - m}{2H} = \frac{2.2558 - 2.0333}{2 \times 4.250} = .026176$$

$$\theta = 1^{\circ}30' \text{ (From table of trigonometric functions)}$$

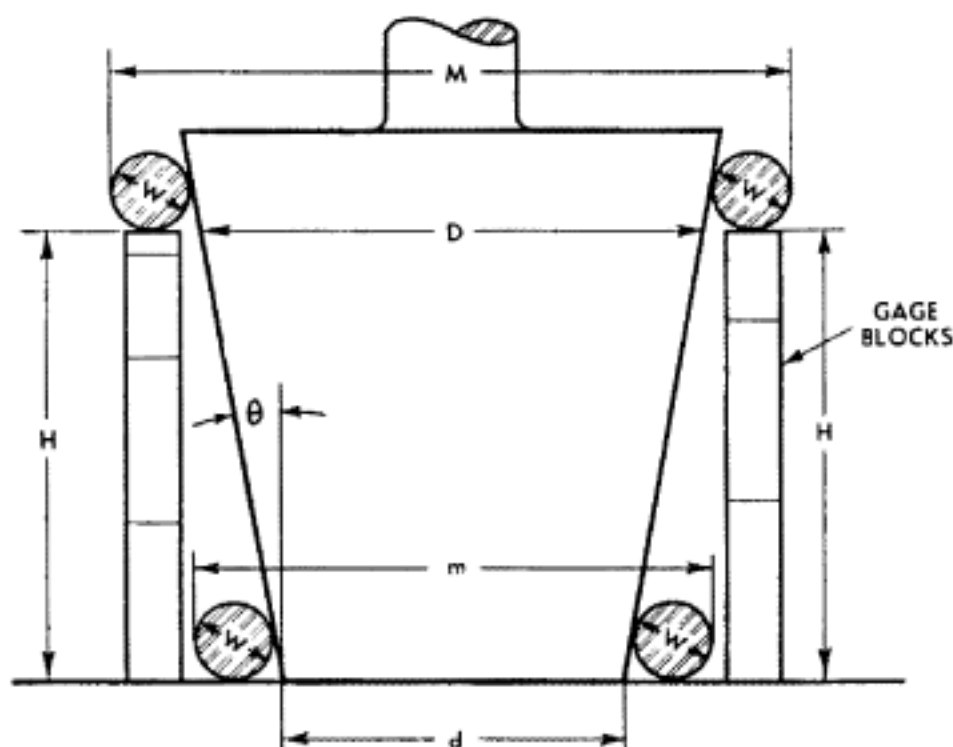


Fig. 10-3. Measuring a taper with gage rolls and precision gage-blocks.

Calculate the diameter of the taper at a distance of 4.250 inches from the small end.

$$\begin{aligned}
 D &= M - W [1 + \cotan \tfrac{1}{2} (90 - \theta)] \\
 &= 2.2558 - .500 [1 + \cotan \tfrac{1}{2} (90 - 1^\circ 30')] \\
 &= 2.2558 - .500 [1 + \cotan 44^\circ 15'] \\
 &= 2.2558 - .500 [1 + 1.0265] \\
 &= 1.2426 \text{ inches}
 \end{aligned}$$

Calculate the diameter at the small end.

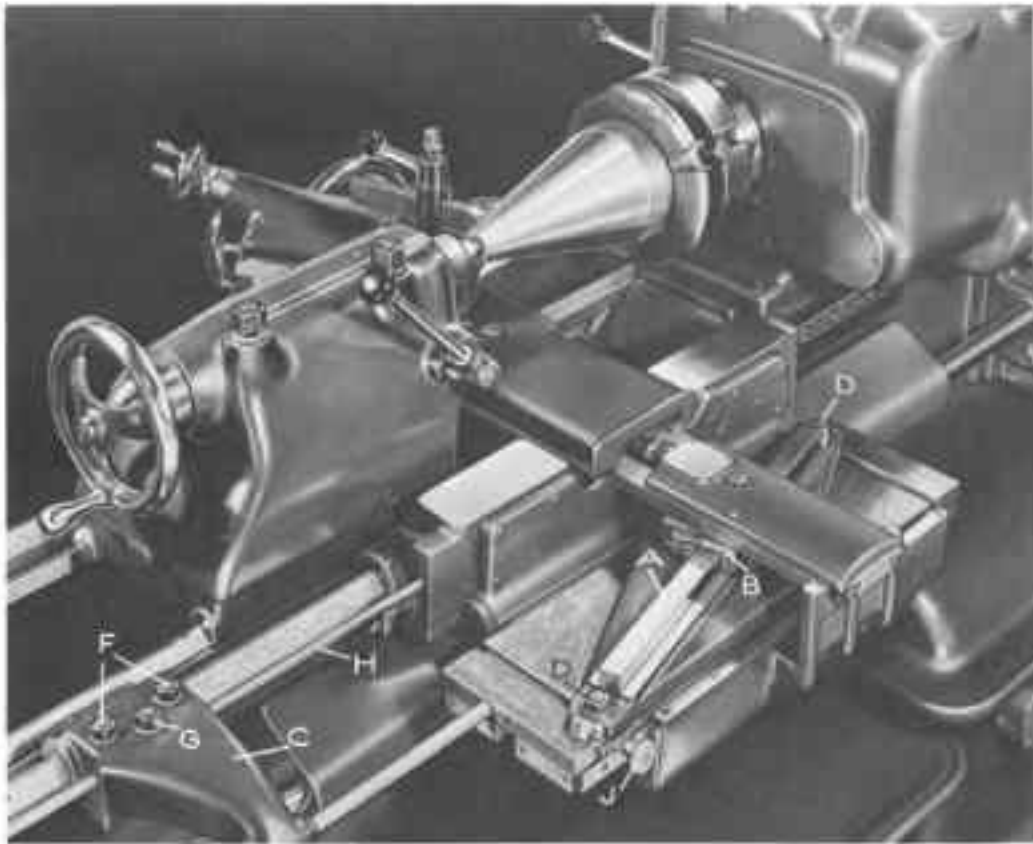
$$\begin{aligned}
 d &= m - W [1 + \cotan \tfrac{1}{2} (90 - \theta)] \\
 &= 2.0333 - .5 [1 + \cotan \tfrac{1}{2} (90 - 1^\circ 30')] \\
 &= 1.0201 \text{ inches}
 \end{aligned}$$

Taper Turning with a Taper Attachment

The taper attachment is a guide bar that imparts a uniform motion to the cross slide as the carriage is moved longitudinally. The rate or extent of the cross-feed movement for a given longitudinal movement determines the angle of the taper. The angle of the taper on the work corresponds to the angle at which the taper attachment is set. This can be varied and provided on the taper attachment are scales that can be read in terms of degrees and taper per foot. The taper attachment does not interfere with straight or cylindrical turning and boring. It is a permanent part of the lathe that is always ready to be used by making a few simple adjustments. A taper part is turned practically the same as a cylindrical part; that is, the power longitudinal feed is used and, as the carriage moves along the bed, the cross slide and cutting tool are gradually moved in or out by the taper attachment. The depth of cut of the cutting tool is set by using the compound-rest slide, or, if the taper attachment is a telescoping type, the cross slide can be used.

A taper attachment and a setup for turning tapers is shown in Fig. 10-4. The bar, *A*, is set either to the required taper per foot or to a given angle, in degrees, as determined by the graduations on the end of the bar. The nuts, *D*, are used to clamp the bar securely in the required position. Arm *C*, is clamped to the lathe bed by tightening nuts *F*, so that the bar, *A*, is held stationary. Nut *G*, is loosened to permit draw-bar *H*, to slide freely through arm *C*. When the attachment is not in use, nut *G*, is tightened so that the auxiliary draw-bar *H*, will insure a smooth, easy movement of arm *C*, along the bed without any tendency to bind. The taper bar *A*, may be adjusted to the exact position required, by means of the knurled knob, *J*. This attachment has a telescoping type of cross-feed screw permitting the cross slide to be used to adjust the tool to the depth of cut. It has a maximum capacity of 6 inches taper per foot. This exceptionally steep capacity is due to the incorporation of antifriction bearings, in order to reduce the frictional resistance.

The taper attachment can be accurately adjusted by placing a dial test indicator in the toolpost and placing the contact point against the tail-



Courtesy of The American Tool Works

Fig. 10-4. Details of an engine-lathe taper attachment.

stock spindle, as shown in Fig. 10-5. The taper per inch of the taper to be cut is calculated and divided by two. This is the distance that the cross slide should move when the carriage is moved exactly one inch, longitudinally. The taper attachment is then adjusted until the dial indicator

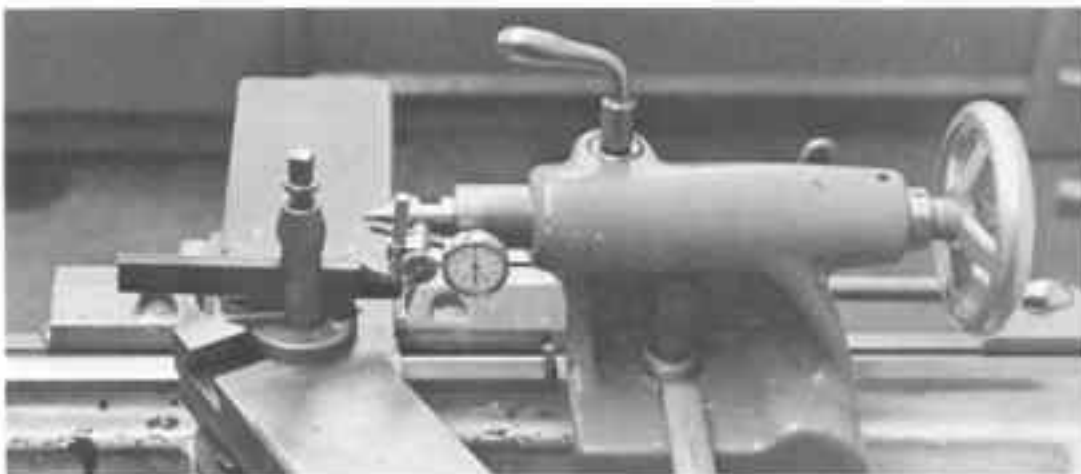


Fig. 10-5. Method of setting the taper attachment. The same setup can be used to offset the tailstock.

reads the number of thousandths of an inch calculated above, when the carriage is moved exactly one inch. Care must be taken to eliminate all of the lost motion in the taper attachment before starting the dial indicator reading. This is done by moving the carriage toward the headstock until the dial indicator hand starts to turn. The one-inch, longitudinal movement of the carriage is started from this position. It must be emphasized, however, that even this method does not guarantee that the taper produced will be absolutely accurate. Accuracy is obtained by measuring or gaging the taper that has been cut and making fine adjustments of the taper attachment until the angle of the taper per foot is correct.

The point, or nose, of the cutting tool must be set at the same height as the center, or axis, of the work when turning tapers. The importance of this will be apparent by referring to Fig. 10-6. To turn the taper shown, the tool, *T*, would be moved back a distance *X*, while traversing the length, *l*. In order to illustrate the principle, assume that the tool could be placed as high as point *a*, the setting of the taper attachment

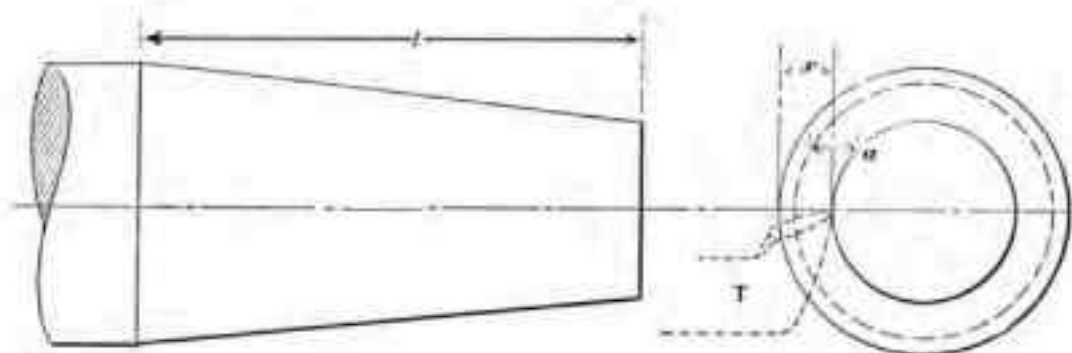


Fig. 10-6. The nose of the cutting tool must be in the same horizontal plane as the axis of the work, for taper turning.

remaining as before. When in this position, the tool would again move back a distance *X*, while traversing a distance *l*; however, in this case the large diameter would be undersize as shown by the dotted line if the diameters of the small end were the same in each case. Of course, if the tool were only slightly above or below the center, the resulting error would also be small. The tool can be set on center by comparing the height of the point, or nose, with one of the lathe centers, before placing the work in the lathe. This precaution must be observed when turning or boring any taper or angle, regardless of the method used.

With the work set up, as shown in Fig. 10-4, the entire length of the taper is cut until the taper ring-gage slips over the work far enough to allow the angle of the taper to be gaged, yet still leaving enough metal for one or more finish cuts to be taken. Several cuts are usually required and the last cut before using the gage should be a light one so that the resulting taper will accurately reflect the setting of the taper attachment. If necessary, the taper attachment is adjusted to correct any error in the angle of the taper as determined by the taper ring-gage and then another

light cut is taken along the entire length of the taper. When the angle of the taper is correct, one or more finish cuts are taken until the taper is completed. If a taper ring-gage is not available, a drill press sleeve, lathe headstock-spindle adapter, or a part having the correct taper may be used in its place. It is best, however, to use a taper gage even if one must specially be made for the job at hand.

Taper Turning—Offset-Center Method

The offset-center method of turning tapers, shown in Fig. 10-7 can be used only to turn external tapers with the work held between the head-

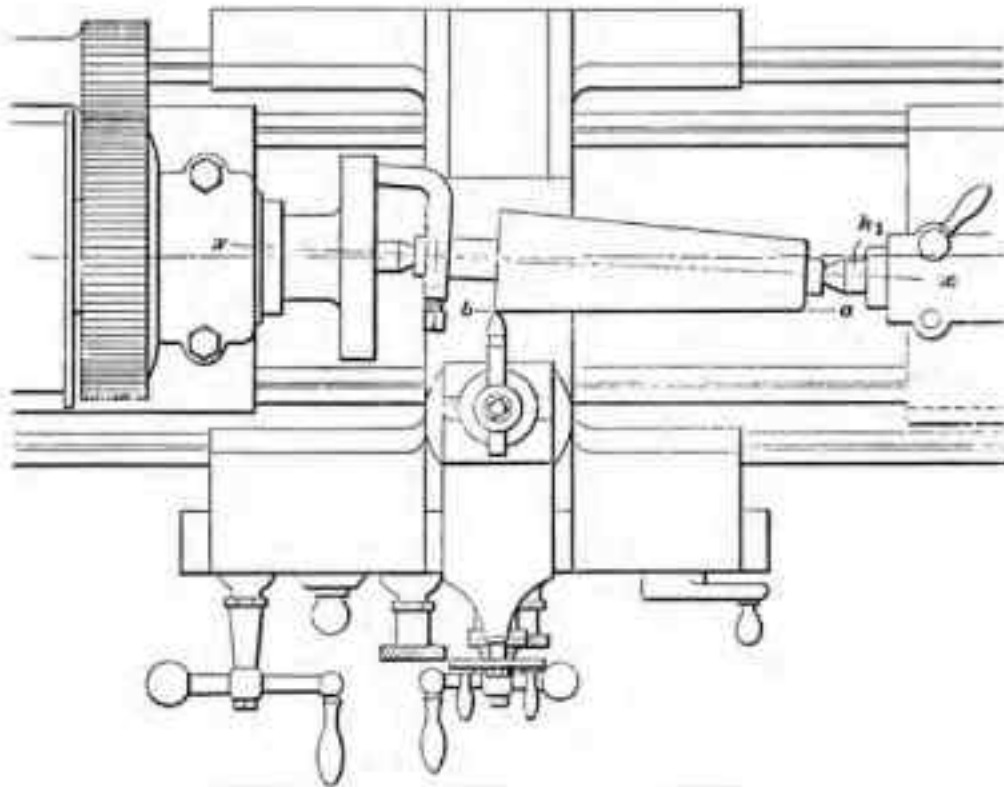


Fig. 10-7. Taper turning by offset-center method on lathe not equipped with a taper attachment.

stock and the tailstock centers. When both of these centers are in line, the movement of the tool is parallel to the axis of the work and, consequently, a cylindrical surface is produced. If, however, the tailstock center is set out of alignment, as shown in Fig. 10-7, the work will then be turned, tapering as the tool is traversed from a to b , because the axis $x-x$ is at an angle with the movement of the tool. The amount of taper, or difference between the diameters at the ends for a given length, will depend upon how much the center h_1 , is set over from the central position.

The amount that the tailstock center should be set over for turning a given taper can be calculated; however, the calculated distance is only a close approximation due to the effect of the centers inside the center holes

of the workpiece. Because of differences in the depth of the center holes and the length of each individual workpiece, each workpiece must be gaged to determine the angle of the taper before it is cut to the finish size. This is not necessary when using a taper attachment which will reproduce the same taper after it has been set to the correct setting. The estimated amount of offset of the center for turning a taper by the offset-center method can be calculated by the following formula:

$$\text{Offset} = \frac{T_{pi} \times L_w}{2} \quad (10-12)$$

where: T_{pi} = Taper per inch
 L_w = Length of the workpiece

Example 10-5:

A taper plug that is $7\frac{1}{2}$ inches long is to be turned in a lathe by the offset-center method. The taper is 4 inches long and the taper per foot is .750 inch. Determine the tailstock offset.

$$T_{pi} = \frac{T_{pf}}{12} = \frac{.750}{12} = .0625 \text{ inch.}$$
$$\text{Offset} = \frac{T_{pi} \times L_w}{2} = \frac{.0625 \times 7.500}{2} = .234 \text{ inch}$$

To adjust the tailstock, place a dial test indicator against the tailstock spindle as was shown in Fig. 10-5. Set both the dial test indicator and the micrometer cross-feed dial to read zero. Using the graduations on the micrometer cross-feed dial to determine the distance, move the carriage back the required distance plus one turn of the cross-feed screw in order to take up the back lash of the screw. The cross slide is then moved forward one turn, plus the distance required to come to the micrometer cross-feed-dial reading. The tailstock is then moved by loosening the tailstock clamp-nut and turning the set-over screw until the tailstock spindle again comes against the dial test indicator and makes it read zero. The tailstock is then clamped in place. The procedure for turning the taper *after* the lathe has been set up, is essentially the same as is used when taper turning with a taper attachment.

Angle Turning—The Compound Rest

Steep tapers and angles can be turned by positioning the compound rest at the desired angle and feeding the cutting tool with the compound-rest slide. An example of this method is illustrated in Fig. 10-8. The correct position of the compound rest is obtained by using the angle graduations on the base of the compound rest. The compound rest can be positioned more accurately by using a vernier bevel-protractor and a dial test indicator which is attached to the compound-rest slide. The vernier bevel-protractor is held against any convenient surface on the lathe or

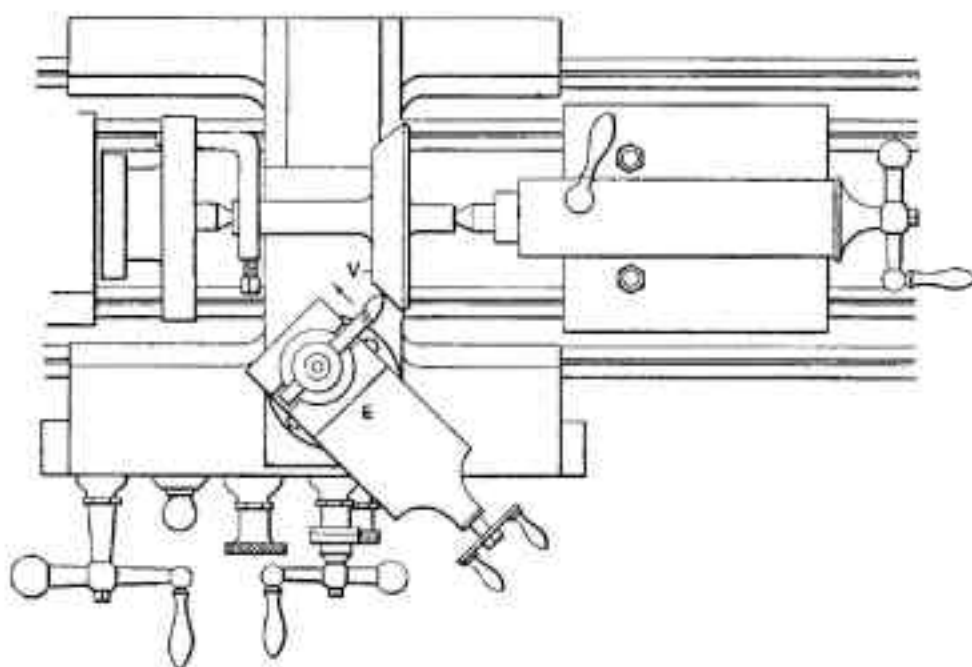
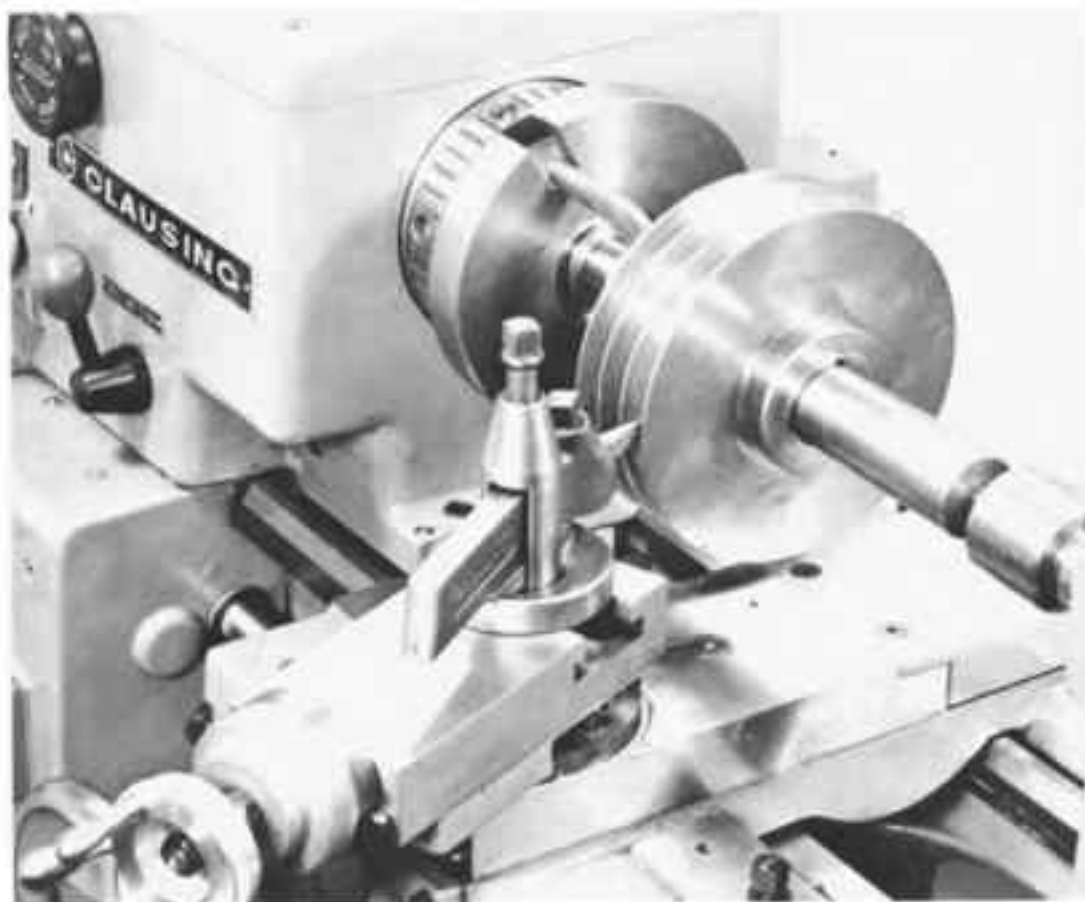


Fig. 10-8. Method of turning angles and steep tapers with the compound rest.



Courtesy of the Clausing Division of the Atlas Press Company

Fig. 10-9. Turning an angle on a V-belt sheave with the compound rest.

on the work and the contact point of the dial test indicator is moved along the blade of the protractor. A convenient reference surface for the protractor would be the face of a faceplate or the tailstock spindle. Very accurate settings of the compound rest are best made by taking a trial cut on the workpiece and measuring the resulting angle. The compound rest is adjusted until the angle being cut is correct. Figure 10-9 shows the angle of a V-belt sheave being turned, by using the compound rest to feed the tool. The sheave is held on a mandrel which is mounted between centers.

Boring Internal Tapers and Angles

Internal tapers can be bored by using the taper attachment as shown in Fig. 10-10. The taper attachment guide-bar *A*, is set to the required taper as shown by the graduations. The guide bar, locked into position by screws *D*, and arm *C* is clamped to the lathe bed by the screws shown. Lever *E*, is tightened and the attachment is ready to use. The point, or nose, of the boring tool must be set at the height of the lathe center. This can be done by comparing the height of the tailstock center (not shown in Fig. 10-10) with the nose of the boring tool using a surface gage, and taking the flat surfaces on the bed of the lathe as a reference surface. The procedure for boring the hole is similar to the procedure for turning a taper. The hole is bored undersize until the tapered plug-gage can be inserted to measure the angle of the taper. Following a light cut, the angle of the taper is measured and the taper attachment is adjusted to

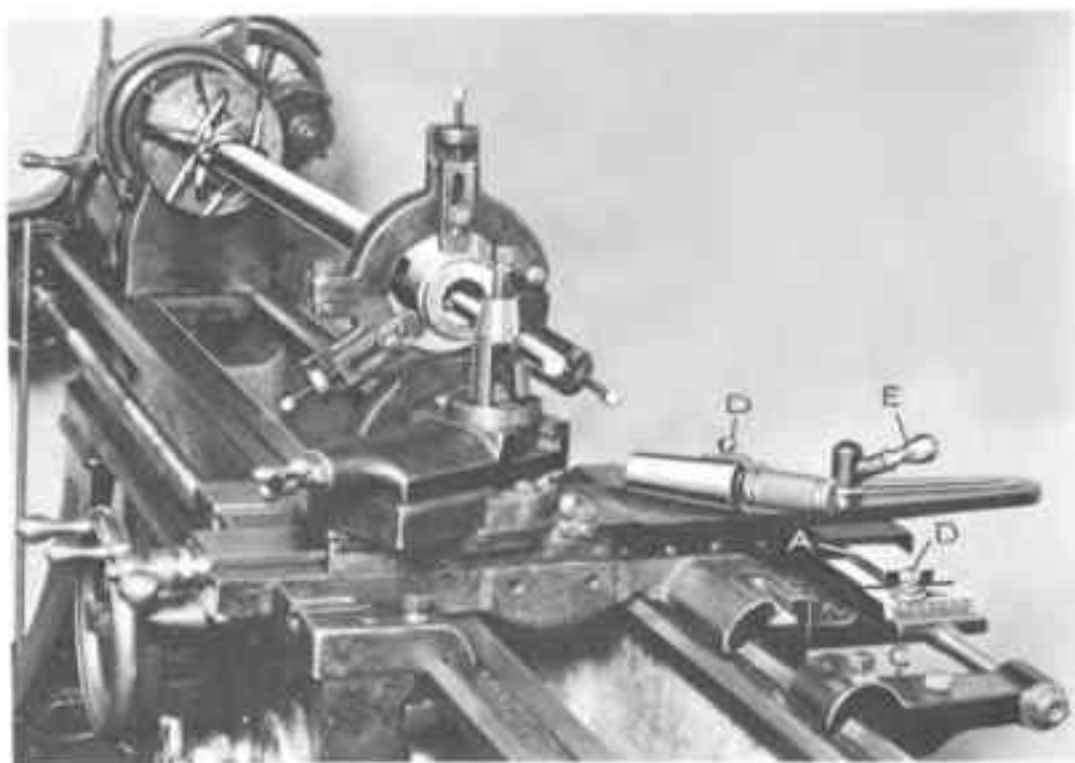
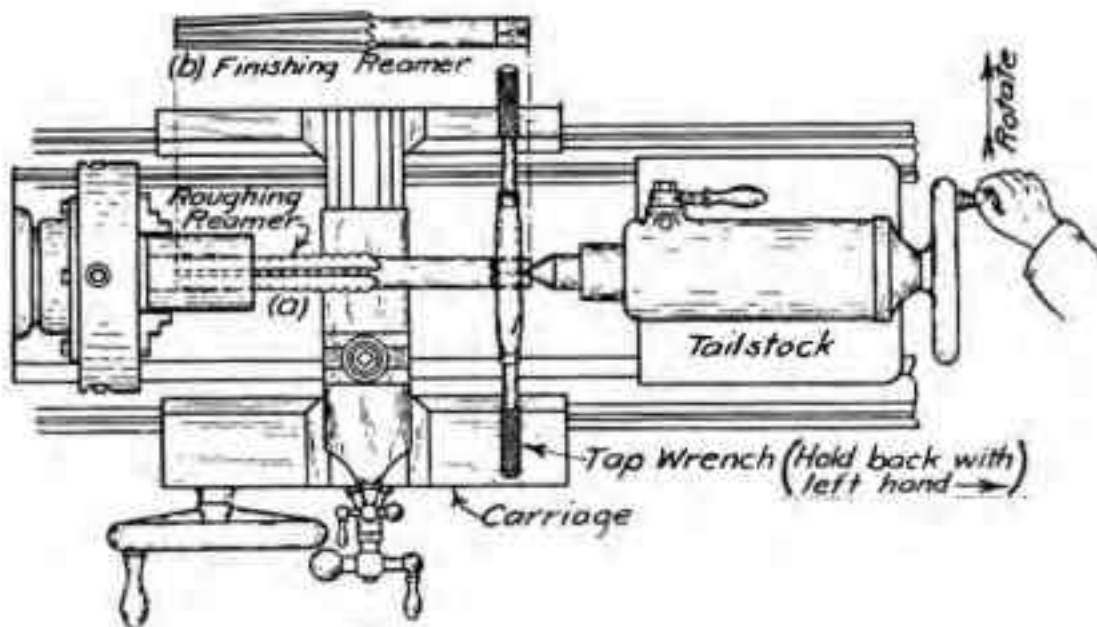


Fig. 10-10. Use of a taper attachment to bore a taper hole in the end of a spindle.



Courtesy of the Midwest Industrial Management Association.

Fig. 10-11. Taper reaming in an engine lathe.

make the necessary small correction. The taper is then bored to size. Internal tapers can usually be finished more rapidly and more accurately with a taper reamer as shown in Fig. 10-11. The taper in the hole is first rough-bored; this is followed by reaming with a roughing reamer and the final finish is obtained by reaming with a finishing reamer. Some-

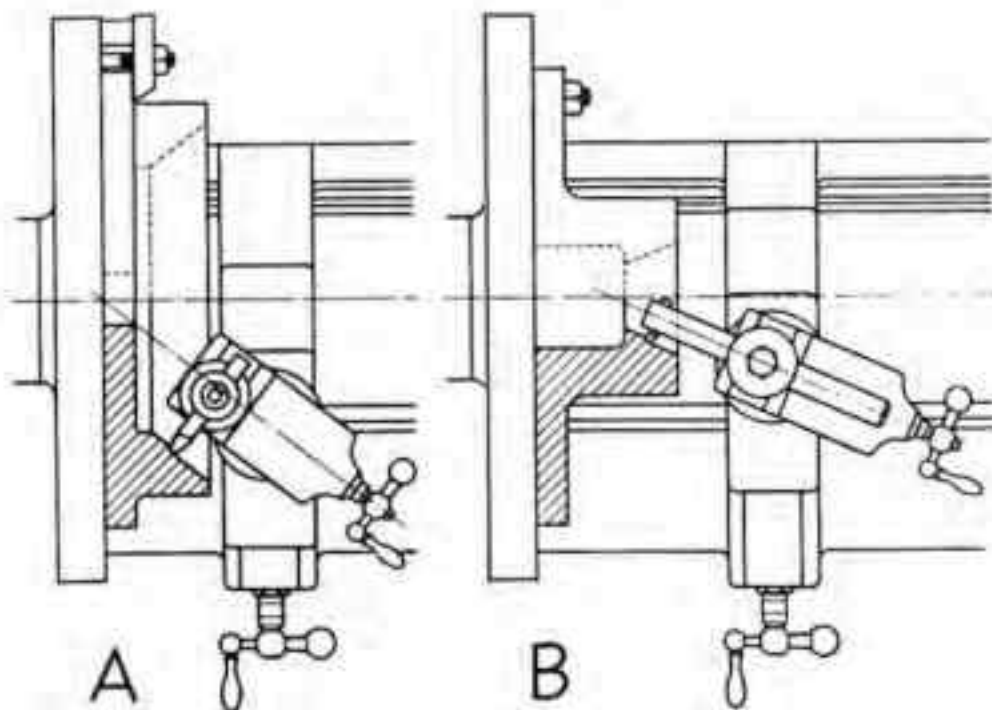


Fig. 10-12. Boring tapered holes by using compound rest set to the angle required.

times, the rough taper-boring operation is omitted and the tapered hole is made by reaming with the roughing and finishing reamers.

The tapered hole is reamed by placing a tap wrench on the square shank of the reamer and inserting the reamer into the bore. The tailstock center is inserted in the center-drilled hole in the end of the reamer. The headstock gears are disengaged to allow the spindle to be rotated by hand, without too much effort. The spindle is then rotated, manually, by turning the chuck, while at the same time, the reamer is slowly fed into the work by moving the tailstock spindle with the handwheel. A good grade of cutting oil should be used in the performance of this operation.

The compound rest can also be used to bore internal tapers and angles. An angle is being bored inside of a large diameter at A, Fig. 10-12. The diameter is large enough to allow the entire compound rest to enter the bore. At B, Fig. 10-12, an angle is being bored in a smaller hole using the compound rest. In this case a boring bar is used in order to get into the bore. Note how the boring bar is mounted on the compound rest.

Faceplate Work

As a result of their unusual shape, some workpieces cannot be conveniently held in a chuck or between the lathe centers, and faceplates are used to hold them. Often, considerable thought and planning must be used in order to determine the best way to set up the workpiece on the faceplate. Care and skill must be exercised to make certain that the part is correctly aligned on the faceplate so that the surfaces that are to be machined will be in the desired location. Faceplate work is so diversified that it can best be learned by a careful study of some typical examples. In this chapter the examples and discussions are closely related to the illustrations. The application of faceplates in obtaining an accurate hole location will be treated in detail in Chapter 15.

Faceplate Reference Surfaces

The faceplate is a reference surface that is perpendicular to the axis of rotation of the lathe. Work can be attached directly to the faceplate with the assurance that it is resting on a surface that is at right angles to that axis. Reasonable care must be exercised to prevent the faceplate from being damaged, thereby destroying its accuracy. Sometimes a faceplate does not run true on the lathe spindle. In this event a facing cut should be taken with the faceplate mounted on the spindle nose, as shown in Fig. 11-1. New faceplates should always have a truing cut taken across the face before they are used. The facing cut should be taken with a broad-nose, cast-iron finishing tool such as is shown in Fig. 11-1. The spindle speed of the lathe is determined by the outside diameter of the faceplate and it should be based on a cutting speed of about 50 fpm. The depth of cut should be just enough to "clean up" the surface of the faceplate. The compound-rest slide can be used to set the cutting tool to the depth of cut; to do this it is positioned to move parallel to the lathe axis. The feed rate for taking the broad-nose, finishing cut should be approximately .9 times the width of the face of the tool. Care must be exercised in positioning the cutting edge of the tool parallel to the cross-feed movement, otherwise an undesirable groove will be cut on the faceplate. This entire operation should be done in one or two shallow cuts (.001 to .003-inch deep) across the faceplate.

An angle plate, Fig. 11-5, can be attached to the faceplate to provide a reference surface that is parallel to the lathe axis. The angle plate must

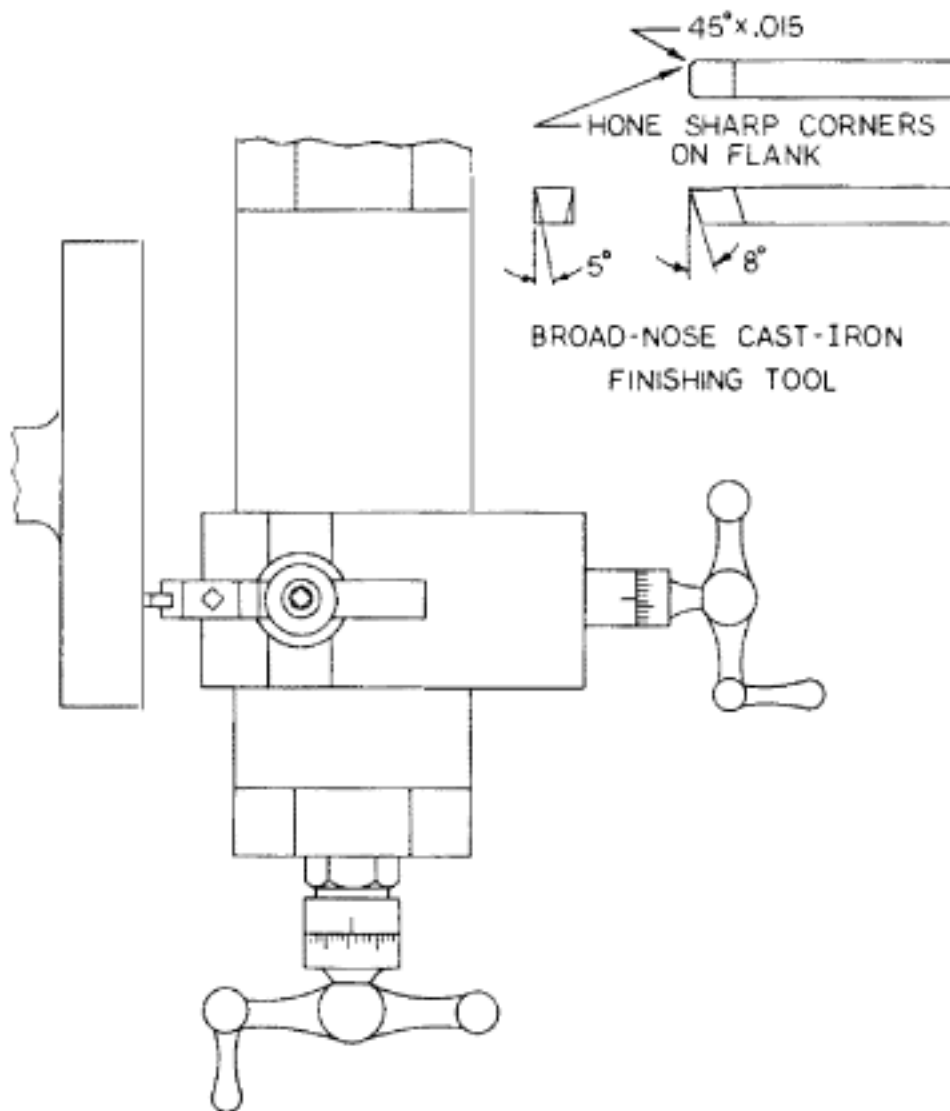
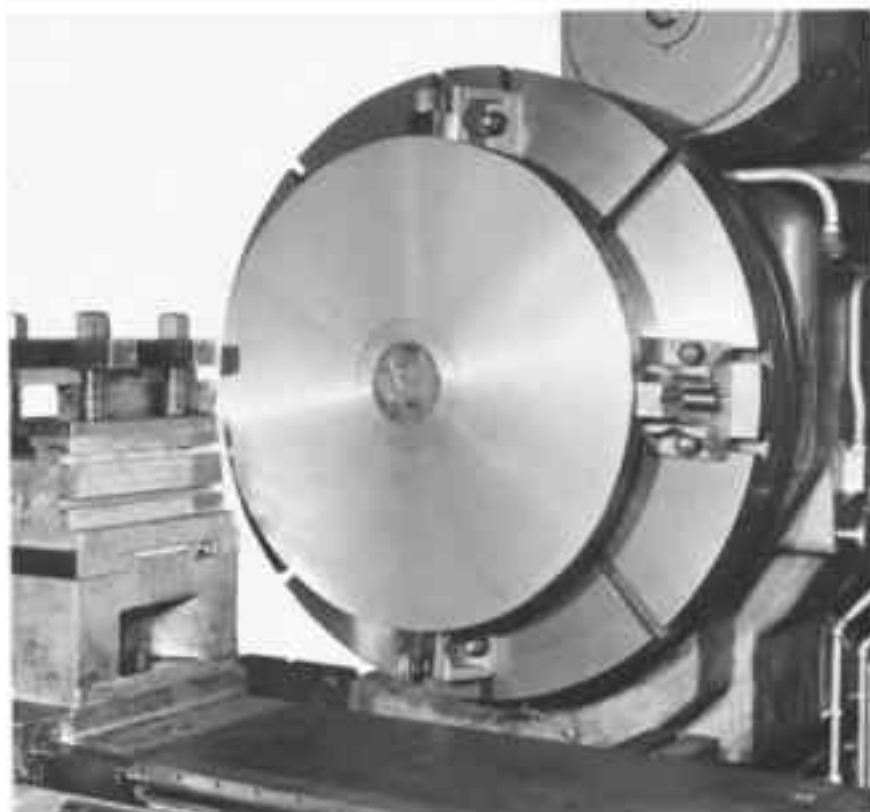


Fig. 11-1. Truing the surface of a faceplate by taking a facing cut with a broad-nose, cast-iron finishing tool.

be square (this should be verified with a precision square); if it is not, it should be machined square on a planer or a shaper. Sometimes angle plates having angles other than 90 degrees are used. In either case the workpiece is attached to the angle plate.

Faceplate Jaws

Faceplate jaws for clamping workpieces are bolted onto the faceplate to provide movable jaws similar to a four-jaw independent chuck. Faceplate jaws are shown in Figs. 11-2 and 8-5. They are used primarily on larger faceplates that are mounted on large lathes in place of four-jaw independent chucks which, in the larger sizes, would add an unnecessarily heavy weight to the nose of the spindle. A chuck would also place the weight of the workpiece further away from the support of the spindle



Courtesy of LeBlond Machine Tool Co.

Fig. 11-2. Faceplate jaws used to hold workpieces for facing operation.

bearings, which is undesirable. This is avoided when using faceplate jaws which are just as effective as a chuck in holding the workpiece. Faceplate jaws can be used to hold symmetrical parts, as shown in Fig. 11-2. They can also be used to hold irregularly shaped parts, as the different jaws can be located at any distance from the center of the faceplate.

Faceplate Setups

The illustrations in this chapter are used as examples of the application of faceplate work on engine lathes. Although the same setup is not likely to be encountered, there is frequently a similarity between the jobs that are illustrated in this chapter and those that are encountered elsewhere. For this reason the examples shown should be carefully analyzed.

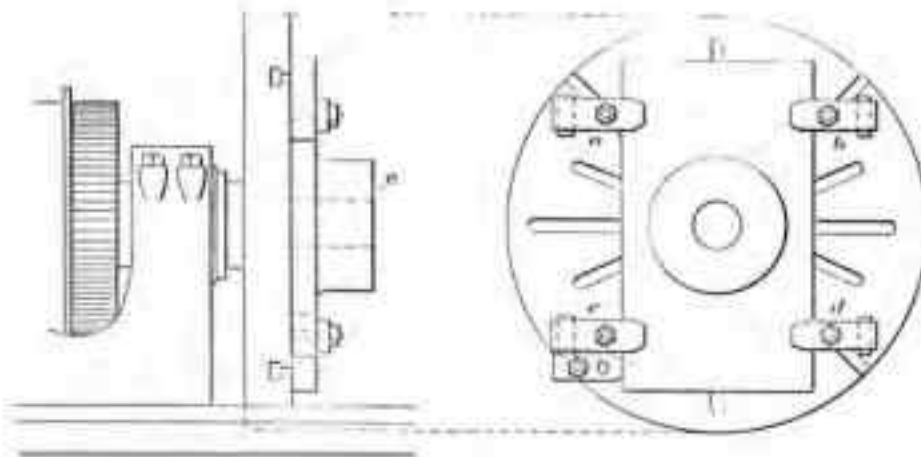
Figure 11-3 shows a rectangular-shape casting with a round boss, or projection, the end of which is to be turned parallel with the finished back face of the casting. A cored hole through the center of the boss is also to be bored true. This is a typical example of a part that can be conveniently machined by bolting it directly to the faceplate of the lathe. To perform this operation in the lathe, clamp the finished surface of the casting directly against the faceplate by bolts and clamps, *a*, *b*, *c*, and *d*, as shown. The work would then be turned in the same way as if it were held in a chuck. By holding the casting in this way, the face, *e*, will

be finished parallel with the back surface, because the latter is clamped directly against the true-running surface of the faceplate. If a casting of this shape were small enough it could also be held in the jaws of a four-jaw independent chuck, but if the surface, *e*, needs to be exactly parallel with the back face, or if the bore must be exactly perpendicular to this surface, it is better to clamp the work to the faceplate.

The proper way to clamp a piece to the faceplate depends largely upon its shape, the location of the surface to be machined, and the location of any previously machined surfaces. Whenever possible, previously machined surfaces should be clamped against the faceplate as they provide reference surfaces from which the work can be aligned. When the work is bolted to an angle plate instead of directly to the faceplate, the previously machined surfaces should be placed against the angle plate. If a hole is to be bored through a workpiece which is larger than the bore in the spindle nose, the workpiece should be clamped against parallel bars, and not directly against the faceplate, to provide clearance for the tool when it reaches the inner end of the hole and to prevent it from cutting the faceplate. The parallels should be located directly underneath the clamps so that the clamping forces will not bend the workpiece.

When deep roughing cuts have to be taken, especially on large diameters, it is well to bolt a piece to the faceplate and against one side of the workpiece, as at *D*, Fig. 11-3, to act as a driver and to prevent the work from shifting. A faceplate driver is always placed to the rear, as determined by the direction of rotation, since the work tends to shift backwards when a cut is being taken. If the surface which is clamped against the faceplate is finished, the work will be less likely to shift when a piece of paper is placed between it and the faceplate.

Sometimes, an assembly of parts is mounted on a faceplate for turning, boring, and facing operations. When machined in this manner, practically perfect alignment can be obtained between all of the parts in the assembly. Figure 11-4 illustrates boring and facing operations being performed on an assembly of parts which is held on the faceplate.



Courtesy of LeBlond Machine Tool Co.

Fig. 11-3. Casting clamped to faceplate for turning and boring.



Courtesy of LeBlond Machine Tool Co.

Fig. 11-4. Boring and facing operations performed on an assembly of parts that are clamped directly to a faceplate.

An angle plate is used in conjunction with a faceplate, for holding elbows or other parts that cannot be clamped against a vertical surface. Figure 11-5 illustrates how an angle plate is used to hold a cast-iron elbow, *E*, the two flanges of which are to be faced true and square with each other. The shape of the casting is such that a flange cannot be held on the faceplate while the other flange is being machined. The angle plate, *P*, provides the reference surface against which a flange can be bolted

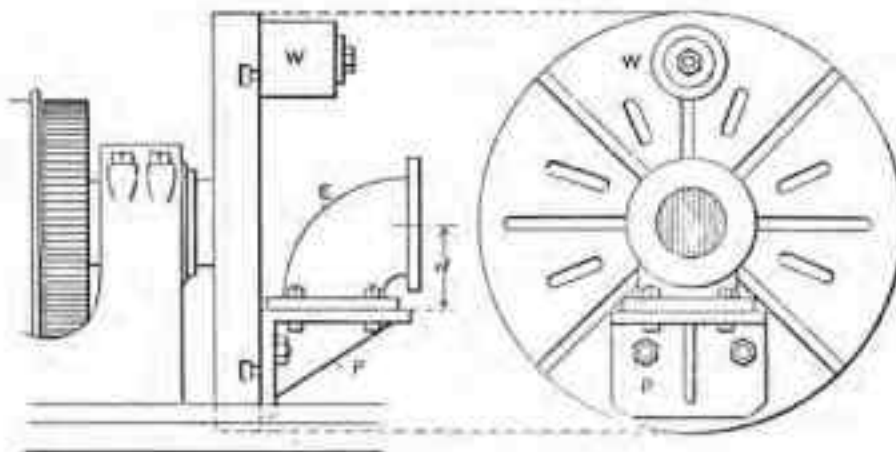
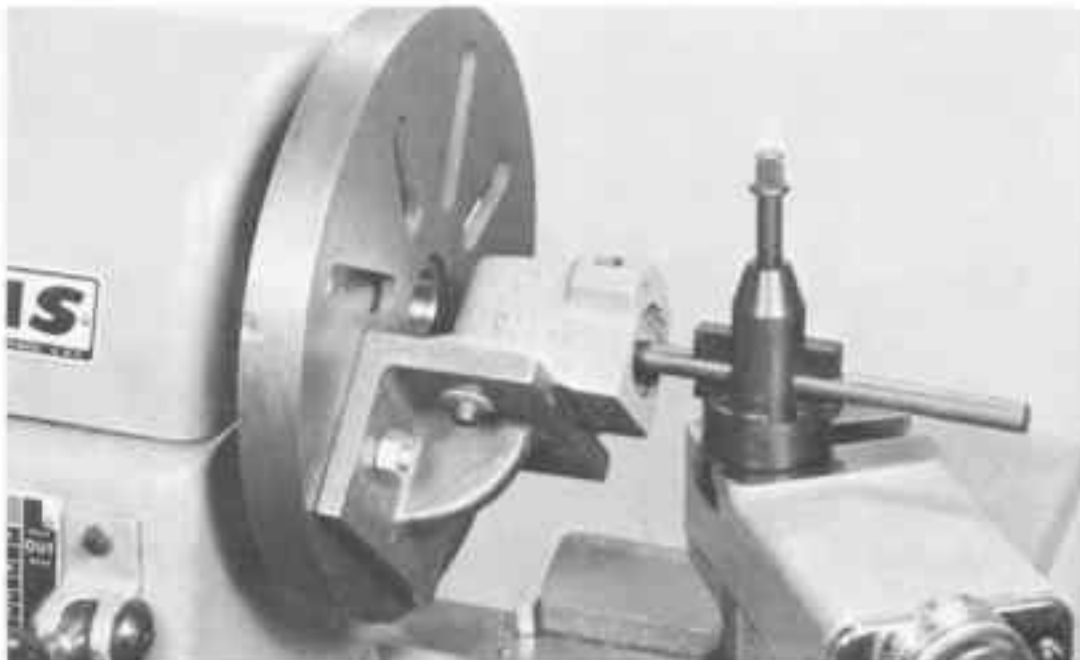


Fig. 11-5. Cast elbow held on angle plate attached to faceplate.

while the other flange is machined at right angles to it. When setting up an angle plate for this operation, the distance from its work-holding side to the center of the faceplate is made equal to the distance, d , between the center of one flange and the face of the other.

As the angle plate and the work are almost entirely on one side of the faceplate, a weight, W , is attached to the opposite side for counterbalancing. Very often, weights are also needed to counterbalance offset parts that are bolted directly to the faceplate. The necessity for counterbalancing depends somewhat upon the spindle speed used for turning. If the surface to be machined is small in diameter so that the lathe can be run quite fast, any unbalanced part should always be counterbalanced. Likewise, if the counterbalancing is not perfect, which is often the case, the spindle speed must be reduced to avoid excessive vibrations. The faceplate and the parts attached to it can be counterbalanced so that no vibrations result from this assembly. It is important, however, to understand that even when balanced, heavy workpieces will place a very severe load on the clamps due to the action of centrifugal force. Thus, the speed of the spindle should be limited when heavy, unbalanced masses are encountered, even if these masses are balanced by counterweights. Balance of the faceplate assembly is achieved only when the force tending to move the counterweights is equal and opposite to the force tending to move the workpiece.

Figure 11-6 illustrates a part being held and bored while clamped to an angle plate that is attached to the faceplate. This is a convenient method of boring and facing parts that cannot otherwise be clamped and located



Courtesy of the Clamping Division of the Atlas Press Company

Fig. 11-6. Boring a part that is clamped to an angle plate attached to the faceplate.

in the lathe. The base of this part has been previously machined and is used as a locating surface for holding the part on the angle plate. The setup for this and for similar operations is usually made in two stages. First, the angle plate is set up so that the perpendicular distance from the clamping surface of the angle plate to the axis of the lathe is equal to the distance from the reference surface on the work to the center of the bored hole. Second, the workpiece must be located along the angle plate so that the axis of the hole to be machined (or surface to be turned) coincides with the axis of the lathe.

The procedure for setting up the workpiece is illustrated in Figs. 11-7 and 11-8. A layout should be made on the workpiece before starting to make the setup on the lathe. The layout lines on the workpiece can be seen in Fig. 11-8. These layout lines define two imaginary planes which intersect at the axis of the hole. The setup is started by clamping the

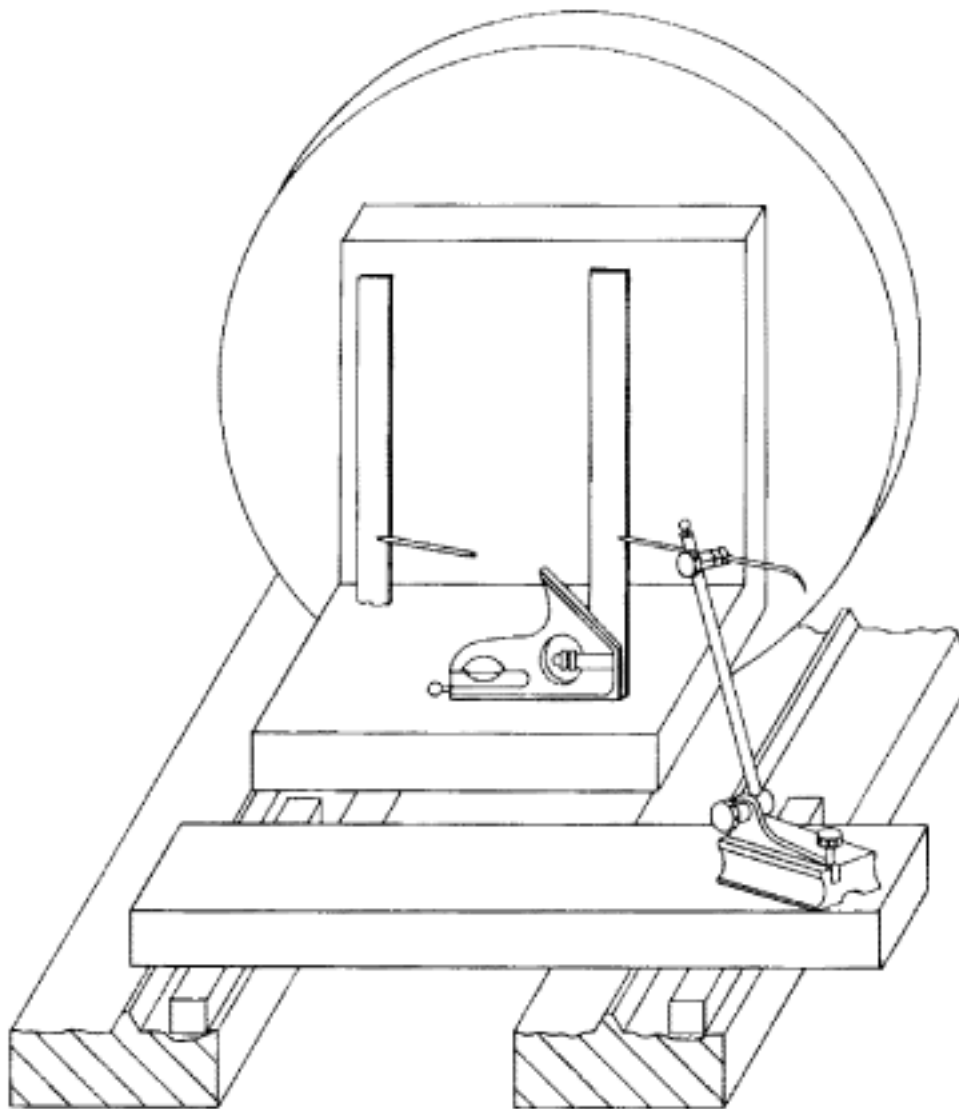


Fig. 11-7. Setting up the angle plate on the faceplate of the lathe.

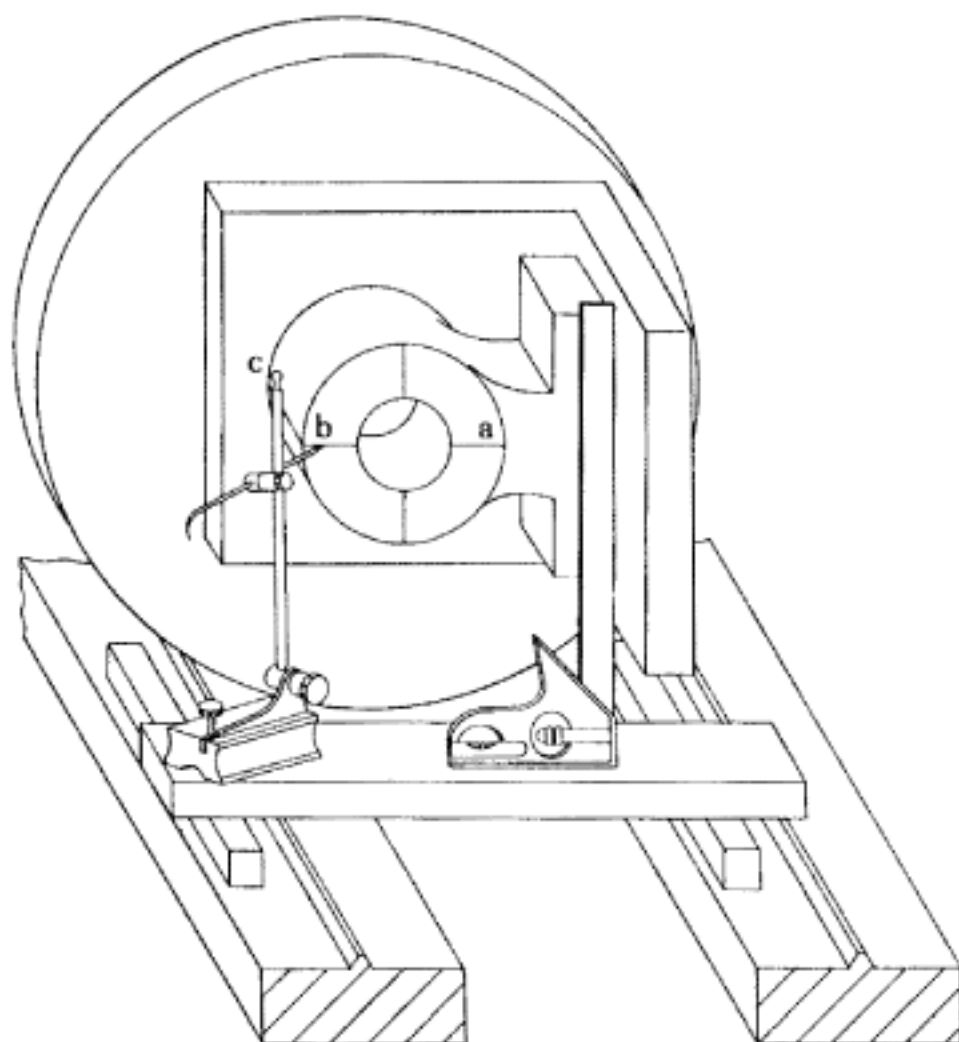
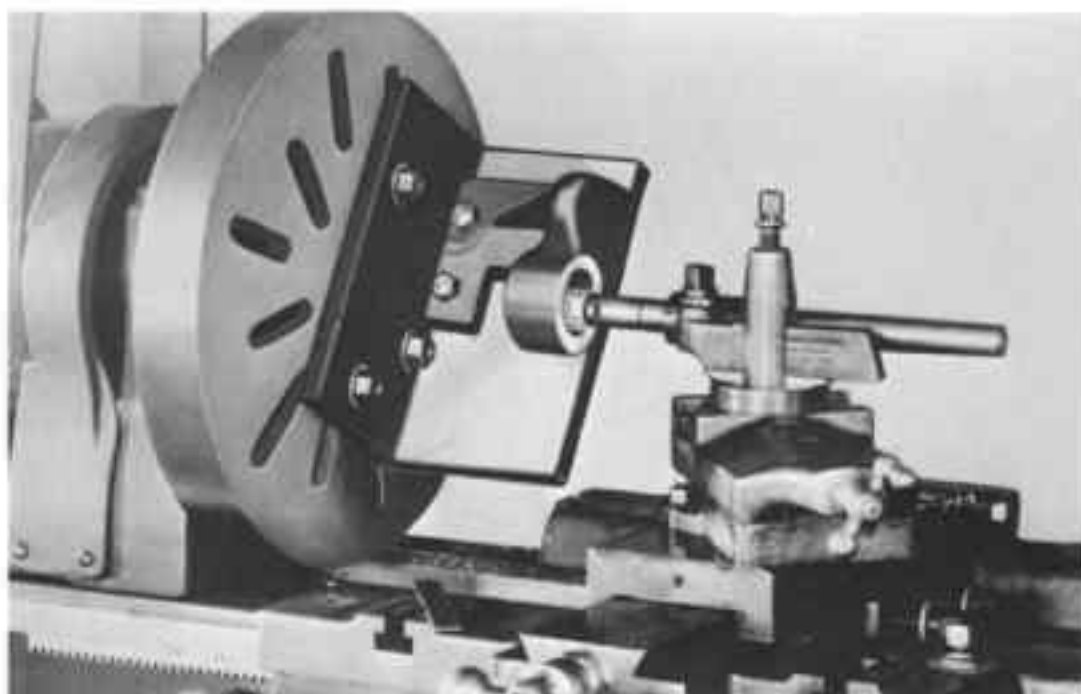


Fig. 11-8. Aligning the workpiece on the angle plate.

angle plate to the faceplate in its approximate position. The angle plate is clamped firmly, yet loose enough so that it can be moved about on the faceplate by tapping with a hammer against a piece of soft metal which is held against the angle plate. The lathe spindle is turned to bring the clamping surface of the angle plate into a horizontal position, as shown in Fig. 11-7, and precision parallel bars are then used to form a bridge over the ways of the bed. With the surface gage placed on the parallels, the scriber point of the surface gage is set at the height of the lathe axis by making it correspond to the point of a lathe center held in the tailstock spindle. A combination square is placed on the clamping surface of the angle plate, as shown. The distance that this surface falls below the lathe axis can then be read directly, by bringing the scriber point against the graduations on the combination square rule. This height must be checked on both sides of the angle plate, as shown. The readings in both of these positions must be equal otherwise correct distance from the lathe axis to the angle plate is not obtainable. The angle



Courtesy of South Bend Lathe

Fig. 11-9. Boring a bracket mounted to an angle plate which is held on a faceplate.

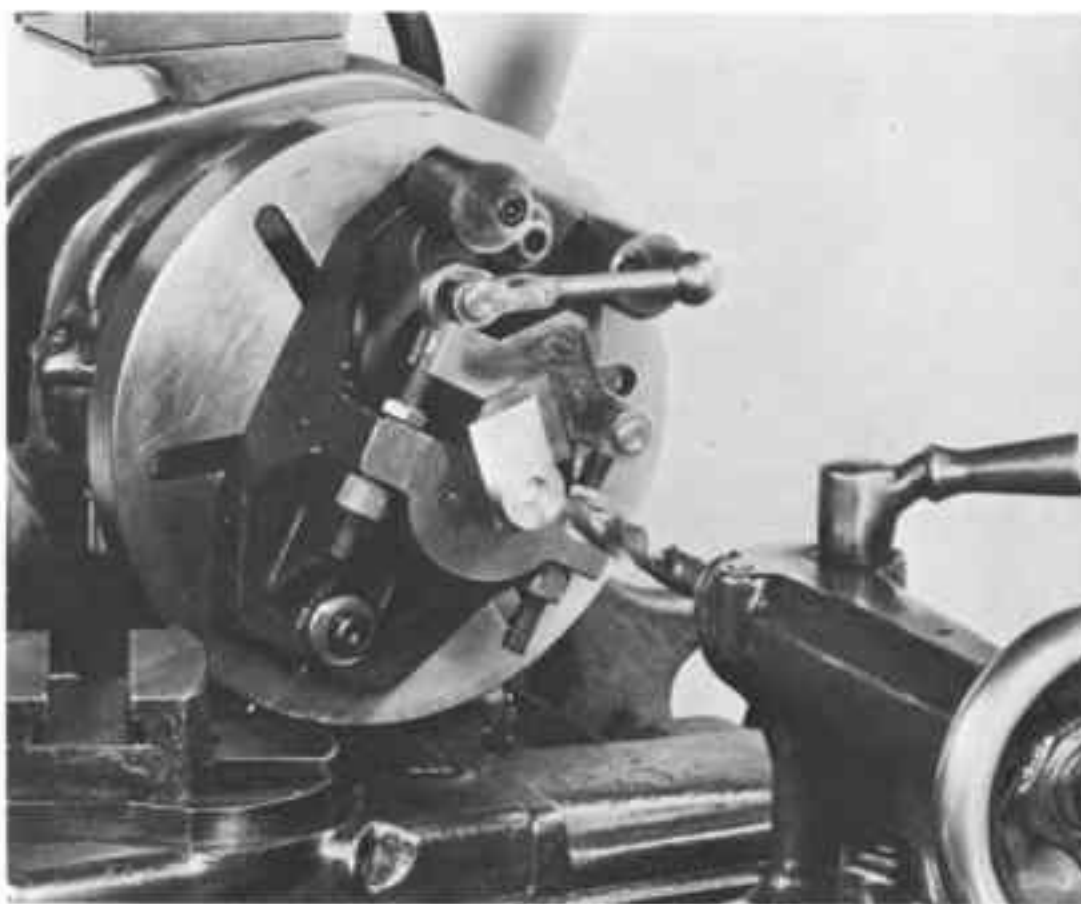


Courtesy of Lassy Tools Inc.

Fig. 11-10. V-block angle plate attached to faceplate and used to hold T-shaped part.

plate is then adjusted until this distance is correct, after which it is securely clamped to the surface plate. The workpiece is then clamped to the angle plate in the approximate position in which it will be machined. The lathe spindle is turned until the work-holding surface of the angle plate is in a vertical position, as shown in Fig. 11-8, using a combination square to check this position. The scriber point of the surface gage is positioned at the height of the lathe centers as before. The location of the workpiece is then adjusted on the angle plate until the layout line, *abc*, is at the height of the surface-gage scriber point at *a*, *b*, and *c*. When in this position the setup is completed by tightening the clamps that hold the work to the angle plate. It would have been possible to locate the position of the angle plate on the faceplate by aligning the layout line that is perpendicular to *abc* with the surface gage; however, the method of doing this, described previously, is usually more accurate and, therefore, preferred.

Another setup for boring a part that is held on an angle plate is shown in Fig. 11-9. This angle plate is similar to those shown in Figs. 11-7 and 11-8. It is different, however, from those shown in Figs. 11-5 and 11-6

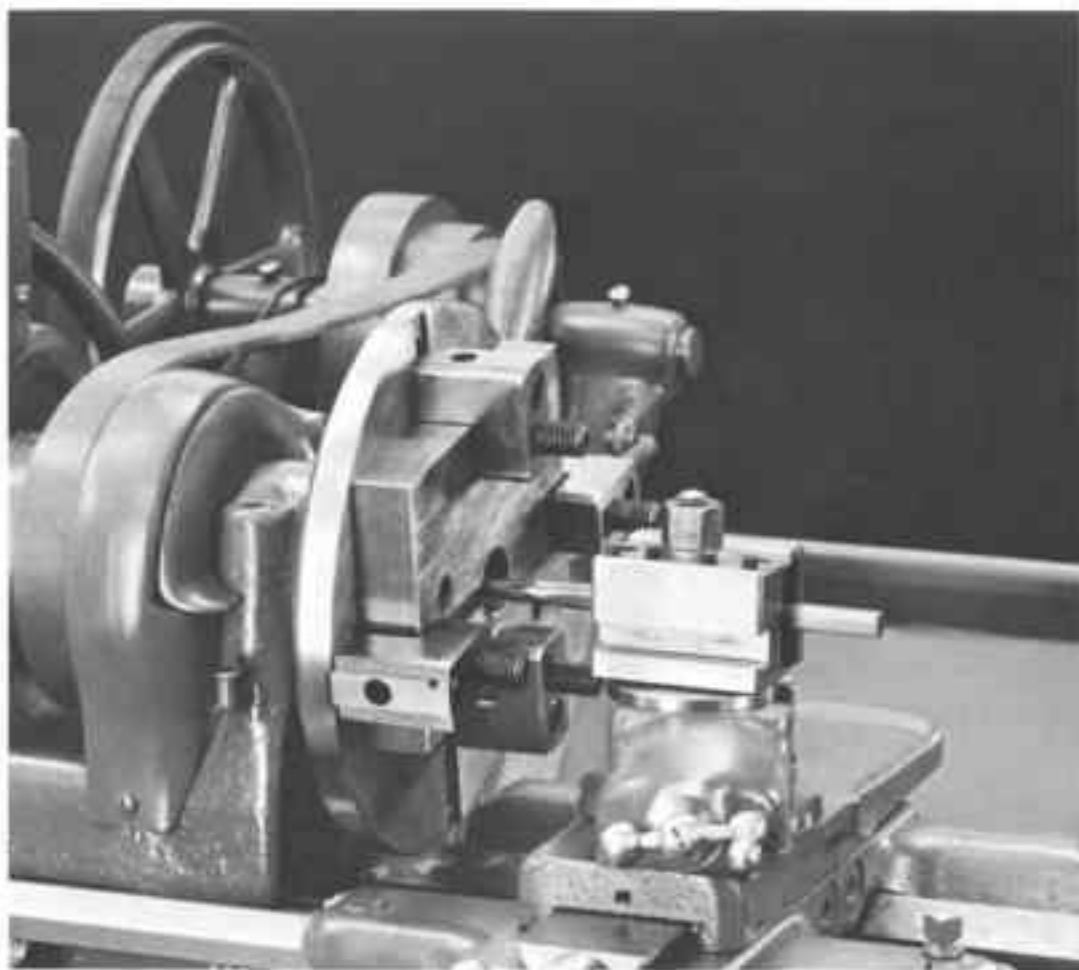


Courtesy of Lanny Tools Inc.

Fig. 11-11. V-block angle plate with special jaws used to hold part for drilling, facing, and turning.

in respect to the location of the work-holding surface. The angle plate in Fig. 11-9 has the work-holding surface on the inside face of the angle plate, while the angle plates in Figs. 11-5 and 11-6 have the work-holding surface on the outside face. In lathe work, having this surface on the inside face has the advantage of allowing larger workpieces to be held.

A very convenient fixture for faceplate work is the V-block angle plate shown in Figs. 11-10 and 11-11. This angle plate has two surfaces attached to the body, that are 90 degrees to each other—forming a V-locating surface. The fixed V-locator in the body is perpendicular to the face of the angle plate. An adjustable clamp, which also has two surfaces in the form of a V, is used to secure the part. The position of this fixture on the angle plate may be adjusted to accommodate the size and form of different workpieces. In Fig. 11-10, the clamp is shown in the open position. The workpiece in this illustration is a T-shaped part that would be very difficult to hold in any other manner. Note that the clamp is actuated by a cam. Special V-shaped jaws are used to hold the part in the V-block angle plate in Fig. 11-11. Two counterweights have been attached



Courtesy of South Bend Lathe

Fig. 11-12. Boring a jig plate with a faceplate chuck.

to this angle plate in order to allow higher spindle speeds to be used in drilling this part. The part in Fig. 11-11 is being drilled, turned, and faced in one setup.

A faceplate chuck made from a standard faceplate is shown in Fig. 11-12. Radial slots are on the face of the faceplate in which the workholding clamps are placed. A series of holes is drilled and tapped in the center of each slot in which the socket-head cap screws that hold the clamps are screwed. Thus, the clamps can be placed in a number of positions on the faceplate. These clamps do not hold the sides of the workpiece, which is a part of a drill jig; they clamp it against the faceplate.

Quite often it is necessary to use two separate setups to bore concentric holes located in the ends of a part. The principle by which the two holes can be bored concentric to each other is illustrated in Fig. 11-13. This principle can be extended to suit a variety of different workpiece configurations and sizes. First one of the holes is accurately bored to size and the end is faced to provide a surface on which the workpiece can be seated on the faceplate. Then a convenient piece of steel is clamped to a faceplate and turned to the size of the bored hole. The turned piece of steel, still in place, now acts as a stub mandrel which is used to locate the workpiece on the faceplate for boring the second hole. With the workpiece mounted on the stub mandrel and clamped to the faceplate, the second hole is bored to size. If only one part is to be machined, the stub mandrel can be discarded after the job is done. If at a later date a like part is to be machined, the stub mandrel is saved and used again by aligning it on the faceplate with a dial test indicator. On repetitive work, the stub mandrel may be a part of a more elaborate lathe fixture.

Lathe fixtures are very often mounted on a faceplate. They may be very complex (see Fig. 14-17), or they may be very simple. On occasion, time may be saved and accuracy assured by making a simple fixture to

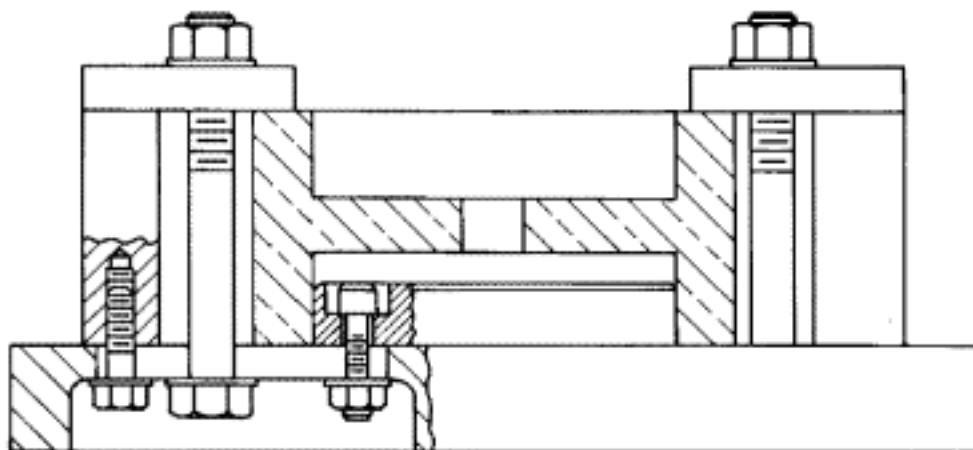


Fig. 11-13. Principle of using a stub mandrel to locate workpiece for boring concentric holes.

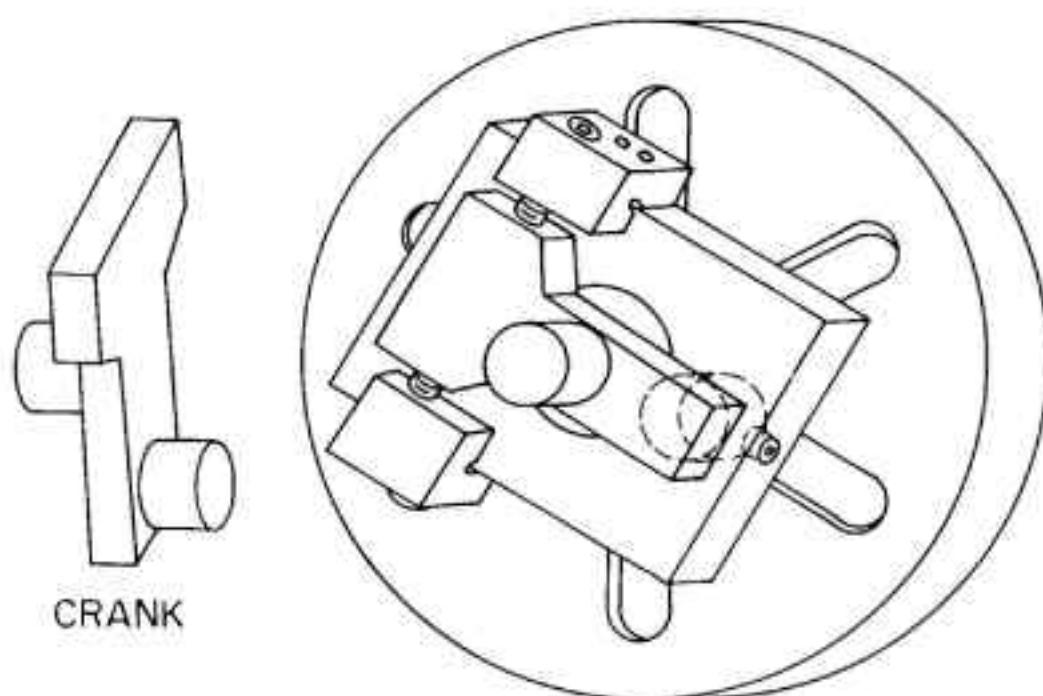
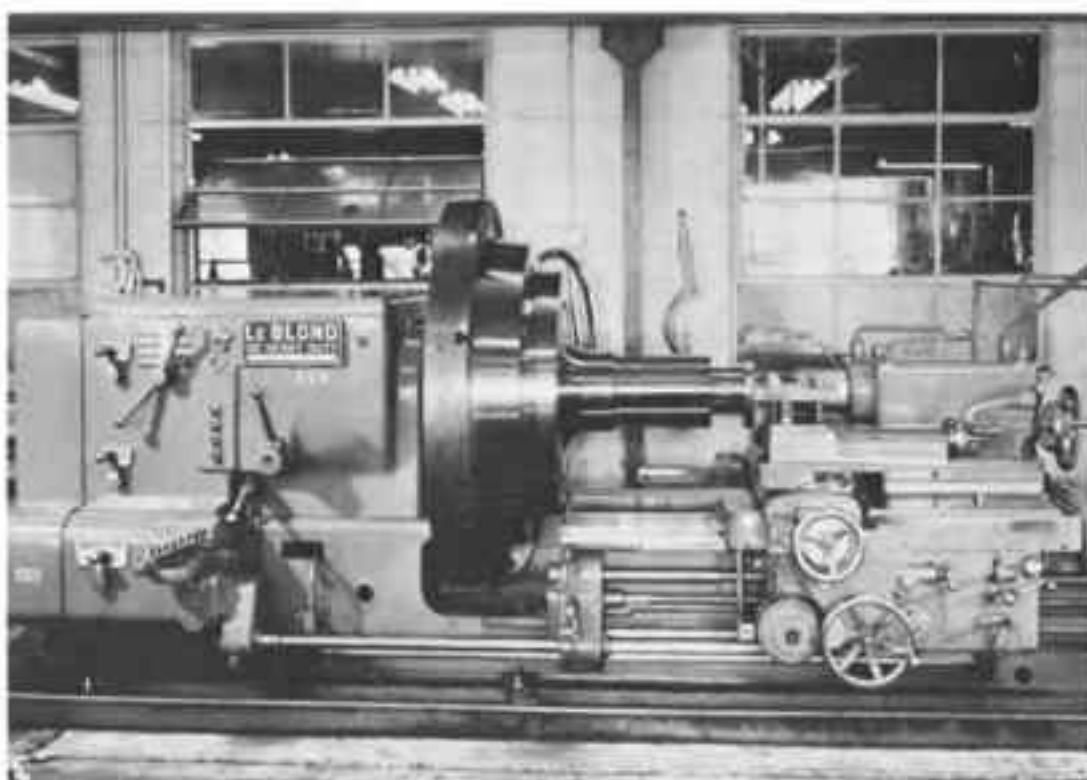


Fig. 11-14. Simple faceplate fixture for turning the main bearing journal of a crank. Fixture is located on faceplate by indicating large hole with a dial test indicator.



Courtesy of LeBlond Machine Tool Co.

Fig. 11-15. Large workpiece with one end mounted on a faceplate and other end supported on a tailstock center.



Courtesy of "Model Engineer" magazine

Fig. 11-16: Machining a model steam-locomotive driving-wheel which is mounted on a faceplate.

machine only one or two parts. As an example, the fixture in Fig. 11-14 is used to machine the main bearing journal and the adjacent face of the crank after the crankpin has been machined to size. The fixture is bolted to the faceplate. A large hole that has previously been bored in the fixture

is used to locate it on the faceplate by indicating the bore with a dial test indicator. The center distance between this hole and the smaller hole used to locate the crankpin is equal to the throw of the crank. The crankpin is located in the small hole in the fixture and clamped in place by a setscrew after the main bearing journal has been centered by the two screws that clamp against the sides of the counterbalance. The main bearing journal can then be turned to size; the fixture will locate the crank so that its throw will be correct.

Longer workpieces that are mounted on a faceplate must have additional support. In some cases a steadyrest can be used to provide the additional support, or the end of the workpiece may be supported by the tailstock center, as shown in Fig. 11-15. By supporting the workpiece by the tailstock center, the entire surface is available for machining. The lathe in Fig. 11-15 is a large gap-bed lathe.

The faceplate is an almost indispensable accessory to a lathe in the fascinating work of model engineering. For example, in Fig. 11-16, a model steam locomotive driving wheel is being turned while mounted on



Courtesy of "Model Engineer" magazine

Fig. 11-17. Facing a model steam-engine trunk-column casting.



Courtesy of "Model Engineer" magazine

Fig. 11-18. Facing the flange of a curved pipe which is clamped to an angle plate mounted on a faceplate.



Courtesy of "Model Engineer" magazine

Fig. 11-19. Using two angle plates mounted on a faceplate to hold a small casting.

the faceplate of a model maker's engine lathe. In this manner, perfect concentricity and perpendicularity between the turned surfaces are assured. A model steam engine trunk column casting is machined while attached to a faceplate in Fig. 11-17. The flange of a curved pipe is faced in Fig. 11-18 while attached to an angle plate that is mounted on a faceplate. A very ingenious setup is shown in Fig. 11-19, where two angle plates are used to hold the workpiece.

Screw Threads and Screw-Thread Measuring

The majority of screw threads used in machine construction conform to some established standard. These screw-thread standards include the profile or cross-sectional shape of the thread, a range of screw thread diameters, and a standard number of threads per inch for each diameter. Tolerances and allowances for obtaining varying degrees of accuracy and different classes of fits are also included in modern screw-thread standardization. Special screw threads are used when there is some mechanical reason for doing so. There are two basic kinds of screw threads—locking or holding threads that are used on fasteners to hold parts together, and power-transmission threads that are used to transmit power and motion. Sometimes locking threads are used to transmit motion when the power required is low. When this is done, however, there is considerable loss in mechanical efficiency in comparison to threads designed to transmit power.

Definitions of Screw-Thread Terms

The definitions of the principal terms relating to screw threads are given below (see Fig. 12-1):

Major Diameter—The largest diameter of a straight screw thread. The term major diameter applies to both internal and external threads.

Minor Diameter—The smallest diameter of a straight screw thread. The term minor diameter applies to both external and internal threads.

Pitch—The distance from a point on a screw-thread profile to a corresponding point on the next thread profile, measured parallel to the axis. The pitch is equal to one divided by the number of threads per inch, or:

$$\text{Pitch} = \frac{1}{\text{No. Threads per inch}}$$

Pitch Diameter—The diameter of an imaginary cylinder, the surface of which would pass through the threads on all sides at such points as to make the width of the groove equal to one-half of the pitch. The pitch diameter is independent of the major and minor diameters. It is a measure of the diameter of the sloping sides of the thread profile. As such, it deter-

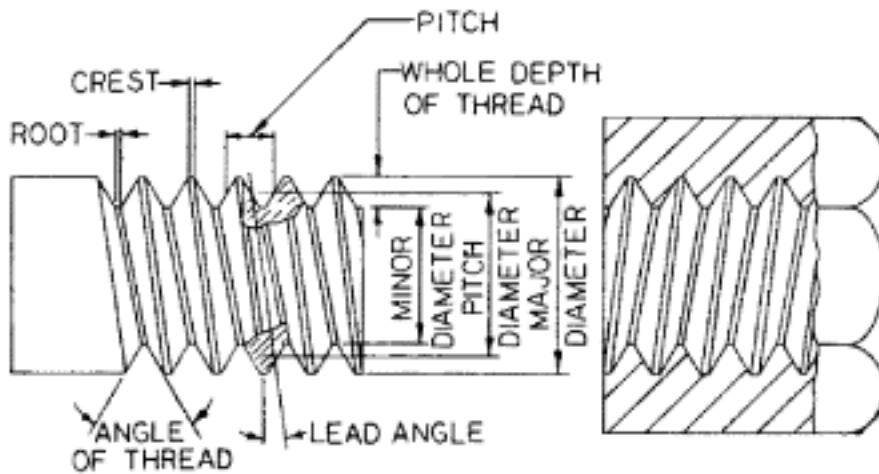


Fig. 12-1. Illustration showing screw-thread terms.

mines the size of the thread and how closely, or loosely, it fits onto its mating thread.

Nominal Size—The designation used to describe the size of the thread for purposes of general identification. For example, the nominal size of a $\frac{1}{2}$ -inch, 20-thread per inch, screw thread is $\frac{1}{2}$ inch, but its actual major diameter for Class 2A limits might be from .4987 to .4906 inch.

Actual Size—The actual measured size.

Basic Size—The theoretical size from which the size limits are derived by the application of allowance and tolerance.

Design Size—The size from which the limits of size are derived by the application of tolerance.

Tolerance—The difference between the maximum and minimum limits of size, or the total permissible variation in size.

Allowance—An intentional difference between the maximum material limits of mating parts. It is the minimum clearance between freely fitting parts, or the maximum interference between parts having an interference fit. (in ISO terminology it is: *Fundamental Deviation*.)

Limits of Size—These are the maximum and the minimum sizes.

Maximum Material Limit—A maximum material limit of size is the maximum limit of size of an external dimension or the minimum limit of size of an internal dimension.

Minimum Material Limit—A minimum material limit of size is the minimum limit of size of an external dimension or the maximum limit of size of an internal dimension.

Fit—The relationship existing between two mating parts with respect to the amount of clearance or interference between them when they are assembled.

Basic Form of Thread—The theoretical profile of the thread for a length of one pitch in the axial plane. It is the form on which the design forms of both external and internal threads are based.

Design Form of Thread—The design forms for a thread are the maximum material forms permitted for external and internal threads.

Lead—The distance that a screw thread advances axially in one turn. On a single thread screw the pitch and the lead are equal; on a double thread screw the lead is twice the pitch; on a triple thread screw the lead is three times the pitch, etc.

Angle of Thread—The angle included between the sides of the thread measured in an axial plane.

Lead Angle—This is the angle made by the helix of the thread at the pitch diameter, measured in an axial plane.

Crest—The top surface joining the two sides, or flanks, of the thread.

Root—The bottom surface joining the two adjacent sides or flanks of the thread.

Flank—The flank (or side) of a thread surface connecting crest and roots.

Truncation—The truncation of the root or crest of a thread is the distance, measured perpendicular to the axis, between a sharp V-shaped root or crest (apex) and the cylinder which bounds the root or crest.

Depth of Engagement—The depth of thread contact or overlap of two mating thread forms, measured radially.

Crest Clearance—In a thread assembly this is the distance, measured perpendicular to the axis, between the crest of a thread and the root of its mating thread.

Right-Hand Thread—A thread with the grooves cut so that a machine screw or a bolt would have to be turned in a clockwise direction, when viewed from the head end, in order to assemble into a threaded hole.

Left-Hand Thread—A thread with the grooves cut so that a machine screw or a bolt would have to be turned in a counterclockwise direction, when viewed from the head end, in order to assemble into a threaded hole.

Multiple Thread—A thread having two or more separate grooves cut adjacent to each other and winding around the screw in the manner of a helix. The two adjacent threads combine to produce the basic form of the thread.

Standard Screw-Thread Forms and Systems

The purpose of this section is to study the basic screw-thread forms and the formulas used to obtain their principal dimensions. Screw-thread systems include, in addition to the basic screw-thread forms, specifications for thread per inch corresponding to a given diameter, classes of fits, and basic dimensions for each thread series. These details are found in engineering handbooks such as *Machinery's Handbook*.

American Standard Unified-Thread Form. The former American National Form of Thread was used for many years in making most of the locking type threads produced in the United States. This form is practically the same as the present standard thread form used in the United States, which is called the American Standard Unified-Thread Form (Fig. 12-2). These two thread forms are completely interchangeable although

can National Thread Form has a flat crest and root. The exact form of the rounding is not specified in the American Standard which, in practice, varies according to tool wear. The width of the flat at the crest and the root is the same for both thread forms, being $.125 \times$ pitch. According to British specifications the root on the internal threads is rounded to a radius of $.10825 \times$ pitch and on the external threads, to a radius of $.14434 \times$ pitch. The preferred American practice is to have a flat crest, while the preferred British practice is to have a rounded crest. The crest of the internal thread is flat. The angle of the thread is 60 degrees.

The American Standard Unified Thread System—This is the system agreed upon by the United States, Canada, and the United Kingdom in

Table 12-1. Coarse-Thread Series, UNC and NC—Basic Dimensions

Size	Basic Major Diam., D	Thds. per Inch, n	Basic Pitch Diam., E	Minor Diameter		Lead Angle at Basic P.D.	Area of Minor Diam. at $D-2a_2$	Tensile Stress Area
	Inches		Inches	Ext. Thds., K_e	Int. Thds., K_i	Deg. Min.	Sq. In.	Sq. In.
1 (.073)*	0.0730	64	0.0628	0.0635	0.0561	4 31	0.00218	0.00263
2 (.094)	0.0940	56	0.0744	0.0841	0.0667	4 22	0.00310	0.00370
3 (.099)*	0.0990	48	0.0856	0.0734	0.0784	4 26	0.00406	0.00487
4 (.112)	0.1120	40	0.0959	0.0813	0.0849	4 45	0.00496	0.00604
5 (.125)	0.1250	40	0.1088	0.0943	0.0979	4 11	0.00672	0.00796
6 (.138)	0.1380	32	0.1177	0.0987	0.1042	4 50	0.00745	0.00909
8 (.164)	0.1640	32	0.1437	0.1257	0.1302	3 58	0.01196	0.0140
10 (.190)	0.1900	24	0.1629	0.1389	0.1449	4 29	0.01450	0.0175
12 (.216)*	0.2160	24	0.1889	0.1649	0.1799	4 1	0.0206	0.0242
$\frac{1}{4}$	0.2500	20	0.2175	0.1887	0.1959	4 11	0.0269	0.0318
$\frac{3}{8}$	0.3125	18	0.2764	0.2443	0.2524	3 40	0.0454	0.0524
$\frac{1}{2}$	0.3750	16	0.3344	0.2983	0.3073	3 24	0.0678	0.0775
$\frac{3}{4}$	0.4375	14	0.3911	0.3499	0.3602	3 20	0.0933	0.1083
$1\frac{1}{4}$	0.6000	13	0.4500	0.4056	0.4167	3 7	0.1257	0.1419
$1\frac{1}{2}$	0.6625	12	0.5084	0.4603	0.4723	3 59	0.162	0.182
$1\frac{3}{4}$	0.6250	11	0.5660	0.5136	0.5206	2 58	0.202	0.226
$2\frac{1}{4}$	0.7500	10	0.6660	0.6273	0.6417	2 40	0.302	0.334
$2\frac{1}{2}$	0.8750	9	0.8029	0.7387	0.7547	2 31	0.419	0.462
1	1.0000	8	0.9188	0.8466	0.8647	2 29	0.551	0.606
$1\frac{1}{8}$	1.1250	7	1.0322	0.9497	0.9784	2 31	0.693	0.763
$1\frac{1}{4}$	1.2500	7	1.1672	1.0747	1.0864	2 16	0.890	0.969
$1\frac{3}{8}$	1.3750	6	1.2867	1.1706	1.1946	2 24	1.054	1.155
$1\frac{1}{2}$	1.5000	6	1.3917	1.2655	1.3106	2 11	1.294	1.405
$1\frac{3}{4}$	1.7500	5	1.6201	1.5046	1.5335	2 15	1.74	1.90
2	2.0000	4 $\frac{1}{2}$	1.8657	1.7274	1.7594	2 11	2.30	2.50
$2\frac{1}{8}$	2.3600	4 $\frac{1}{2}$	2.1057	1.9774	2.0084	1 55	3.02	3.25
$2\frac{1}{4}$	2.5000	4	2.3376	2.1933	2.2294	1 57	3.72	4.00
$2\frac{3}{8}$	2.7500	4	2.5876	2.4433	2.4784	1 46	4.62	4.93
3	3.0000	4	2.8376	2.6933	2.7294	1 38	5.62	5.97
$3\frac{1}{8}$	3.2500	4	3.0876	2.9433	2.9784	1 29	6.72	7.10
$3\frac{1}{4}$	3.5000	4	3.3376	3.1933	3.2294	1 22	7.92	8.33
$3\frac{3}{8}$	3.7500	4	3.5876	3.4433	3.4784	1 16	9.21	9.60
4	4.0000	4	3.8376	3.6933	3.7294	1 11	10.61	11.08

* Secondary sizes.
 * British: Effective Diameter.

order to obtain screw thread interchangeability among these nations. The transition of industry from the use of the former American National thread system to the use of the unified threads is practically completed.

American Standard Unified Thread Series—Unified threads consist of a Coarse-Thread Series, Table 12-1; a Fine-Thread Series, Table 12-2; and an Extra-Fine Thread Series. There are also 4-, 6-, 8-, 12-, 16-, 20-,

Table 12-2. Fine-Thread Series, UNF and NF—Basic Dimensions

Size	Basic Major Diam., D	Thds. per Inch, n	Basic Pitch Diam., E	Minor Diameter		Lead Angle at Basic P.D.		Area of Minor Diam. at $D-2a_s$	Tensile Stress Area
	Inches		Inches	Ext. Thds., K_s	Int. Thds., K_n				
				Inches	Inches	Deg. Min.	Sq. In.	Sq. In.	
0 (.000)	0.0000	80	0.0519	0.0447	0.0465	4 33	0.00151	0.00160	
1 (.073)*	0.0730	72	0.0640	0.0560	0.0580	3 57	0.00237	0.00278	
2 (.086)	0.0860	64	0.0759	0.0668	0.0691	3 45	0.00339	0.00394	
3 (.099)*	0.0990	56	0.0874	0.0771	0.0797	3 43	0.00451	0.00523	
4 (.113)	0.1129	48	0.0985	0.0864	0.0894	3 51	0.00566	0.00661	
5 (.125)	0.1250	44	0.1102	0.0971	0.1004	3 45	0.00716	0.00830	
6 (.138)	0.1380	40	0.1218	0.1073	0.1109	3 44	0.00874	0.01015	
8 (.164)	0.1640	36	0.1400	0.1299	0.1339	3 38	0.01285	0.01474	
10 (.190)	0.1900	32	0.1607	0.1517	0.1562	3 21	0.0175	0.0200	
12 (.216)*	0.2160	28	0.1928	0.1722	0.1773	3 22	0.0226	0.0258	
$\frac{1}{8}$	0.2500	28	0.2268	0.2067	0.2113	2 52	0.0320	0.0364	
$\frac{3}{16}$	0.3125	24	0.2854	0.2614	0.2676	2 40	0.0524	0.0580	
$\frac{1}{4}$	0.3750	24	0.3478	0.3239	0.3299	2 11	0.0809	0.0878	
$\frac{5}{16}$	0.4375	20	0.4060	0.3762	0.3834	2 15	0.1090	0.1187	
$\frac{3}{8}$	0.5000	20	0.4673	0.4387	0.4459	1 57	0.1480	0.1599	
$\frac{7}{16}$	0.5625	18	0.5284	0.4943	0.5024	1 55	0.189	0.203	
$\frac{1}{2}$	0.6250	18	0.5889	0.5568	0.5649	1 43	0.240	0.256	
$\frac{9}{16}$	0.7500	16	0.7094	0.6733	0.6823	1 36	0.351	0.373	
$\frac{5}{8}$	0.8750	14	0.8286	0.7874	0.7977	1 24	0.480	0.509	
1	1.0000	12	0.9450	0.8978	0.9098	1 36	0.625	0.663	
$1\frac{1}{8}$	1.1250	12	1.0709	1.0228	1.0348	1 25	0.812	0.856	
$1\frac{1}{4}$	1.2500	12	1.1959	1.1478	1.1598	1 16	1.024	1.073	
$1\frac{3}{8}$	1.3750	12	1.3209	1.2728	1.2848	1 9	1.260	1.315	
$1\frac{1}{2}$	1.5000	12	1.4450	1.3978	1.4098	1 3	1.621	1.581	

* Secondary sizes.

* British: Effective Diameter.

28-, and 32-thread series, the pitch in each of these cases being constant for all diameters. The Coarse Series is recommended for general use where conditions do not require a fine thread. The Fine, and Extra-Fine, Series are particularly useful in the automotive and aircraft industries. The Fine Series is also used where the length of engagement of the threads is short, where a smaller lead angle is desired, and where the wall thickness demands a fine pitch. The 8-thread series was originally intended for high-pressure-joint bolts and nuts but is now widely used as a substitute for the Coarse-Thread Series, for diameters larger than one inch. The 12-thread series was originally intended for boiler practice. It is now

used extensively as a continuation of the Fine-Thread Series, for diameters larger than $1\frac{1}{2}$ inches. The 16-thread series is used for adjusting collars, bearing retaining nuts, and other applications requiring a fine thread. It also provides a continuation of the Extra-Fine-Thread Series for diameters larger than $1\frac{1}{16}$ inch. The 20-, 28-, and 32-thread series have been used more or less widely in industry for various applications where the Standard Coarse, Fine, or Extra-Fine Series could not be used. They are now given recognition as Standard Unified Thread Series in a specified selection of diameters for each pitch.

Each thread series has been given a symbol for easy identification. These are given below:

UNC	Unified Coarse Series
UNF	Unified Fine Series
UNEF	Unified Extra-Fine Series
UN	Used for all constant-pitch series, which are given as follows: 4UN, 6UN, 8UN, 12UN, 16UN, 20UN, 28UN, and 32UN
UNS	Special threads designed through applications of formulas given in the Unified Standard, but not given in this text.

Thread Classes—Thread classes are used to specify the fit between mating threads and the tolerance to which the thread must be made. The detailed specification for each thread is to be found in engineering handbooks; however, it is helpful to have a general understanding of the difference between the classes because of the wide usage of the terms associated with this subject. The classes are identified by a numeral followed by a letter A and B. The numeral designates the class of fit, while the A designates an external thread and the B designates an internal thread.

Classes 1A and 1B—These classes are intended for ordnance work and other special applications where quick and easy assembly is necessary. They have a liberal allowance to permit the parts to assemble easily, even with slightly bruised or dirty threads. The tolerances specified for the 1A, or external, threads are not the same as those specified for the 1B, or internal, threads.

Classes 2A and 2B—These classes are the most commonly used for general applications including bolts, nuts, screws, and similar fasteners. This series provides for a *minimum clearance* between mating threads to permit the threads to be assembled without being excessively loose, but also without binding or seizing.

Classes 3A and 3B—These classes have closer tolerances than those provided by Classes 2A and 2B. The maximum diameter of a Class 3A, or external, thread is equal to the minimum diameter of a Class 3B, or internal, thread. Therefore, no clearance or allowance is provided for the assembly of maximum metal components and mating threads will bind or fit very tightly at assembly.

The selection of a screw-thread class depends upon the end use of the threads. In almost all cases a given class of thread will be available to provide required fit. When the requirements for screw-thread fit cannot be met by a single class, it is possible to specify a proper combination of thread classes for the components. For example, a class 2A external thread may be used with a class 1B, 2B, or 3B internal thread.

Screw Thread Designation—Standard thread designations are given on drawings and specifications in order to specify a given thread. These specifications are frequently encountered on machine drawings, and their meaning should be committed to memory. The standard method of designating a screw thread is to specify, in sequence, the nominal size, number of threads per inch, thread series symbol, and thread-class symbol. For example:

$$\frac{1}{4}\text{-}20\text{ UNC-}2\text{A}$$

This designation specifies a thread having a nominal size of $\frac{1}{4}$ inch, 20 threads per inch, and that it is a Coarse-Thread Series having a class 2A tolerance for external threads. The pitch-diameter limits may also be designated, but are not required. The thread in the above example may be specified as follows:

$$\frac{1}{4}\text{-}20\text{ UNC-}3\text{A} \\ \text{PD } 0.2175\text{-}0.2147$$

The additional designation specifies that the limits of size of the pitch diameter are .2175 to .2147 inch.

It is occasionally necessary to modify the limits of size of the major diameter of an external thread or the minor diameter of an internal thread from the limits in the standard in order to fit a special purpose, but without changing the class of the thread or the pitch diameter. In such cases the established thread designation is used which is followed by a statement of the modified diameter limits and the designation "MOD." For example:

$$\frac{3}{4}\text{-}10\text{ UNC-}2\text{A} \\ \text{Major Diameter } 0.740\text{-}0.735\text{ MOD}$$

American National Thread System—There are some older drawings used in machine shop practice on which threads are specified according to the American National Thread System; therefore, a brief description of thread specifications in accordance to this system will be given. The designations used are: NC, NF, N-8, N-12, and N-16 which indicate National Coarse, National Fine, 8-pitch, 12-pitch, and 16-pitch series respectively. There are three classes of fits of which Class 1 is the least exacting, Class 2 is recommended for general use, and Class 3 is the most exacting. The thread is designated by specifying in sequence, the nominal size, number of threads per inch, thread-series symbol, and thread-class symbol. For example, a $\frac{1}{2}$ inch, 13 threads per inch thread having a Class 2 fit would be specified as follows:

$$\frac{1}{2}\text{-}13\text{ NC-}2$$

American Standard for Unified Miniature Screw Threads—This standard introduces a new thread series that is intended for general purpose fastening screws and similar uses on watches, instruments, and other mechanisms. The basic profile of this thread is identical with the Unified, and American, basic thread form except for the basic height and depth of engagement which is $.52 \times \text{pitch}$ instead of $.54127 \times \text{pitch}$. The included angle of the thread is 60 degrees and the thread-series symbol is UNM. There are fourteen sizes, and one class of thread which has a zero allowance on all diameters. The basic sizes and the pitches for this thread are metric, although the engineering tables give the equivalent inch measurements as well as the metric.

American Standard Microscopic Objective Threads—This thread system is based on the thread system originated by the Royal Microscopical Society of Great Britain, known as the RMS thread. This system is used for microscopic objective mountings and for other optical assemblies of microscopes and associated apparatus. The basic thread form is like the British Standard Whitworth thread having a 55-degree angle of thread and rounded threads and roots.

Interference Fit Threads—Interference-fit threads are holding threads in which the externally threaded member is larger than the internally fitted member. They are assembled by expanding the internal thread with the application of heat or by shrinking the external thread by subjection to a low temperature. At assembly they become the same size and develop a great holding power by elastic compression of the members. By custom, these threads are designated as Class 5.

American Standard Acme Screw Threads—The American Standard for Acme Screw Threads provides for three types of Acme threads, namely General Purpose, Stub, and Centralizing. General Purpose Acme threads have sufficient clearance on all diameters for free movement whereas Centralizing Acme threads have the clearance limited at the major diameters of the external and internal threads so that the threads can contact each other at these surfaces in order to maintain an approximate alignment of the thread axis, thereby preventing the thread flanks from wedging and causing the thread to bind. There is also an alternate American Standard for Centralizing Acme threads that has the clearance limited on the minor diameter instead of on the major diameter which then acts as a centralizing control. The Centralizing Acme thread has a particular advantage in supporting long, unsupported screws such as the lathe lead screw. The Stub Acme thread is used for unusual applications where, due to mechanical or metallurgical considerations, a coarse-pitch thread of shallow depth is required.

The thread form for General Purpose Acme threads and for Stub Acme threads is shown in Fig. 12-3. The angle of thread is 29° . The crest corners may, as an option, be chamfered to an angle of 45° and a maximum depth

of $\frac{P}{15}$, where P is the pitch. For Standard Acme threads, the recommended combinations of pitch diameter and threads per inch are: $\frac{1}{4}$ -16; $\frac{3}{8}$ -14;

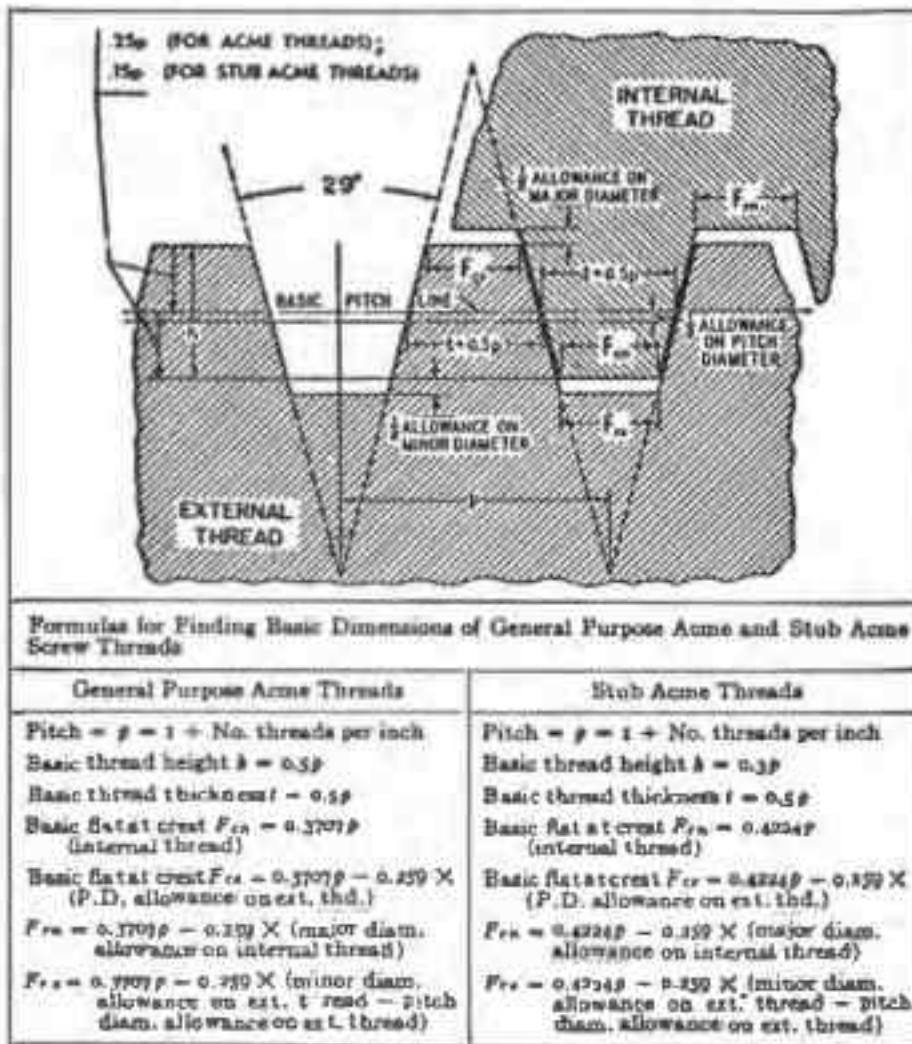


Fig. 12-3. American Standard General Purpose Acme Screw Thread Form and American Standard Stub Acme Screw Thread Form.

$\frac{3}{8}$ -12; $\frac{7}{16}$ -12; $\frac{1}{2}$ -10; $\frac{5}{8}$ -8; $\frac{3}{4}$ -6; $\frac{7}{8}$ -6; 1-5; $1\frac{1}{8}$ -5; $1\frac{1}{4}$ -5; $1\frac{3}{8}$ -4; $1\frac{1}{2}$ -4; $1\frac{3}{4}$ -4; 2-4; $2\frac{1}{4}$ -3; $2\frac{1}{2}$ -3; $2\frac{3}{4}$ -3; 3-2; $3\frac{1}{2}$ -2; 4-2; $4\frac{1}{2}$ -2; 5-2. Other combinations can, however, be used for special applications. Three classes of General Purpose threads are provided in the standard. The Class 2G is the preferred class for most general-purpose thread assemblies. If less backlash, or end play, between the nut and the screw is required, the Class 3G or Class 4G can be used. The following examples show how Acme threads are designated on drawings and tools:

1. $1\frac{3}{4}$ -4 Acme-2G indicates a General Purpose Class 2G Acme thread of $1\frac{3}{4}$ inch major diameter and 4 threads per inch, single thread, right hand. The same thread, but left hand, would be designated $1\frac{3}{4}$ -4 Acme-2G-LH.
2. $2\frac{7}{8}$ -0.4p-0.8L Acme-3G indicates a General Purpose Class 3G Acme thread of $2\frac{7}{8}$ -inches major diameter, pitch .4 inch, lead .8 inch, double thread, right hand.

3. $\frac{1}{2}$ -20-Stub Acme indicates a $\frac{1}{2}$ -inch major diameter, 20 threads per inch, right hand, single thread, Standard Stub Acme Thread.

The Acme thread is extensively used in preference to the square thread because it is stronger and it may be cut more readily with a die or a tap than a square thread. The split nut on the lathe apron may be more readily engaged on an Acme thread; therefore, the Acme thread is preferred on lathe lead screws.

Square Thread—A square thread is shown in Fig. 12-4. The square thread has not been standardized and has been superseded, to a large extent, by the Acme thread. The pitch of a square thread is usually made twice the pitch of an American Standard thread of corresponding diameter. The thread groove of a square-threaded screw is $.5 \times \text{pitch}$. The groove in the nut is made slightly larger than one-half the pitch in order to provide a clearance for the screw. There is also a 10-degree Modified Square Thread which has an angle of thread equal to 10 degrees. This thread is practically equivalent to a square thread and yet is capable of economical production.

American Standard Taper Pipe Threads—This thread has an angle of thread of 60 degrees when measured in an axial plane, and the line bisecting this angle is perpendicular to the axis. The depth of the pipe thread is greater than the American Standard Unified Thread Form and it equals $.8 \times \text{pitch}$ except for 8 threads per inch which is $.788 \times \text{pitch}$. The tapering pipe thread has a standard taper of $\frac{3}{4}$ inch per foot measured on the diameter and along the axis. For each size thread the length of the thread and the length of engagement of the two mating threads must be held within specified limits. These threads are designed to be self-holding and self-sealing when properly assembled. It is sometimes necessary to turn or to bore a pipe thread with a single-point tool on an engine lathe; however, most pipe threads are cut with a die or with a tap. The thread designation for American Standard Pipe Taper Threads is NPT. A typical thread specification would read $\frac{3}{8}$ -18-NPT.

There are other pipe threads that are widely used for specialized applications. These include: USA Standard Hose Coupling Threads; National Standard Fire Hose Coupling Screw Thread; American Standard Pipe Threads for Mechanical Joints; American Standard Dryseal Taper Pipe Threads for Pressure-Tight Joints; and British Standard Pipe Threads.

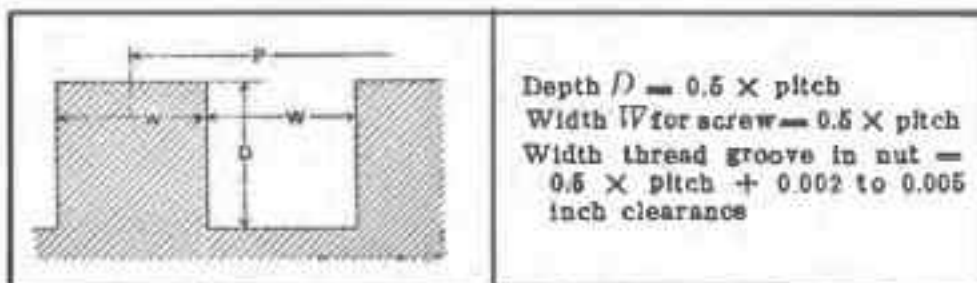


Fig. 12-4. Square Thread.

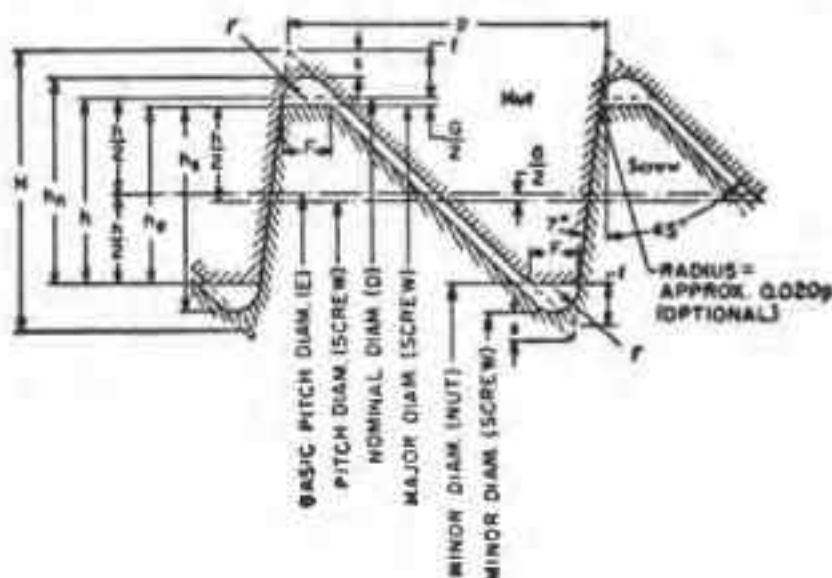


Fig. 12-5. Basic Form of the American Standard Buttress Screw Threads.

Details for the specification of these threads are to be found in the Standards for the specific thread, or in engineering handbooks, such as *Machinery's Handbook*.

American Standard Buttress Screw Thread—The basic form of the thread for the American Standard Buttress Screw Thread is shown in Fig. 12-5. One face, or flank, of this thread is made 7 degrees, with respect to a line perpendicular to the axis, in order to resist heavy loads placed against it. This flank is called the pressure flank of the thread. The opposite flank is 45 degrees, with respect to a line perpendicular to the axis. The basic depth of engagement of the thread is $.6 \times \text{pitch}$. A series of nominal major diameters and a series of preferred pitches are provided in the standard. Since Buttress threads are used mainly on specially designed components, specific recommendations concerning combinations of these preferred diameters and pitches are not made.

Buttress threads are used to withstand heavy axial loads such as encountered in the breech mechanism of large guns and airplane propeller hubs. In the past, Buttress threads were not standardized and, as a consequence, variations in small details of the basic form of the thread were common. A metric standard for this thread, that was used in Germany and Italy, did exist. This thread is called *Stängengewinde* in Germany, and *Fillettatura a Dente di Sega* in Italy.

British Standard Whitworth (B.S.W.) and British Standard Fine (B.S.F.) Threads—The basic form of the British Standard Whitworth Thread is shown in Fig. 12-6. This thread was used in Great Britain for general machine construction. The included angle of the thread is 55 degrees and the crest and root are rounded to a radius of $.137329 \times \text{pitch}$. The depth is equal to $.640327 \times \text{pitch}$. The thread form for the British Standard Whitworth, designated B.S.W., and the British Standard Fine, designated B.S.F., are the same. The B.S.W. series corresponds to the

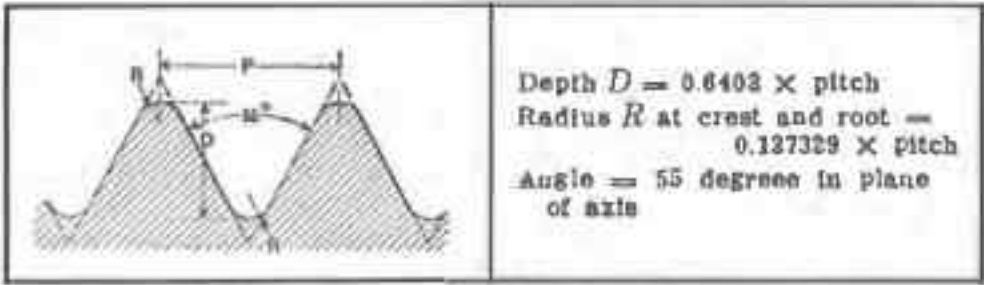


Fig. 12-6. British Standard Whitworth Thread Form.

American coarse series of threads and the B.S.F. corresponds to the fine series. The British Standard specifies a specific number of threads per inch for every nominal diameter.

British Association Standard Thread (B.A.)—This screw-thread system, shown in Fig. 12-7, is recommended for use in preference to the B.S.W. and B.S.F. systems for all screws smaller than $\frac{1}{4}$ inch. The thread has an included angle of 47.5 degrees. The root and the crest of the thread are rounded. The abbreviation B.A. is used to designate British Association Threads.

Metric Threads

The size of the thread profile of all metric threads is specified by the thread pitch. In contrast, the size of the profile of customary inch threads is specified by the threads per inch. Metric threads used in the United States are specified by the American National Standards Institute, or ANSI. To achieve worldwide standardization of screw threads, the ANSI metric thread standard (B1.13) conforms to the ISO metric thread standard (ISO 261), where ISO stands for International Standards Organization. Over 300 metric thread sizes are specified in these standards, their major diameters ranging in size from 1 mm to 300 mm (0.039 in. to 11.811 in.). The thread pitches are named coarse, fine, and extra-fine, corresponding to the coarse, fine, and extra-fine thread series of the customary inch Unified thread system. The standard metric thread sizes

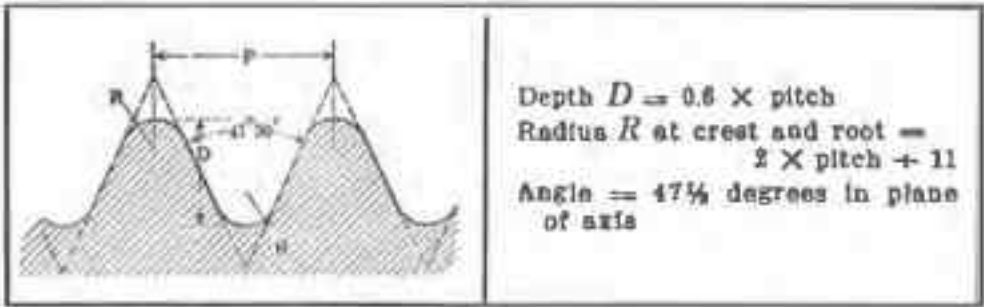


Fig. 12-7. British Association Thread Form.

up to 45 mm (1.772 in.) are given in Table 12-3. For application on screws, bolts, and nuts only 25 thread sizes are recommended by ANSI, which are listed in Table 12-4. Controversy surrounds the M6.3×1 thread, which is listed as non-standard in this table. This thread compares closely to the 1/4-20 customary inch Unified thread, and it has certain other advantages; however, the continued use of the M6.3×1 thread is uncertain.

Table 12-3. ISO Metric Thread Sizes, to 45 mm *

Metric Screw Thread Size	Thread Type	Nominal Major Dia., in.	Closest Approx. thd/in.	Metric Screw Thread Size	Thread Type	Nominal Major Dia., in.	Closest Approx. thd/in.
*M1 × 0.25	Coarse	.039	101.6	M17 × 1	Extra Fine	.669	25.4
M1 × 0.2		.309	127	M18 × 2.5	Coarse	.709	10.2
M1.1 × 0.25		.042	101.6	M18 × 2	Fine	.709	12.7
M1.1 × 0.2		.043	127	M18 × 1.5		.709	16.0
*M1.2 × 0.25	Coarse	.047	101.6	M18 × 1	Extra Fine	.709	25.4
M1.2 × 0.2		.047	127	*M20 × 2.5	Coarse	.787	10.2
M1.4 × 0.3		.056	84.6	M20 × 2	Fine	.787	12.7
M1.4 × 0.2		.061	127	M20 × 1.5		.787	16.0
*M1.6 × 0.35	Coarse	.065	72.6	M20 × 1	Extra Fine	.787	25.4
M1.6 × 0.20		.063	127	M22 × 2.5	Coarse	.866	10.2
M1.6 × 0.35		.071	72.6	M22 × 2	Fine	.866	12.7
M1.6 × 0.2		.071	127	M22 × 1.5		.866	16.0
*M2 × 0.4	Coarse	.070	63.5	M22 × 1	Extra Fine	.866	25.4
M2 × 0.35		.079	101.6	*M24 × 2	Coarse	.945	8.5
M2.2 × 0.45		.087	56.5	M24 × 2	Fine	.945	12.7
M2.2 × 0.25		.087	101.6	M24 × 1.5		.945	16.0
*M2.5 × 0.45	Coarse	.088	56.4	M24 × 1	Extra Fine	.945	25.4
M2.5 × 0.35		.094	72.6	M25 × 2		.984	12.7
M2.5 × 0.5		.118	50.8	M25 × 1.5	Extra Fine	.984	16.0
M2.5 × 0.35		.118	72.6	M25 × 1		.984	25.4
M2.5 × 0.6	Coarse	.138	42.3	M26 × 1.5	Coarse	1.024	16.0
M2.5 × 0.35		.138	72.6	M27 × 2		1.063	8.5
*M3 × 0.7		.102	34.3	M27 × 2	Fine	1.063	12.7
M3 × 0.5		.152	50.8	M27 × 1.5		1.063	16.0
M3.5 × 0.75	Coarse	.177	32.8	M27 × 1	Extra Fine	1.063	25.4
M3.5 × 0.5		.177	50.8	M32 × 2		1.102	12.7
*M3.5 × 0.8		.197	31.8	M32 × 1.5	Coarse	1.102	16.0
M3.5 × 0.5		.197	50.8	M32 × 1		1.102	25.4
*M3.5 × 0.5	Coarse	.216	50.8	*M36 × 3.5	Coarse	1.181	7.3
M3.5 × 1		.236	25.4	M36 × 2		1.181	8.5
M3.5 × 0.75		.250	32.8	M36 × 2	Fine	1.181	12.7
M3.5 × 1		.275	26.4	M36 × 1.5		1.181	16.0
M3.7 × 0.75	Coarse	.275	32.8	M36 × 1	Extra Fine	1.181	25.4
*M3.8 × 1.25		.315	20.3	M36 × 2		1.220	12.7
M3.8 × 1		.315	25.4	M36 × 1.5	Coarse	1.220	16.0
M3.8 × 0.75		.315	32.8	M36 × 2.5		1.220	7.3
M3.8 × 1.25	Coarse	.354	30.2	M36 × 2	Fine	1.220	8.5
M3.8 × 1		.354	25.4	M36 × 2		1.220	12.7
M3.8 × 0.75		.354	32.8	M36 × 1.5	Extra Fine	1.220	16.0
*M3.8 × 1.5		.394	16.9	M36 × 1		1.220	25.4
M3.8 × 1.25	Coarse	.394	20.3	*M38 × 1	Coarse	1.417	8.5
M3.8 × 1		.394	25.4	M38 × 2		1.417	8.5
M3.8 × 0.75		.394	32.8	M38 × 2	Fine	1.417	12.7
M3.8 × 1.5		.433	16.9	M38 × 1.5		1.417	16.0
M3.8 × 1	Coarse	.433	25.4	M38 × 1.5	Extra Fine	1.417	25.4
M3.8 × 0.25		.433	32.8	M38 × 1		1.456	16.0
*M3.8 × 1.75		.472	14.6	M38 × 4	Coarse	1.535	8.5
M3.8 × 1.5		.472	16.9	M38 × 3		1.535	8.5
M3.8 × 1.25	Fine	.472	20.3	M38 × 2	Extra Fine	1.535	12.7
M3.8 × 1		.472	25.4	M38 × 1.5		1.535	16.0
M3.8 × 2		.551	12.7	M40 × 3	Coarse	1.575	8.5
M3.8 × 1.5		.551	16.9	M40 × 2		1.575	12.7
M3.8 × 1.25	Extra Fine	.551	20.3	M40 × 1.5	Extra Fine	1.575	16.0
M3.8 × 1		.551	25.4	*M42 × 4.5		1.654	8.5
M3.8 × 1.5		.591	16.9	M42 × 4	Coarse	1.654	8.5
M3.8 × 1		.591	25.4	M42 × 3		1.654	8.5
*M3.8 × 2	Coarse	.630	12.7	M42 × 2	Extra Fine	1.654	12.7
M3.8 × 1.5		.630	16.9	M42 × 1.5		1.654	16.0
M3.8 × 1		.630	25.4	M45 × 4.5	Coarse	1.772	8.5
M3.8 × 1.5		.669	16.9	M45 × 4		1.772	8.5
				M45 × 3	Fine	1.772	8.5

* ISO first choice thread sizes.

Table 12-4. Thread Sizes Specified by ANSI, for Screws, Bolts, and Nuts, with Comparison to Closest Unified Inch Size

Metric Thread Size			Closest Unified Inch Size		
Metric Thread Size	Major Diameter, inch	Pitch, inch	Unified Thread Size	Major Diameter, inch	Pitch, inch
M1.6 x 0.35	.063	.014	0-80	.060	.013
M2 x 0.4	.079	.016	1-64	.073	.016
M2.5 x 0.45	.098	.019	3-48	.099	.021
M3 x 0.5	.118	.020	5-40	.125	.025
M3.5 x 0.6	.138	.024	6-32	.138	.031
M4 x 0.7	.157	.028	8-32	.164	.031
M5 x 0.8	.197	.031	10-24	.190	.042
M6 x 1	.236	.039	1/4-20	.250	.050
*M6.3 x 1	.248	.039	1/4-20	.250	.050
M8 x 1.25	.315	.049	5/16-18	.313	.056
M10 x 1.5	.394	.059	3/8-16	.375	.063
M12 x 1.75	.472	.069	1/2-13	.500	.077
M14 x 2	.551	.079	9/16-12	.562	.083
M16 x 2	.630	.079	5/8-11	.625	.091
M20 x 2.5	.787	.098	3/4-10	.750	.100
M24 x 3	.945	.118	1-8	1.000	.125
M30 x 3.5	1.181	.138	1 1/8-6	1.188	
M36 x 4	1.417	.157	1 1/4-6	1.438	
M42 x 4.5	1.654	.177	1 3/8-5	1.625	
M48 x 5	1.890	.197	1 7/8-5	1.875	
M56 x 5.5	2.205	.217	2 1/4-4	2.250	
M64 x 6	2.520	.236	2 1/2-4	2.500	
M72 x 6	2.835	.236	2 7/8-4	2.875	
M80 x 6	3.150	.236	3 1/4-4	3.125	
M90 x 6	3.543	.236	3 1/2-4	3.500	
M100 x 6	3.937	.236	4	4.000	

* Not standard. With permission from *World Metric Standards for Engineers*, by E. Kverneland.

Metric Thread Profile. The design profiles of external and internal ISO metric threads are shown in Fig. 12-8. The key to the symbols used in this illustration is given below:

- d = Major diameter of external thread
- D = Major diameter of internal thread
- d_2 = Pitch diameter of external thread
- D_2 = Pitch diameter of internal thread
- d_3 = Minor diameter of external thread
- D_1 = Minor diameter of internal thread
- h_3 = Thread depth of external thread
- H_1 = Thread depth of internal thread
- P = Thread pitch
- R = Thread root radius

Allowance has been defined as the intentional difference in the maximum material limits of two engaged threads; in ISO terminology this is

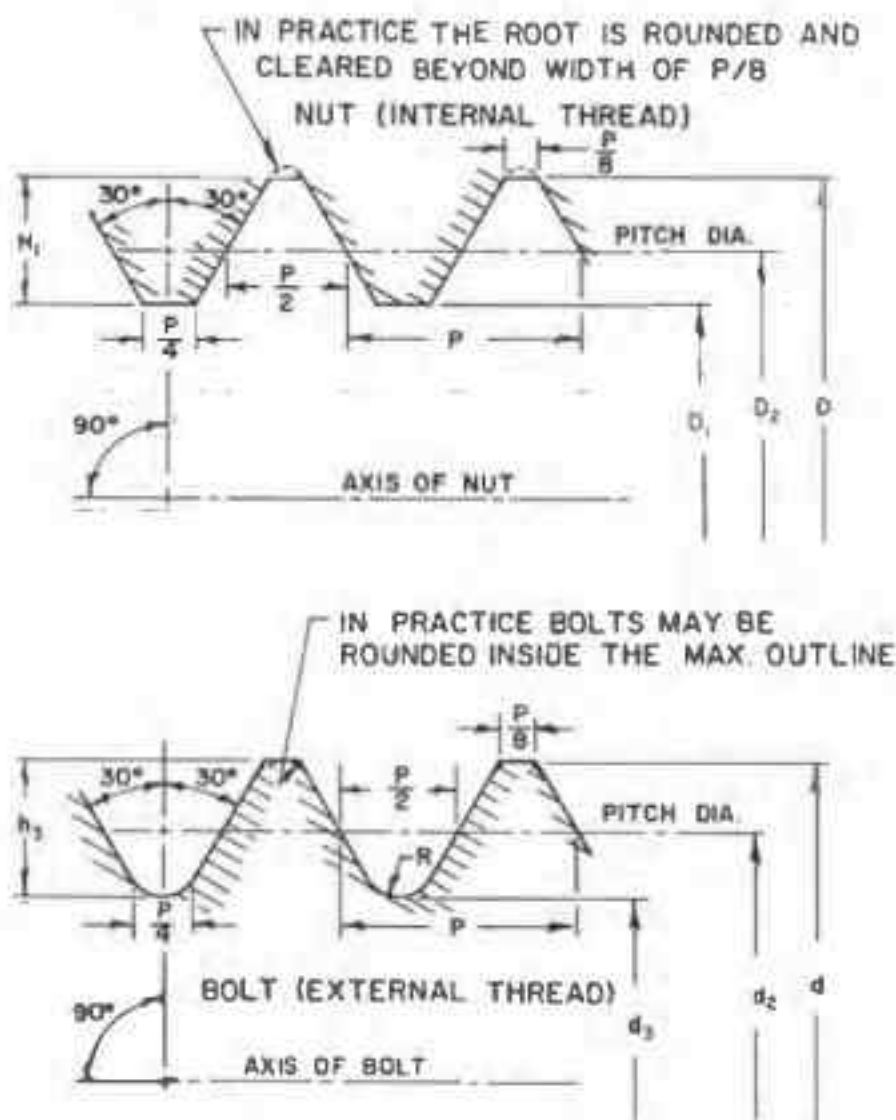


Fig. 12-8. Design forms of ANSI and ISO metric threads.

called the *fundamental deviation*. It is provided to allow the threads to operate freely. The formulas given below can be used to calculate the thread dimensions of ISO metric threads when the external and internal threads are in contact. Whereas the size of the thread depth and the root radius are unaffected by the fundamental deviation, the sizes of the diameters will deviate slightly from the calculated values. The actual thread diameters should be obtained from a table of ISO metric threads, which can be found in handbooks, such as *Machinery's Handbook* or *World Metric Standards for Engineering* by Kvernland.

$$h_a = 0.613435 P$$

$$H_s = 0.541265 P$$

$$R = 0.125 P$$

$$d_2 = D_2 = d - 0.64953 P$$

$$d_3 = d - 1.22687 P$$

$$D_1 = d - 1.08253 P$$

Thread Tolerance Class. In the ISO metric screw thread system, the fundamental deviation is designated by capital letters for internal threads and small letters for external threads. The magnitude of the tolerance is indicated by a number. The combination of the tolerance number and the fundamental deviation letter form a tolerance class designation; e.g., 6g or 6H.

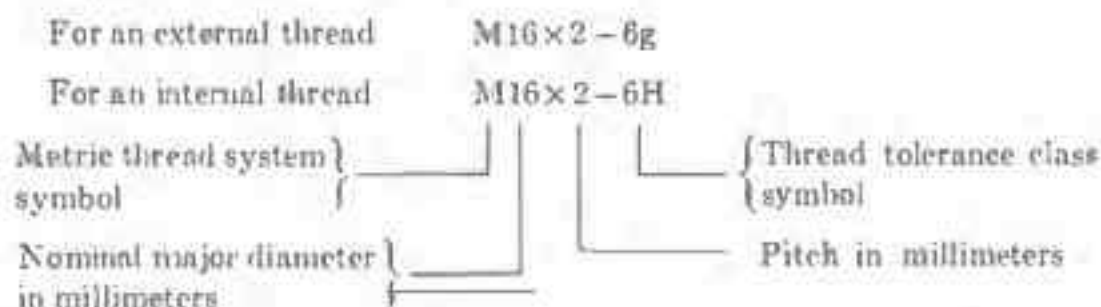
The thread tolerance classes of external and internal threads determine their fit. There are three tolerance classes in general use:

For external threads: 8g, 6g, and 4h (similar to Unified Class 1A, 2A, and 3A)

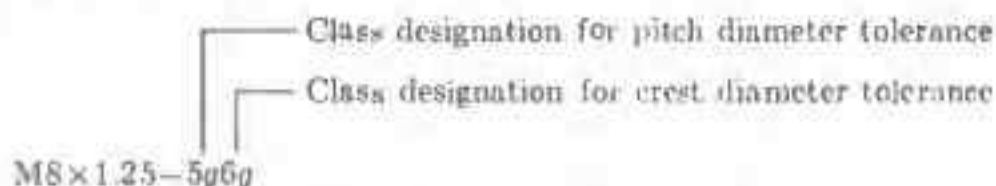
For internal threads: 7H, 6H, and 5H (similar to Unified Class 1B, 2B, and 3B)

The three classes of fits are similar to the Unified Thread Classes 1A/1B, 2A/2B, and 3A/3B, namely 8g/7H, 6g/6H, and 4h/5H, respectively. The medium fit (6H/6g) is approximately equivalent to the Unified Class 2 fit (2A/2B); this fit is used for most screw thread applications.

Metric Thread Designation. The ANSI standard requires that screw threads be specified by a series of symbols which are explained in the example shown below:



The ISO standard for screw thread designation is identical except that for coarse threads the pitch designation is omitted. A thread may be specified to have two tolerance classes, one for the pitch diameter and another for the crest diameter, which is the major diameter of the external thread and the minor diameter of the internal thread. In this event the thread is designated as shown below:



A fit between a pair of threaded parts is indicated by the tolerance design-

nation of the internal thread followed by the tolerance designation of the external thread, separated by a stroke, e.g.,

$$M12 \times 1.75 - 6H/6g$$

Trapezoidal Screw Threads. The ISO metric trapezoidal thread, shown in Fig. 12-9, has an included angle of 30 degrees but is otherwise similar to the Acme thread, which has an included angle of 29 degrees. The design formulas for this thread and the key to the symbols in Fig. 12-9 are given below:

Nominal Diameter	$d = D$ (given in ISO standard)
Thread Pitch	P (given in ISO standard)
Pitch Diameter	$d_2 = D_2 = d - 0.5P$
Minor Diameter (Bolt)	$d_3 = d - 2h_3$
Minor Diameter (Nut)	$D_3 = d - P$
Major Diameter (Nut)	$D_1 = d + 2a_e$
Crest Clearance	a_e (given in ISO standard)
Radius	$R_{1 \text{ max}} = 0.5a_e$
Radius	$R_{2 \text{ max}} = a_e$
Thread Height (Bolt)	$h_3 = 0.5P + a_e$
Thread Height (Nut)	$H_1 = 0.5P + a_e$

These formulas will give the dimensions when the external and internal threads are in contact. In practice the diameters will deviate slightly from the calculated values to allow for the fundamental deviation and for the thread tolerances. The actual values can be found in the ISO standard.

The thread tolerance class designations for ISO metric trapezoidal threads are similar to the designation used for the 60 degree thread previously described. However, the recommended classes are not the same. For threads having a normal length of engagement and a "medium" fit, a 7H/7e fit is recommended; for a long thread engagement an 8H/8e fit is recommended. The corresponding "coarse" fit would be 8H/8e for a normal length of engagement and 9H/9e for a long thread engagement.

ISO metric trapezoidal threads are designated by the letters Tr; the complete designation of a single start trapezoidal thread is shown below:

For an external thread $Tr 40 \times 7 - 7c$

For an internal thread $Tr 40 \times 7 - 7H$



Multiple start threads are specified as shown by the following example:

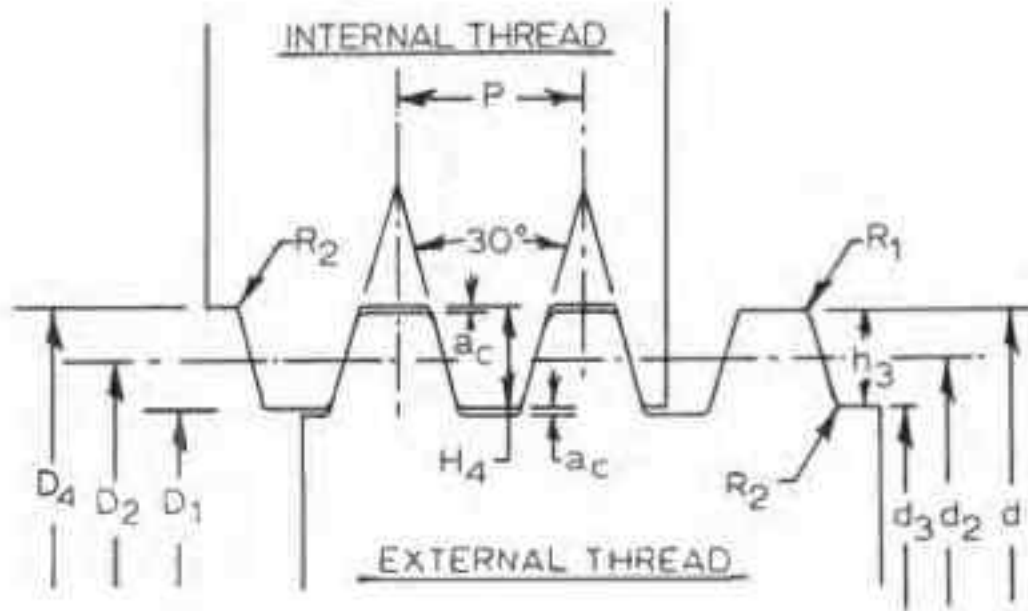
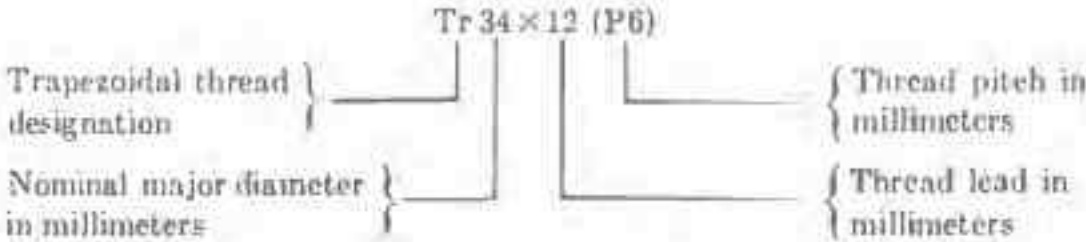


Fig. 12-1. Design forms of internal and external trapezoidal threads.



The tolerance class may be added to the end of the multiple start thread designation in the previously shown manner. Left-hand threads are designated by the addition of LH in parenthesis; e.g., $Tr\ 36 \times 6 - 7e\ (LH)$. A fit between threaded parts may be shown by the internal thread tolerance designation followed by the external thread tolerance designation, separated by a stroke; e.g., $TR\ 40 \times 7 - 7H/7e$.

Measuring and Gaging Screw Threads

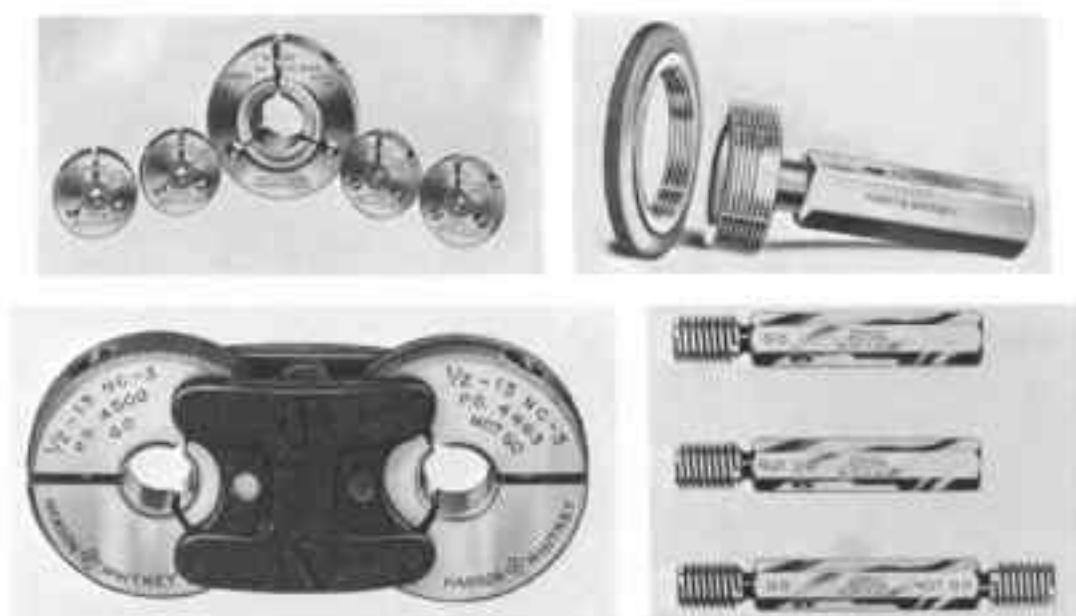
Measuring means to determine the exact dimension, while gaging is done to determine if the part is within specified dimensional limits. It is usually more difficult and time consuming to measure an object, including screw threads, than it is to gage the object. Since gaging achieves the desired result of determining the dimensional acceptability of a part, it is usually the preferred method when the cost of the gage per part to be examined is not excessive. Generally, the cost of making gages for low-volume production work is prohibitive; however, standard screw-thread gages are relatively inexpensive and are often kept available in the tool rooms of machine shops and in tool and die shops. Where the volume required is large enough, and where gaging is a requirement of the customer, there can be no question about the availability of thread gages. Measur-

ing methods of determining the acceptability of screw threads are required in making thread gages, when making other threads to exceptionally high standards of accuracy, and when gages are not available. Some of the thread-measuring instruments used now are very elaborate. This chapter will, however, only concern itself with measuring and gaging methods that are commonly used in the machine shop.

Thread Elements to Be Inspected—All of the elements of a screw thread that are listed below are not always inspected. In many cases complete reliance is placed upon an inspection based on GO-NOT GO gages, or even on the "feel" of the fit between the thread and a gage. At other times, threads are measured and gaged with great exactness. The thread elements that are subject to inspection are pitch, pitch diameter, lead, thread form, angle of thread, major diameter, minor diameter, taper, and surface finish.

Thread Plug and Thread Ring Gages—Gaging affords a fast, reliable method of determining the acceptability of external and internal threads. The skill required to use thread gages can, in most cases, be acquired without too much difficulty. One disadvantage of gaging is that part of the thread tolerance must be given to the gage in allowing for a gage-makers tolerance and gage wear. The practical effect of this is that the thread must be made more accurate than called for by the drawing in order to be accepted by the gages. There are often three classes of thread gages used in a shop: *working gages*, *inspection gages*, and *master gages*. The working gage is to be used on the machine in making the thread. The inspection gage is used by the inspector to accept or reject the finished product. Master gages represent the basic size of the thread. They are made with no allowance for wear and with the minimum gagemakers tolerance. These tolerances and the wear allowances of inspection and working gages are usually so arranged that the inspection gage will accept any thread that the working gage will accept, and will reject any thread that has been rejected by the working gage. In some shops an additional class of gages called *setting gages* is used to set thread ring and snap gages.

Thread ring and snap gages are illustrated in Fig. 12-10. They are usually based on the GO-NOT GO principle. For example, an external thread is accepted if the GO ring gages will screw over the entire length of the thread and if the NOT GO gage cannot be turned onto the thread more than one and one-half turns. The thread is rejected if either the GO gage fails to go on the thread for its entire length, or if the NOT GO gage can be turned for more than one and one-half turns. The GO ring gage should fit onto the thread without binding, or "catching," anywhere along the length of the thread. In some shops some tightness is permitted on the first two threads. *Thread ring gages* for inspecting tapered threads should go on until the back face of the gage is flush with the end of the thread. In inspecting parallel screw-threads an attempt should be made to "shake" the gage endwise, and to "rock" it, in order to detect any looseness in the fit of the thread gage on the work. Differences in the effort required to turn the gage will indicate the existence of taper or a



*Courtesy of Colt Industries, Pratt & Whitney Machine Tool Div.
 Courtesy of Hanson-Whitney Co.
 Courtesy of Greenfield Tap & Die Corp.*

Fig. 12-10. Typical thread ring and thread plug-gages.

lead error in the thread. In any case the gage should never be forced over the thread. If it does not go with a light effort it should not be forced to go further. If the gage loosens markedly, after it has been turned along the length of the thread, a more comprehensive check of the back section of the thread is warranted. In this case it is possible for the back section of the thread to be undersize without being detected by the NOT GO gage because it will not pass over the front of a thread which is within the dimensional limits.

A thread ring gage cannot be used to analyze the individual errors that could be present. For example, if the GO gage will not go onto a thread, the difficulty may be due to the pitch diameter being oversize, the minor diameter may be oversize, or there may be dirt in the threads. The GO gage may pass over threads having the major diameter undersize or errors in the thread form. The threads on the NOT GO ring gage have their roots cleared to a greater extent and their crests truncated more than the GO ring gages. This is done in an attempt to assure that any contact between the threads on the gage and the work will be on the sides of the threads and that other conditions will not affect the fit. The NOT GO gage thus is affected only by the pitch diameter of the thread.

Regardless of its limitations, the thread ring gage is a very reliable method of inspecting screw threads. It has one great advantage in that it actually passes over and inspects the entire thread area. The *thread plug gage* also possesses this advantage. GO and NOT GO thread plug gages are used to inspect internal threads much in the same manner as thread ring gages are used to inspect external threads. For an internal thread to be accepted the GO thread plug gage must pass through the entire length

of the internal thread and the NOT GO plug gage must not enter the threads for more than one and one-half turns.

The Mating Part as a Gage—A very effective method of gaging a screw thread on certain classes of work is to use a mating part as a gage. In situations where thread gages are not available, and where a thread is being made to fit a particular machine part or a tool, using the work as a gage will bring about the desired fit. In this case extra care should be taken to see that both threads are clean before they are assembled for gaging purposes. Care should also be exercised in examining the "feel" of the thread fit since the lower limit of the thread cannot be verified with a NOT GO gage. The two threads should fit together with no endwise shake and without binding. When no gages are available, and it is inconvenient to use the mating part as a gage, a bolt, cap screw, or a nut, as the case may be—that is known to have accurate threads—can be used as a gage. In many cases this method produces very acceptable results. Like the thread ring or plug gage, the mating part envelops and gages the entire thread area. It is not recommended, however, when threads are to be made to tolerances supplied by the customer.

Thread Snap Gages—When the workpiece is turned between centers on a lathe it must be removed in order to try the ring gage. Furthermore a certain amount of time is required to turn the ring gage on and off the work. The thread snap gage, Fig. 12-11, is used somewhat like a micrometer caliper and there is no need to remove the work from the lathe. It has two pairs of anvils that are adjusted by plug type setting gages to GO and NOT GO dimensions. The gage should be carefully aligned with the threads on the workpiece and then passed over the workpiece with a light pressure. It should not be forced because the C frame can spring open enough to pass oversize threads.

Another version of the thread snap gage is the *roll thread snap gage*. The gaging surfaces are free-turning rolls that can move laterally in order to align themselves with the threads on the workpiece. The rolls are not actually threads but are annular rings shaped to the form of the thread. They are made to work on the GO-NOT GO principle. Indicating type thread comparators, Fig. 12-12, are similar in some respects to thread snap gages. The thread is placed between the rolls and against a backstop. The dial test indicator will then read the variation in the pitch diameter, directly. The comparator is set to master plug gages. Care must be taken in order to make certain that the backstop is set so that the thread-gaging rolls contact on the true diameter of the workpiece. On some indicating types of thread comparators, three sets of rolls, instead of two, are used. Two lower sets of rolls are used to nest the thread of the workpiece in the manner of a V-block. An advantage of the indicating type of thread gage is its ability to check out-of-round, and taper, in the thread. A roll-type thread gage for inspecting tapered threads is shown in Fig. 12-13.

Optical Comparators—The optical comparator, Fig. 12-14, can be used to make precise measurements of screw threads. It is possible to measure the thread form, thread angle, major diameter, minor diameter, pitch



Fig. 12-11. Thread snap-gage.

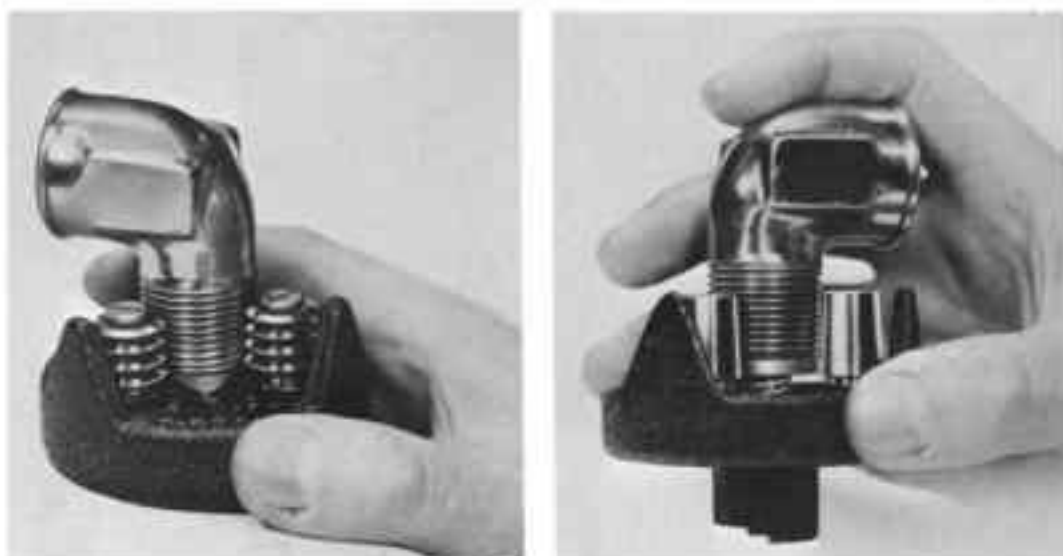


*Courtesy of Federal Products Corp.
Courtesy of Colt Industries, Pratt & Whitney Machine Tool Div.*

Fig. 12-12. Indicating-type thread comparator for inspecting external thread pitch-diameters. Initial setting established with a setting plug.

diameter, pitch, and lead error in one step. The condition of the surface of the thread can be observed by using the surface illuminator.

The optical comparator casts a beam of high-intensity light against the thread. The shadow cast by the thread profile is magnified and projected onto a ground-glass screen where it can be focused to form an enlarged silhouette of the thread. The range of magnifications available is usually from $5\times$ to $250\times$. The thread to be measured is held in position by a V-block, or between centers, as in B, Fig. 12-15. The table of the projector can be moved in three directions: perpendicular to the lens, parallel to the lens, and vertically. The table movements can be very accurately



Courtesy of Colt Industries, Pratt & Whitney Machine Tool Div.

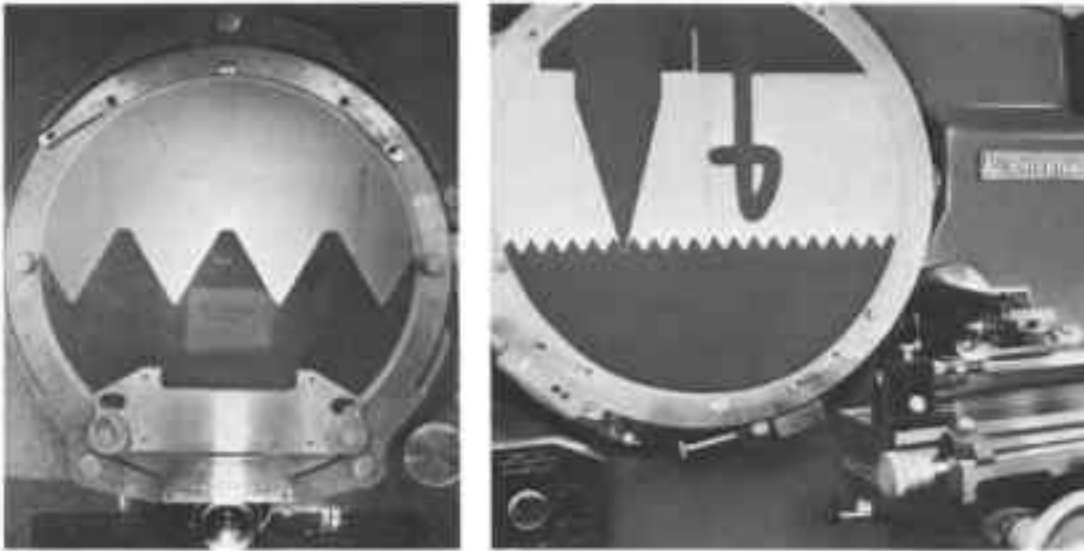
Fig. 12-13. (Left) Roll-type thread gage for checking tapered thread. (Right) Roll-type gage for checking major diameter of a tapered thread.

determined by a micrometer and end-measuring rods. The hairlines on the viewing screen can be rotated 360 degrees to measure angles. The angle can be measured to an accuracy of one minute of arc by a vernier



Courtesy of Jones & Lamson Division of Waterbury-Farrel Company

Fig. 12-14. Optical Comparator.



Courtesy of Jones & Lamson Division of Waterbury-Farrel Company

Fig. 12-15. A (Left). Projection of a 13-pitch thread at 50 magnifications. B (Right). Measuring the lead of a screw thread on an optical comparator.

scale. Standard thread comparison charts can be obtained as shown at A, Fig. 12-15. The chart may be made of plate glass, substituted for the viewing screen supplied with the projector; or, it may be made from a vinyl plastic which is placed over the viewing screen and held in place by chart clips which are attached to the projector.

Toolmakers Microscope—Toolmakers microscopes can be used to measure the thread form, thread angle, major diameter, minor diameter, pitch diameter, pitch, and lead error. The table can be moved in two perpendicular directions by precision lead-screws. The lead screws are actuated by micrometer thimbles that have graduations reading to an accuracy of .0001 inch. The eyepiece can be equipped with standard reticles that have hairlines in the form of the thread.

Three-Wire Method of Thread Measurement—The three-wire method of measuring the pitch diameter of a thread utilizes three precision wires, or pins of exactly the same diameter, which are positioned in the threads as shown in Fig. 12-16. The size of the pitch diameter is determined by measuring over the wires with a micrometer caliper and by applying this measurement in an appropriate formula. The three-wire method is the accepted method of measuring the pitch diameter on precision threads, such as thread gages, instrument threads, etc. It is not customarily employed in ordinary commercial work because thread gages are faster and easier to use. The three-wire method can be used to compare the size of two threads without the use of formulas. For example, suppose that a certain external thread must be reproduced and that there is no gage or mating thread available with which to inspect the new thread; one method would be to make a gage to fit the original thread; however, this would be time consuming and expensive. Another method would be to measure the original part with three wires in several places to obtain an accurate

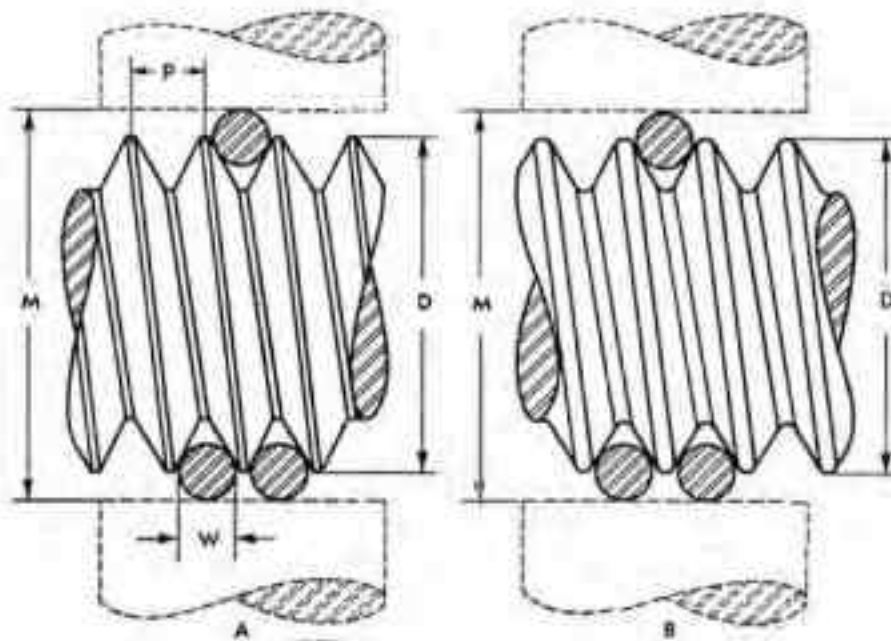


Fig. 12-16. Measuring pitch diameter by the three-wire method.

representative measurement, M . The new thread is then made to the same size as M when measured over the three wires that were used to measure the original thread. Although this cannot be classified as the three-wire method, ordinary calipers can be modified to compare the sizes of screw threads in a similar manner. The outside calipers are modified by filing the points of the legs round, so that they contact only the sides of the thread and not the roots.

The following notation applies to the formulas for measuring threads by the three-wire method:

- D = Basic major outside diameter
- N = Number of threads per inch = $1 \div P$
- E = Pitch diameter
- M = Dimension over wires
- P = Pitch = $1 \div N$
- W = Wire or pin diameter
- A = One-half of the included thread angle in the axial plane
- A_n = One-half of the included thread angle in the normal plane, or the plane perpendicular to the sides of the thread
- B = Lead angle at the pitch diameter
- L = Lead of thread
- T = Width of thread in the axial plane at diameter E .

Formulas 12-1, 12-2, and 12-3, given here, are simplified formulas that do not include the effect of the lead angle. They are, however, sufficiently accurate for many applications involving single-thread screw threads. These equations are used for threads having the following thread forms:

American Standard Unified, American National, British I.S.O. Metric, International Metric.

To find M , when the standard major diameter is known:

$$M = D - (1.5155 \times P) + (3 \times W) \quad (12-1)$$

To find M , when the pitch diameter E , is known:

$$M = E - (.86603 \times P) + (3 \times W) \quad (12-2)$$

To find the best wire size:

$$W = .57735 \times P; \text{ or } W = .57735 \div N \quad (12-3)$$

The effect of the lead angle of the thread is included in Formula 12-4. It can be used to calculate the pitch diameter of any thread form having symmetrical sides (American Standard Unified and National, B.A., B.S.W., British I.S.O. Metric, International Metric). It is recommended for calculating the pitch diameter for Acme and for worm threads that do not have an excessively large lead angle which is the case for most single-thread screws. Formula 12-4 is a simplification of one that is much more cumbersome and difficult to apply; however, the wire diameter used in Formula 12-4 must be the exact wire diameter obtained in Formula 12-4a. This, in effect, means that special wires of the size calculated from Formula 12-4a must be used.

$$M = E + W (1 + \sin A_n) \quad (12-4)$$

$$W = \frac{T \times \cos B}{\cos A_n} \quad (12-4a)$$

$$\tan A_n = \tan A \times \cos B \quad (12-4b)$$

$$\tan B = L \div 3.1416 \times E \quad (12-4c)$$

$$T = .5 \times P \quad (12-4d)$$

Formula 12-4 combines simplicity with a degree of accuracy that meets all but the most exacting requirements, particularly for lead angles below 8 or 10 degrees and for the higher thread angles. Additional equations for B.S.W. and B.S.A. threads are given below. These equations do not compensate for the lead angle; however, they do not require the use of special wires. Formula 12-7 gives the recommended wire size, although deviations from this size can be made.

Formulas for the British Standard Whitworth Thread Form (B.S.W.) are:

$$M = D - (1.6008 \times P) + (3.1657 \times W) \quad (12-5)$$

$$M = E - (.9605 \times P) + (3.1657 \times W) \quad (12-6)$$

$$W = .56368P; \text{ or } W = .56368 \div N \quad (12-7)$$

Equations for the British Association Thread Form (B.A.) are:

$$M = D - (1.7363 \times P) + (3.4829 \times W) \quad (12-8)$$

$$M = E - (1.1363 \times P) + (3.4829 \times W) \quad (12-9)$$

$$W = .5 \times P \times \sec 23^\circ 45' = .5462P \quad (12-10)$$

The application of the formulas for the three-wire method can best be shown by the following examples.

Example 12-1:

A $1\frac{1}{2}$ -12 UNF-3A thread is to be measured by the three-wire method. Calculate the dimension, M , over the three wires.

$$\begin{aligned} W &= .57735 \div N \\ &= .57735 \div 12 \\ &= .04811 \text{ inch} \end{aligned} \quad (12-3)$$

The nearest available wire size is .050 inch.

$$\begin{aligned} M &= D - (1.5155 \times P) + (3 \times W) \\ &= 1\frac{1}{2} - (1.5155 \times \frac{1}{12}) + (3 \times .050) \\ &= 1.5237 \text{ inches.} \end{aligned} \quad (12-1)$$

The thread should be cut until the measurement over the three wires is 1.5237 inches.

Example 12-2:

According to the thread standards (*Machinery's Handbook*, 21st Ed., p. 1292) for Unified Threads, the pitch diameter for the $1\frac{1}{2}$ -12 UNF-3A thread must be 1.4459 inches maximum and 1.4411 inches minimum. Calculate the maximum and minimum measurement M , over the three wires.

$$\begin{aligned} M &= E - (.86603 \times P) + (3 \times W) \\ &= E - (.86603 \times \frac{1}{12}) + (3 \times .050) \\ &= E - .07217 + .150 = E + .0778 \end{aligned} \quad (12-2)$$

Thus:

$$M_{\max} = 1.4459 + .0778 = 1.5237 \text{ inches}$$

$$M_{\min} = 1.4411 + .0778 = 1.5189 \text{ inches}$$

Example 12-3:

The pitch diameter, E , of a $\frac{1}{2}$ -10 Acme-2G thread is to be .436 inch and the lead, L , is .100 inch. Calculate the diameter of the wire, W , that must be used to measure this thread by the three-wire method and calculate the measurement, M , over the three wires.

$$\begin{aligned} \tan B &= L \div 3.1416 \times E = .100 \div 3.1416 \times .436 = .07301 \\ B &= 4^\circ 11' \end{aligned} \quad (12-4c)$$

$$\begin{aligned} \tan A_n &= \tan A \times \cos B = \tan 14^\circ 30' \times \cos 4^\circ 11' \\ &= .25862 \times .99733 = .25793 \end{aligned} \quad (12-4b)$$

$$A_n = 14^\circ 28'$$

$$T = .5 \times P = .5 \times .100 = .050 \text{ inch} \quad (12-4d)$$

$$\begin{aligned} W &= \frac{T \times \cos B}{\cos A_n} = \frac{.050 \times \cos 4^\circ 11'}{\cos 14^\circ 28'} \\ &= \frac{.050 \times .99733}{.96829} \end{aligned} \quad (12-4a)$$

$$W = .0515 \text{ inch}$$

$$\begin{aligned} M &= E + W (1 + \sin A_n) = .436 + .0515 (1 + \sin 14^\circ 28') \quad (12-4) \\ &= .436 + .0515 (1 + .24982) \end{aligned}$$

$$M = .5004 \text{ inch}$$

The accuracy of the three-wire method of thread measuring is affected by the accuracy of the measuring wires, or pins. If the accuracy of the pitch diameter of the screw thread gage to be measured is within .0001 inch, it is necessary to know the diameter of the wires within .00002 inch. Each wire should be round, within .00002 inch, and should be straight within the same amount over any quarter-inch section. A set of three wires should have the same diameter within .00002 inch; moreover, this common diameter should be within the "best size" for any given pitch. The "best size" is one that makes contact at the pitch line at one-half of the thread depth. This is where any given error in the thread angle will have the least effect upon the measurement over the wires. If the proper precautions are taken, regarding wire accuracy and contact pressure of the micrometer, it should be possible to check plug gages within an accuracy of .0001 inch. If the wire diameters are accurate to only .0001 inch, then the pitch-diameter measurement is not likely to be more accurate than .0003 inch. This, however, may be accurate enough for many classes of work.

Other sources of inaccuracy, when measuring threads by the three-wire method, are errors in thread form and in lead. The three-wire method will not detect errors in thread form; and the errors in the thread form will cause the calculated values of the measurement over the wires, M , or of the pitch diameter, E , to be incorrect. Errors in lead do not cause errors in the measurements or in the calculations. The error in lead cannot be detected by the three-wire method of measuring screw threads, thus other methods must be used to check the lead.

Screw Thread Micrometer—A screw-thread micrometer may be used for measuring the pitch diameter. This micrometer has a fixed anvil which is V-shaped, to fit over the thread, while the spindle is cone shaped at the end in order that it can enter the space between two threads. (See Fig. 12-17.) The anvil and the spindle make contact with the sides of the thread, enabling the pitch diameter to be determined. The cone-shaped point of the measuring screw is slightly rounded so that it will not bear at the bottom of the thread. There is also sufficient clearance at the bottom of the groove in the V-shaped anvil to prevent it from touching at the top of the thread. When the micrometer is closed and the 60

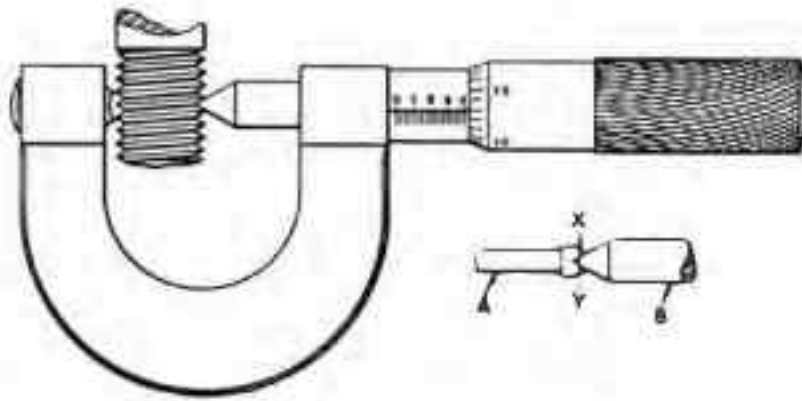


Fig. 12-17. Screw-thread micrometer.

degree point *B*, and anvil *A*, are together (as shown by the diagram), the zero on the micrometer barrel represents a line drawn through the plane *X-Y*, and, if the micrometer is opened, say, .5 inch, this reading represents the distance that two such planes are apart. Hence, the micrometer reading represents the pitch diameter. The thread micrometer does not give an absolutely accurate pitch-diameter measurement as there is a slight error which depends upon the lead angle of the thread. Usually, this error is not large enough to be of practical importance. There is no error, however, when the thread micrometer is set to a standard gage of the plug type and is then used to compare the measurement over the plug with the measurement of the thread on the workpiece. Thread micrometers are made to cover a very limited range of pitches, or threads per inch, which is governed by the pitch of the V-shape grooves in the anvil.

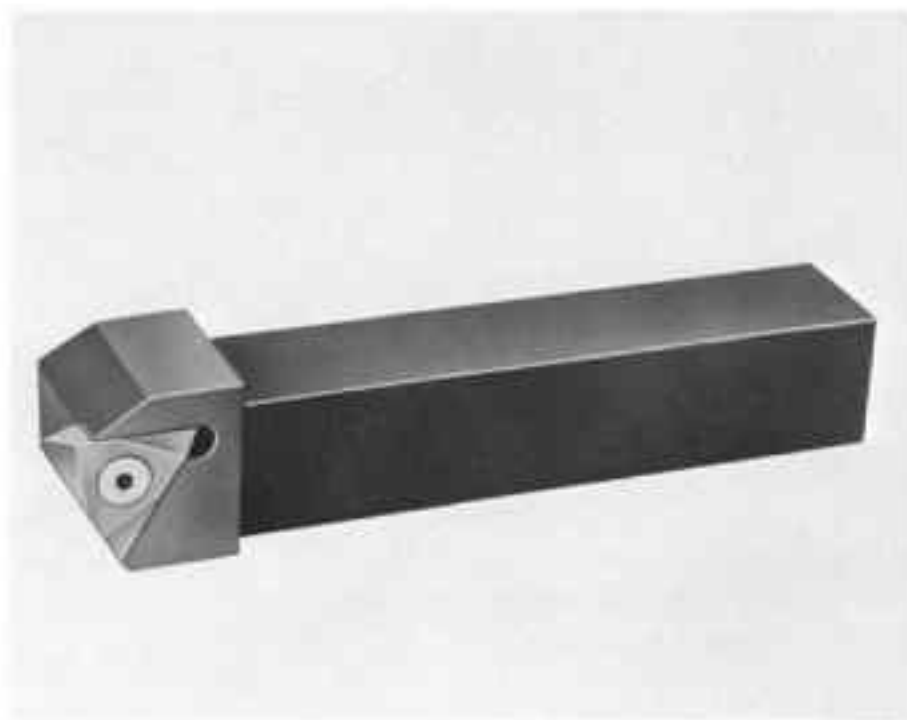
Cutting Screw Threads on a Lathe

Engine lathes are designed with features that are provided primarily for cutting screw threads. The gear ratios in the quick-change gearbox are selected to enable the lathe to cut a sequence of precise screw-thread leads. Most engine lathes are made so that these gear ratios can be modified by change gears to enable metric threads, or threads having special leads, to be cut. The lead screw, the split nut (or half nut), and the thread chasing dial are built onto the lathe for the exclusive purpose of cutting screw threads. The compound rest has many applications among which is to feed the cutting tool into the work at the proper angle when cutting screw threads. The degree of precision that is built into the lathe enables it to cut very accurate threads.

Thread cutting on a lathe is often called thread chasing. Practically every kind of thread can be cut on an engine lathe. A single-point cutting tool is used to cut screw threads on engine lathes. This tool has the advantages of low initial cost, is easily ground and resharpened, and is simple to set up on the lathe. The low tooling and set-up costs provide an advantage for producing screw threads on an engine lathe when the number of parts to be made is small. Even with larger production volumes, the lathe sometimes has an inherent advantage. For example, threads are cut on parts that are turned on a lathe in order to avoid an additional setup on another machine. Other methods of cutting screw threads are: Chasing threads with a threading die, tapping, thread milling, thread rolling, and thread grinding. Although each of these methods has its special advantages on certain classes of work, there is still much thread cutting done on the lathe.

The Single-Point Thread-Cutting Tool

The start of a good thread is the thread-cutting tool. It must have the correct shape, or profile, in order to cut an accurate thread. Usually this shape corresponds exactly with the shape of the thread to be cut. Accurate thread profiles can be ground on tool and cutter-grinding machines. While it is possible to grind a thread-cutting tool by hand on a pedestal grinder, the tools for extremely accurate threads should always be ground on the cutter and tool-grinding machines. Cemented-carbide thread-cutting tools are usually purchased already formed to the required shape, as shown in Fig.13-1. High-speed steel thread-cutting tools are usually



Courtesy of the Valenite Div. of the Valgren Corp.

Fig. 13-1. Cemented carbide thread-cutting tool.

ground to the required shape. Whether the thread-cutting tool is ground to the required shape on a cutter and tool-grinding machine, or by hand, on a pedestal grinder, it should be carefully inspected before it is used.

The design of a high-speed-steel cutting tool for cutting American Standard Unified and National Thread Forms is shown in Fig. 13-2. This tool has a 60-degree included angle and the width of the flat, F , is made appropriate to the pitch of the thread to be cut. View A, is a top view of the tool showing the shape, or profile. An alternate shape is also shown. The advantage of the alternate shape is that it reduces the surface area on the flanks of the tool which must be ground to obtain this shape, thereby reducing the area in contact with the grinding wheel. This, in turn, reduces the heat generated by the grinding wheel. It also makes it easier to control the tool while grinding, in order to obtain the desired profile. View B, is a side view of this tool, showing the angle made by the flat. This angle should be 10-to-17 degrees depending upon the hardness of the metal being cut. The right side of the tool is shown at C. This view is a true auxiliary projection which makes the tool appear to be "upside down." It is an end view of the tool sighting, parallel to the left side cutting edge and it shows the relief angle below this edge. The magnitude of the left side-cutting edge relief angle should be 10 to 17 degrees when cutting right-hand screw threads. The lower value should be used when cutting hard materials, and for cutting soft materials the size of this angle is increased to 17 degrees. View D is a true auxiliary view obtained when sighting parallel to the right side-cutting edge of the threading tool. It

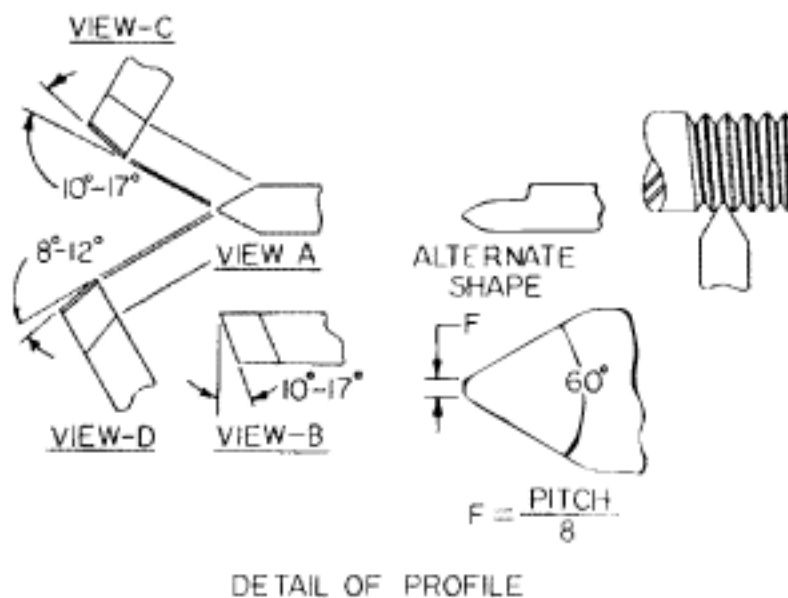


Fig. 13-2. Thread-cutting tool for American Standard Unified and National Threads.

shows the right side-cutting edge relief angle which should be 8 to 12 degrees when cutting right-hand screw threads. The face of the thread-cutting tool is ground flat, or without a rake angle. It should, however, be ground smooth in order for all the cutting edges to be sharp. When cutting large threads (say, 5 threads per inch, and larger), the threads are sometimes rough cut with a cutting tool having a side-rake of 8 to 12 degrees, and having an included angle of 50 degrees, instead of 60 degrees. The large thread is, however, finished to size with a tool having the correct included angle and no rake angle.

An offset thread-cutting tool is shown in Fig. 13-3. This threading tool is recommended when it is necessary to cut close to a shoulder or close to a lathe dog. The offset thread-cutting tool is set at an angle, as shown. This allows it to cut ahead of the cross slide and compound rest to prevent these parts from bumping into large shoulders or the lathe dog. View A is the top view of the offset thread-cutting tool, showing its profile. The alternate shape shows a 5- to 15-degree "hook," or relief, ground into the right side-cutting edge. The purpose of this relief is to reduce the area of the flank under the right side-cutting edge in order to facilitate the grinding of this surface. View B, is of the left side of this tool, showing the relief angle under the flat at the nose of the tool. It should be mentioned, however, that View B does not show this angle in its true projection. An end view showing the left side-cutting edge relief angle in its true projection is shown at E. The auxiliary View, D, is a view from the end of the tool, looking along the edge which is 25 degrees, with respect to the shank. A clearance angle of 3 to 5 degrees is customarily ground below this edge. Note that both Views D and E appear to be "upside down," which is in accord with drawing practice.

The relief angles below the cutting edge that are recommended for

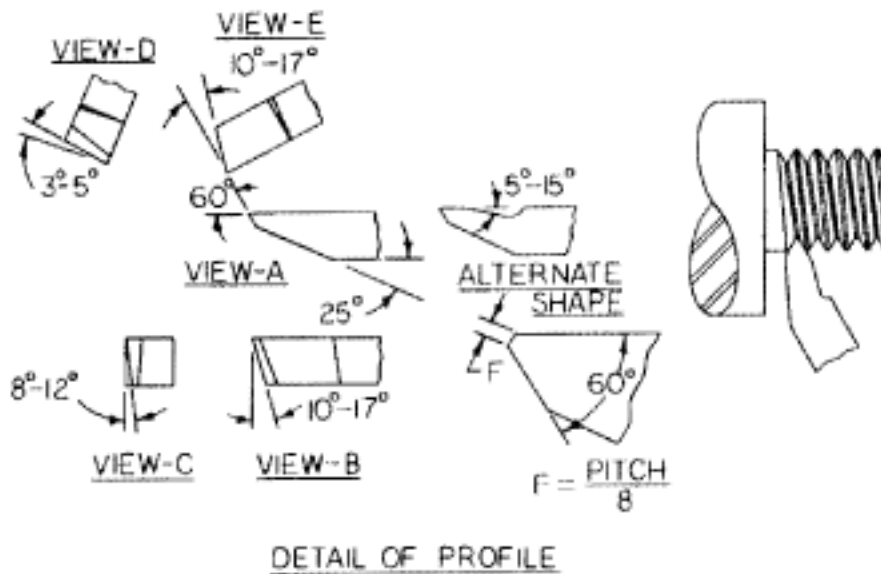


Fig. 13-3. Offset thread-cutting tool for American Standard Unified and National Threads.

cemented-carbide tools should be less than those recommended for high-speed steel. The reason for this is that cemented carbides, while being harder than high-speed steel, are also more brittle, and the cutting edge requires the additional support provided by having smaller relief angles. Thread-cutting tools must be ground to have the form of the thread to be cut. The relief angles below the cutting edges, however, should be essentially the same as recommended in Figs. 13-2 and 13-3. The profiles of other thread-cutting tools and thread forms are shown in Figs. 13-4 through 13-7.

The form of the thread-cutting tool must be inspected for accuracy. Typical gages for checking the thread forms are shown in Fig. 13-8. A center gage, such as is used for checking lathe centers, is shown at A. This is used for checking the 60-degree included angle of an American Standard Unified and National Thread Form. Form gages such as those in Fig. 13-8 should be held on the same plane as the face of the thread-cutting tool, shown at A. If the tool is made to fit the gage when the gage is held as shown by the dotted line, the included angle will be incorrect. The center gage has no provisions for checking the width of the flat at the nose of the tool. This can be done with the circular thread-gage shown at B. Opposite each notch around the circumference of the gage is stamped the number of threads per inch for which the notches have the correct form. The tool is first ground to the 60-degree angle. This angle is checked by the notch at *d* which has a sharp V to permit the tool to be ground to a sharp point. The correct width is then ground as gaged by the notch corresponding to the threads per inch to be cut by the tool.

An Acme gage is shown at C, Fig. 13-8. This also has notches for different pitches or threads per inch. The thread-cutting tool is first ground to the correct angle using the 29-degree notch at the end of the gage, as

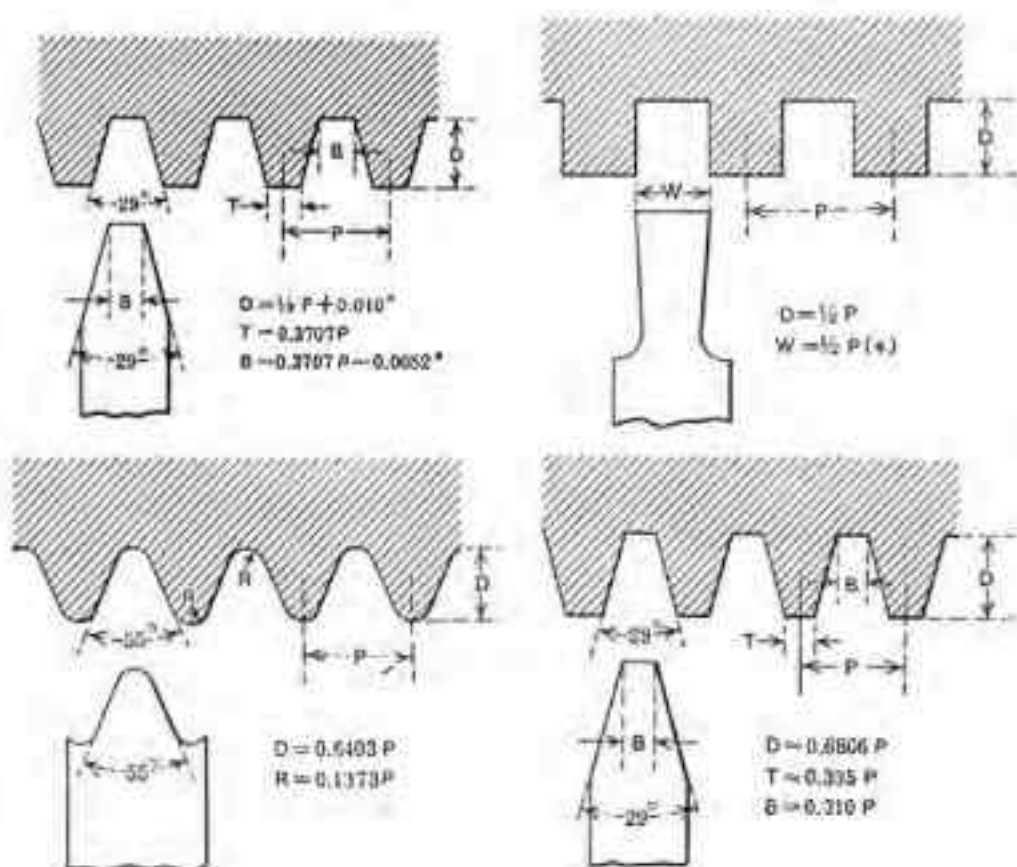


Fig. 13-4. (Upper left) American Standard Acme thread-cutting-tool profile. Fig. 13-5. (Upper right) Square thread-cutting-tool profile. Fig. 13-6. (Lower left) British Standard Whitworth thread-cutting-tool profile. Fig. 13-7. (Lower right) 29-degree worm thread-cutting-tool profile.

shown at *g*. The width at the end of the cutting tool is then ground—using the appropriate shallow notch to gage this dimension—as at *h*. The numbers opposite each notch represent the number of threads per inch. The angle between the side of the tool and end-width may be tested as illustrated at *j*. This gage may also be set square with the work by placing one edge of the gage against the turned surface and adjusting the tool until it coincides with the gage, as indicated by the dotted lines at *j*.

The end width of the Acme tool may also be measured with a gear-tooth vernier, as shown in Fig. 13-9. The distance, *A*, is usually made to equal .250 inch by setting the vertical vernier scale to this dimension. The width of the flat is then equal to the caliper-jaw, vernier scale reading minus $2 \times .250 \times \text{Tangent } 14^\circ 30'$, or $2 \times .250 \times .2586 = .1293$ inch. This can be simplified by using the following rule: The width of the end of the Acme thread tool is equal to the reading of the caliper-jaw, vernier scale minus .1293 inch, when the vertical scale is set to .250 inch and the caliper jaws touch on the side of the tool. It is, of course, assumed that the included angle of the tool is 29 degrees. For example, assume that the distance *A*, Fig. 13-9, is .250 and when the caliper jaws are touching the sides of the tool, the vernier reading is .315 inch. The width of the end of the Acme thread-cutting tool is then $.315 - .1293 = .1857$ inch.

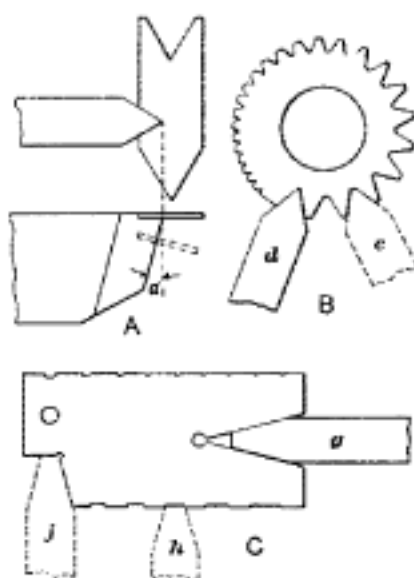


Fig. 13-8. Gages for inspecting thread-cutting tools.

The optical projector, Fig. 12-14, can also be used to measure thread-cutting tools. This is a very accurate method and its use is recommended. Likewise, a toolmakers microscope can be used for this purpose. The instruments provide the best method of checking thread-cutting tools when they are available.

Machine Setting—Change Gear Ratios

Almost all modern engine lathes are equipped with quick-change gear-box (Fig. 6-9). The lathe is set to cut the required number of threads per inch by simply placing a few levers or handles in their correct position according to the instructions given on the chart attached to the feed-thread gear box.

Many older lathes were not equipped with a quick-change gear box; instead, they had change gears such as shown in Fig. 13-10. The calculation of the change gears required to cut a given thread introduces the principles of gear ratios and shows, in principle, how the feed-thread gear box works. For this reason a study of the calculation of change gears is worthwhile.

First, one must understand why a lathe cuts a certain number of threads per inch and how this number is changed by the use of different gears. As the carriage and the cutting tool are moved by the lead screw, which is connected by gearing with the lathe spindle, the number of threads per inch that are cut depends, in every case, upon the number of turns the work makes while the lead screw is moving the carriage one inch. If the lead screw has six threads per inch, it will make six revolutions while the carriage and the thread tool travel one inch along the piece to be threaded. If change gears a and c (A, Fig. 13-10) are so proportioned that the spindle makes the same number of revolutions as the lead screw in a given

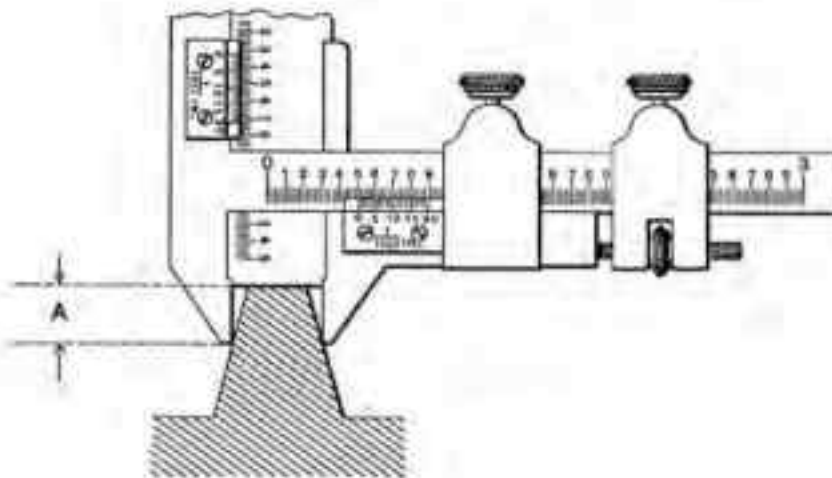


Fig. 13-9. Measuring the end width of an Acme thread-cutting tool with vernier gear-tooth caliper.

time, it is evident that the tool will cut six threads per inch because the spindle has made six revolutions while the carriage and the tool have traveled one inch. If the spindle revolved twice as fast as the lead screw it would make twelve turns while the tool moved one inch, and consequently, twelve threads per inch would be cut. To obtain this difference in speed it is necessary to use a combination of gearing that will cause the lead screw to revolve only once while the lathe spindle and work make *two* revolutions. On lathes equipped with a quick-change gear box this is done by the gear ratios inside the box. On lathes not so equipped, this is done by selecting appropriate change gears.

For calculating change gears, the lead number of the lathe must first be obtained. The lead number of the lathe, also called the lathe screw constant, is defined as the number of threads per inch cut by the lathe when gears of the same size are placed on the lead screw and the spindle stud. (See Fig. 13-10.) This can be determined experimentally by placing two, equal-size gears on the lead screw and stud, or this number is given on a chart accompanying the gears. The gears for any thread are then calculated by the following equation:

$$\frac{\text{Lead number of lathe}}{\text{Threads per inch to be cut}} = \frac{\text{Number of teeth on spindle stud gear, or driving gear}}{\text{Number of teeth on lead screw gear, or driven gear}} \quad (13-1)$$

Example 13-1:

A $\frac{3}{4}$ -10 UNC-2A thread is to be cut on a lathe having a lead number equal to 6. Calculate the change gears required:

$$\frac{\text{Lead number of lathe}}{\text{Threads per inch to be cut}} = \frac{6}{10} = \frac{6 \times 4}{10 \times 4} = \frac{24}{40} = \frac{\text{Driving gear}}{\text{Driven gear}}$$

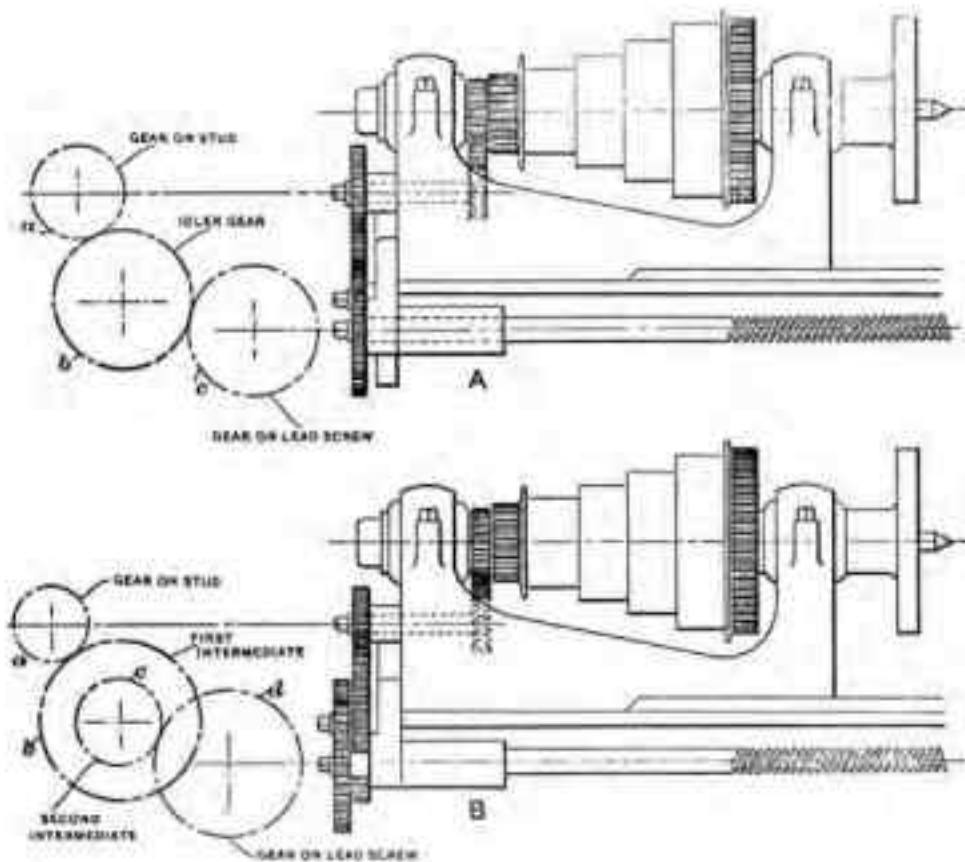


Fig. 13-10. A. Lathe with simple gearing for thread cutting. B. Lathe with compound gearing for thread cutting.

Note that the ratio $\frac{6}{10}$ has not changed because it was multiplied by $\frac{4}{4}$ which is equal to 1. Furthermore, the number of teeth on the intermediate, or idler gear, b , which connects the stud and lead-screw gears, is not considered because it does not affect the ratio between gears a and c ; however, it does affect their direction of rotation. The gearing as shown is for cutting right-hand screw threads. One additional idler gear (not a compound gear as at B, Fig. 13-10) is used between gears a and c when the direction of rotation of the lead screw must be reversed in order to cut left-hand threads. The second idler gear, like the first, affects only the direction of rotation and not the ratio of gears a and c .

Sometimes the change gears cannot be arranged, as shown at A, Fig. 13-10, because the required gears to cut the desired threads per inch are unavailable. In this case it is necessary to use four gears, or compound gearing, as shown at B. The calculations required to determine the size of the compound gears are shown by the following example:

Example 13-2:

A $\frac{5}{16}$ -24 UNF-2A thread is to be cut on a lathe that has a lead number

equal to 6. Calculate the size of the change gears required to cut this thread by using a compound-gear ratio.

$$\begin{aligned}\frac{\text{Lead number of lathe}}{\text{Threads per inch to be cut}} &= \frac{6}{24} = \frac{2 \times 3}{4 \times 6} = \frac{(2 \times 20) \times (3 \times 10)}{(4 \times 20) \times (6 \times 10)} \\ &= \frac{40 \times 30}{80 \times 60} = \frac{\text{Driving gear}}{\text{Driven gear}}\end{aligned}$$

Note again, that the original ratio $\frac{6}{24}$ is never changed. In order to calculate compound gears the ratio was changed from $\frac{6}{24}$ to $\frac{2 \times 3}{4 \times 6}$ and each term is then multiplied by a ratio equal to 1; i.e. $\left(\frac{20}{20} = 1 \text{ and } \frac{10}{10} = 1\right)$. The position of the gears should be such that the driving gears act as drivers and the driven gears follow the drivers. At B, Fig. 13-10, gear *a* is a driving gear that drives gear *b*, and gear *c* is a driving gear that drives gear *d*. The gears may be positioned in any of the following combinations in order to cut 24 threads per inch:

	Driving Gear <i>a</i>	Driven gear <i>b</i>	Driving Gear <i>c</i>	Driven Gear <i>d</i>
	Teeth per gear			
Combination 1	40	80	30	60
Combination 2	40	60	30	80
Combination 3	30	80	40	60
Combination 4	30	60	40	80

Observe that the 40- and 30-tooth gears are always driving gears and that the 80- and 60-tooth gears are always driven gears.

Metric threads can be cut on lathes having a lead screw that is dimensioned in threads per inch by using a 127-tooth translating gear in the gear train connecting the headstock spindle to the lead screw. The size of metric threads is specified by their pitch. Since there are 25.4 mm in an inch, we have:

$$\begin{aligned}25.4 \times \text{pitch in inches} &= \text{pitch in mm} \\ \frac{1}{25.4 \times \text{pitch in inches}} &= \frac{1}{\text{pitch in mm}} \\ \frac{1}{\text{pitch in inches}} &= \frac{25.4}{\text{pitch in mm}}\end{aligned}$$

Then:

$$\text{Threads per inch} = \frac{1}{\text{pitch in inches}} = \frac{25.4}{\text{pitch in mm}}$$

Applying this relationship to Equation 13-1:

$$\begin{aligned} \frac{\text{Lead number of lathe}}{\text{Threads per inch}} &= \frac{\text{Lead number of lathe}}{\frac{25.4}{\text{pitch in mm}}} \\ &= \frac{\text{Lead number of lathe} \times \text{pitch in mm}}{25.4} = \frac{5 \times \text{Lead number of lathe} \times \text{pitch in mm}}{5 \times 25.4} \end{aligned}$$

Thus:

$$\frac{5 \times \text{Lead number of lathe} \times \text{pitch in mm to be cut}}{127} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-2)$$

Example 13-3:

A M10 × 1.5 thread must be cut on an inch lead screw lathe having a lead number of 4. Calculate the change gears required to cut this thread when using a plain gear train and when using compound gears.

For a plain gear train:

$$\begin{aligned} \frac{5 \times \text{Lead number of lathe} \times \text{pitch in mm to be cut}}{127} &= \frac{5 \times 4 \times 1.5}{127} \\ &= \frac{30}{127} = \frac{\text{Driving gears}}{\text{Driven gears}} \end{aligned}$$

For a compound gear train:

$$\begin{aligned} \frac{5 \times \text{Lead number of lathe} \times \text{pitch in mm to be cut}}{127} &= \frac{5 \times 4 \times 1.5}{127} = \frac{5 \times 6}{127} \\ &= \frac{(6 \times 4) \times (5 \times 10)}{(4 \times 10) \times 127} \\ &= \frac{24 \times 50}{40 \times 127} = \frac{\text{Driving gears}}{\text{Driven gears}} \end{aligned}$$

The lead screw thread of a lathe may be made to have a standard number of threads per inch, or to have a standard pitch in millimeters. When the lead screw is metric, with the thread having a standard pitch in millimeters, it is necessary to know the metric lead number of the lathe in

order to calculate the change gears for cutting threads. *The metric lead number of a lathe is the pitch in millimeters that will be cut on a part when equal size gears are placed on the spindle stud and on the lead screw* (Fig. 13-10). The following equation can then be used to calculate the change gears to cut a thread having any pitch in millimeters.

$$\frac{\text{Pitch in mm to be cut}}{\text{Metric lead number of lathe}} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-3)$$

The driving and driven gears are defined as before. Either a plain gear train or compound gears may be used, depending on the available gears and the space on the gear train.

Example 13-4:

Calculate the change gears required to cut a M30×3.5 thread on a lathe having a metric lead screw and a metric lead number of 6.

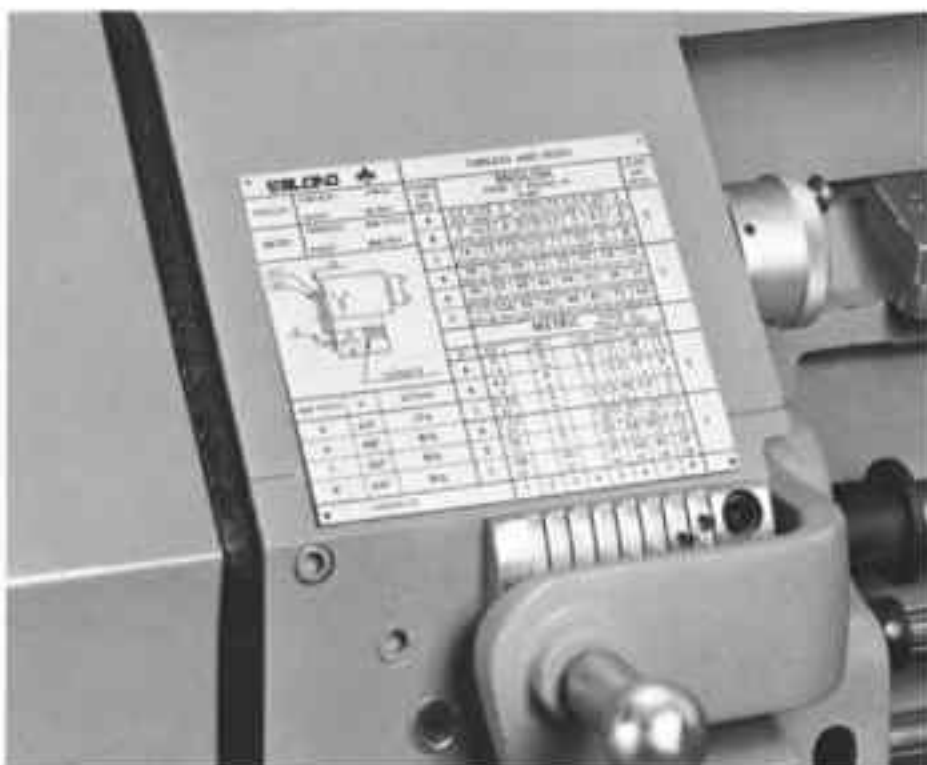
$$\begin{aligned} \frac{\text{Pitch in mm to be cut}}{\text{Metric lead number of lathe}} &= \frac{3.5}{6} = \frac{3.5 \times 8}{6 \times 8} \\ &= \frac{28}{48} = \frac{\text{Driving gears}}{\text{Driven gears}} \end{aligned}$$

Inch threads can also be cut on lathes having a metric lead screw; the required change gears can be calculated by using the following equation:

$$\begin{aligned} &\frac{127}{5 \times \text{tds/in. to be cut} \times \text{metric lead number of lathe}} \\ &= \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-4) \end{aligned}$$

Quick-Change Gearbox

The quick-change gearbox, Fig. 13-11, is a part of the gear train connecting the headstock spindle with the lead screw and the feed rod. Instead of using different change gears, it is only necessary to position the handles of the gearbox to adjust the speed ratio of the spindle and the lead screw in preparation for cutting a thread. The quick-change gearbox in Fig. 13-11 has inch-metric capability; i.e., it has settings for cutting both inch and metric threads. The standard gears, shown in Fig. 13-12, provide the required speed ratio between the spindle and the gearbox input shaft. This gear train must contain a 127-tooth translating gear in order to provide dual inch-metric capability to the gearbox. Dual inch-metric gearboxes can be found on lathes having either an inch thread or a metric thread lead screw. Many quick-change gearboxes on lathes having an



Courtesy of LeBlond Machine Tool Co.

Fig. 13-11. Quick-change gearbox for lathe equipped with a translating gear. Instructions are provided on gearbox plate for setting-up to cut either inch or metric threads.

inch thread lead screw have settings for cutting only inch threads. Likewise, many metric thread lead screw lathes have gearboxes which have only metric thread settings.

Modifying the Output of Quick-Change Gearboxes

There are occasions when a thread must be cut for which there is no quick-change gearbox setting. In this event it is necessary to modify the normal or standard gear ratio between the spindle and the gearbox by installing modifying change gears to replace the standard gears normally used. This enables metric threads and other odd pitch threads to be cut on inch thread lead screw lathes equipped with quick-change gearboxes having only inch thread settings. Likewise, inch and other odd pitch threads can be cut on metric thread lead screw lathes equipped with gearboxes having only metric thread settings. Modifying change gears can be used for cutting odd pitch threads on lathes having a quick-change gearbox that has both inch and metric thread settings.

The sizes of the modifying change gears can be calculated by the formulas given further on; they depend on the thread to be cut and the setting of the quick-change gearbox. Many different sets of modifying

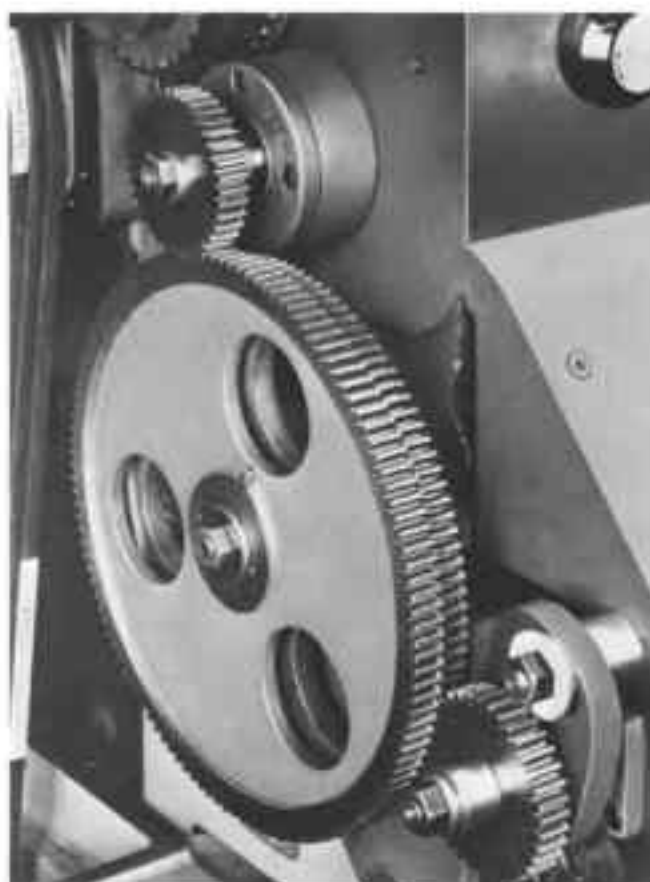


Fig. 13-12. Translating gears for cutting metric threads on a lathe having an inch dimensioned lead screw. The largest gear is the 127-tooth translating gear.

change gears can be found for each thread to be cut. It is usually worthwhile to make several calculations in order to find the set of gears that is most suitable for installation on the lathe. The following formulas are based on the type of lead screw with which the lathe is equipped; i.e., whether the lead screw has inch or metric threads.

Metric Threads/Inch Lead Screw Lathes—A 127-tooth translating gear must be used in the modifying gear train to enable metric threads to be cut on inch thread lead-screw lathes. The formula for calculating these gears is given below:

$$\frac{5 \times \text{gearbox setting thds/in.} \times \text{pitch in mm to be cut}}{127} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-5)$$

In order to use this formula it is necessary to select a quick-change gearbox setting. While any setting could be used, many would require gears that are not suitable for installation on the lathe. Often a gearbox setting that is close to the actual thread being cut will result in the calculation of

suitable gears. Since in this case the gearbox settings are in terms of threads per inch, it is advisable to calculate the actual number of metric threads in a one inch length, which is called the equivalent threads per inch.

$$\text{Equivalent thds/in.} = \frac{25.4}{\text{Pitch in mm to be cut}} \quad (13-6)$$

A gearbox setting that is close to the equivalent threads per inch may be selected. If convenient gears cannot be found by this method, a second method is to manipulate Formula 13-5 slightly before the gearbox setting is selected, as will be shown in the following example.

Example 13-4:

Select the quick-change gearbox setting and calculate the modifying change gears required to set up a lathe having an inch thread lead screw in order to cut an M12 $\times$ 1.75 thread.

Method 1:

$$\begin{aligned} \text{Equivalent thds/in.} &= \frac{25.4}{\text{Pitch in mm to be cut}} = \frac{25.4}{1.75} \\ &= 14.5 \text{ (Use 14 thds/in.)} \\ \frac{5 \times \text{gearbox setting in thds/in.} \times \text{pitch in mm to be cut}}{127} &= \frac{5 \times 14 \times 1.75}{127} \\ &= \frac{(24) \times 5 \times 14 \times 1.75}{(24) \times 127} = \frac{(5 \times 14) \times (24 \times 1.75)}{24 \times 127} = \frac{70 \times 42}{24 \times 127} = \frac{\text{Driving gears}}{\text{Driven gears}} \end{aligned}$$

Method 2:

$$\begin{aligned} &\frac{5 \times \text{gearbox setting in thds/in.} \times \text{pitch in mm to be cut}}{127} \\ &= \frac{5 \times \text{gearbox setting in thds/in.} \times 1.75}{127} \\ &= \frac{(6 \times 4) \times 5 \times \text{gearbox setting in thds/in.} \times 1.75}{(6 \times 4) \times 127} \\ &= \frac{(6 \times 5) \times (4 \times \text{gearbox setting in thds/in.} \times 1.75)}{24 \times 127} \\ &= \frac{30 \times (4 \times \text{gearbox setting} \times 1.75)}{24 \times 127} \end{aligned}$$

With the formula in this form, three of the required gears (30, 24, and 127) have been found. It is now only necessary to find the fourth gear and the gearbox setting. A gearbox setting is *selected* which when multi-

plied by 4 and 1.75 will result in a convenient gear size. In this case, the gearbox setting selected is 16 threads per inch. Thus:

$$\frac{30 \times (4 \times 16 \times 1.75)}{24 \times 127} = \frac{30 \times 112}{24 \times 127} = \frac{\text{Driving gears}}{\text{Driven gears}}$$

Odd Inch Pitch Threads/Inch Lead Screw Lathes—In order to calculate the modifying change gears to cut odd pitch threads that are specified by their pitch in inches, it is necessary to know the sizes of the standard gears, which can be found by counting their teeth. Standard gears are those used to enable the lathe to cut the threads for which the gearbox is set; they are the gears that are normally used. The threads on worms, which are used with worm gears, are among the odd pitch threads that can be cut by this method. Here again, it is advisable to calculate the actual number of threads per inch of the odd pitch thread and to select a gearbox setting that is close to the calculated value. The following formula is used to calculate the modifying change gears required to cut these threads:

$$\frac{\text{Standard driving gear} \times \text{pitch to be cut in inches} \times \text{gearbox setting in thds/in.}}{\text{Standard driven gear}} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-7)$$

Example 13-5:

Select the quick-change gearbox setting and calculate the modifying change gears required to cut a single thread worm having a pitch equal to .065 inch. The standard driving and driven gears both have 48 teeth.

$$\begin{aligned} \text{Thds/in.} &= \frac{1}{\text{pitch}} = \frac{1}{.065} = 15.4 \quad (\text{Use } 15 \text{ thds/in.}) \\ &\frac{\text{Standard driving gear} \times \text{pitch to be cut in inches} \times \text{gearbox setting in thds/in.}}{\text{Standard driven gear}} \\ &= \frac{48 \times .065 \times 15}{48} = \frac{(1000) \times .065 \times 15}{(1000)} = \frac{65 \times 15}{500 \times 2} = \frac{65 \times 15 \times (2)}{50 \times 10 \times 2 \times (2)} \\ &= \frac{65 \times 30}{50 \times 40} = \frac{\text{Driving gears}}{\text{Driven gears}} \end{aligned}$$

The procedure for calculating the modifying change gears using the formulas that follow is the same as illustrated by the two previous examples. Odd pitch threads, where the pitch is given in millimeters, can be cut on inch thread lead screw lathes by calculating the modifying change gears using Formula 13-5, or by converting the pitch into inches and using Formula 13-7. Likewise, the gears for cutting odd inch pitch

threads can be calculated by converting the pitch into millimeters and using Formula 13-10.

Odd Threads per Inch/Inch Lead Screw Lathes—

$$\frac{\text{Standard driving gear} \times \text{gearbox setting in thds/in.}}{\text{Standard driven gear} \times \text{thds/in. to be cut}} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-8)$$

*Inch Threads/Metric Lead Screw Lathes—*A 127-tooth translating gear must be used to cut inch threads that are specified in terms of threads per inch on metric lead-screw lathes. It is advisable to calculate the inch threads in terms of the pitch in millimeters and then to select a quick-change gearbox setting that is close to this value.

$$\frac{\frac{\text{Pitch in mm} = \frac{25.4}{\text{thds/in.}}}{127}}{5 \times \text{gearbox setting in mm pitch} \times \text{thds/in. to be cut}} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-9)$$

Odd Metric Pitch Threads/Metric Lead Screw Lathes—

$$\frac{\text{Standard driving gear} \times \text{mm pitch to be cut}}{\text{Standard driven gear} \times \text{gearbox setting in mm pitch}} = \frac{\text{Driving gears}}{\text{Driven gears}} \quad (13-10)$$

Machine Setting—Compound Rest

In setting up the machine for cutting screw threads, the compound rest should be positioned to feed the cutting tool into the work at an angle, so that only one cutting edge forms a chip. For American Standard Unified and National threads, or any thread having a 60-degree included angle, this angle should be 60 degrees with respect to the axis of the lathe or 30 degrees with respect to the cross slide. Some machinists prefer to set the compound rest at 29 degrees, with respect to the cross slide. For Acme threads the compound rest should be set at $14\frac{1}{2}$ degrees with respect to the cross slide, or $75\frac{1}{2}$ degrees with the axis of the lathe.

Cutting Speed for Thread Cutting

Theoretically, the same cutting speed can be used to cut screw threads on an engine lathe as is recommended for turning. In practice, threads can sometimes be cut at this speed. There are limitations, however, that are imposed by the design of the lathe, the workpiece, the lathe operator, and sometimes by the setup of the workpiece on the lathe, that must be weighed in arriving at a lathe spindle speed. Some lathes are easier to manipulate for thread cutting than others. For example, it is easier to engage the split nut over the lead screw on a 16-inch lathe with a coarse pitch thread than on a smaller lathe with a finer-pitch lead screw thread.

The cross slide can also be moved more rapidly on the 16-inch lathe because of the coarser-pitch thread on the cross-feed screw. Fine-pitch threads on the workpiece can usually be cut at a faster speed than coarse-pitch threads. The spindle speed must often be reduced when cutting deep threads such as Acme, or square, threads. Often, the speed must be reduced when the thread ends close to the shoulder. In some cases the setup of the workpiece necessitates a reduction in speed. An example of this can be seen in Fig. 13-26. The skill of the operator and his familiarity with a particular machine must also be taken into account.

A learner should start out by using a slow spindle speed for cutting threads. As skill and confidence are gained, the spindle speed of the lathe can be increased. It must be emphasized that thread cutting is not necessarily a slow-speed operation. The spindle speed should be as fast as conditions will permit, up to the limit imposed by the cutting speed recommended for the material being cut. (See Tables 7-1 through 7-5.)

Thread-Chasing Dial—Inch Lead Screw Lathes

In order to cut, or "chase," a complete thread on a lathe with a single-point tool, a number of cuts is required, each successive cut enlarging the thread groove until it has reached the depth required. The tool is withdrawn at the end of each cut and returned to the start of the thread at the tailstock end of the work. When starting the next cut the tool must enter the thread groove made by the first cut. One way to do this is to leave the split nut engaged and reverse the lathe in order to return the cutting tool. This method is objectionable because it is slow. It is better to disengage the split nut at the end of each cut and return the carriage by hand to its starting point. (Some large lathes have a power rapid traverse for moving the carriage.) When the split nut is disengaged from the lead screw, the tool must be made to enter the thread groove that is being cut when the carriage is again engaged with the lead screw. This is the function performed by the thread-chasing dial, Fig. 13-13.

The thread-chasing dial gives a visual indication as to when the split nut lever should be actuated to engage the carriage to the lead screw, in order to make the tool enter the thread groove. It is attached to the carriage and has a worm gear that engages the lead screw. The worm gear will revolve when the lead screw is turning and the carriage is stationary, or when the carriage is moved longitudinally, with the split nut disengaged and the lead screw stationary. When the carriage is moved by the lead screw, the worm gear will not revolve. A graduated dial is connected to the worm gear by a vertical shaft. The graduated dial is used by engaging the lead screw when one of the graduations is opposite the reference mark.

The thread-chasing dial operates on the following principle: The number of teeth on the worm gear of the thread-chasing dial is some multiple of the number of threads per inch of the lead screw. The number of teeth on the worm gear divided by the threads per inch of the lead screw will equal the number of graduations on the dial. For example, if the worm

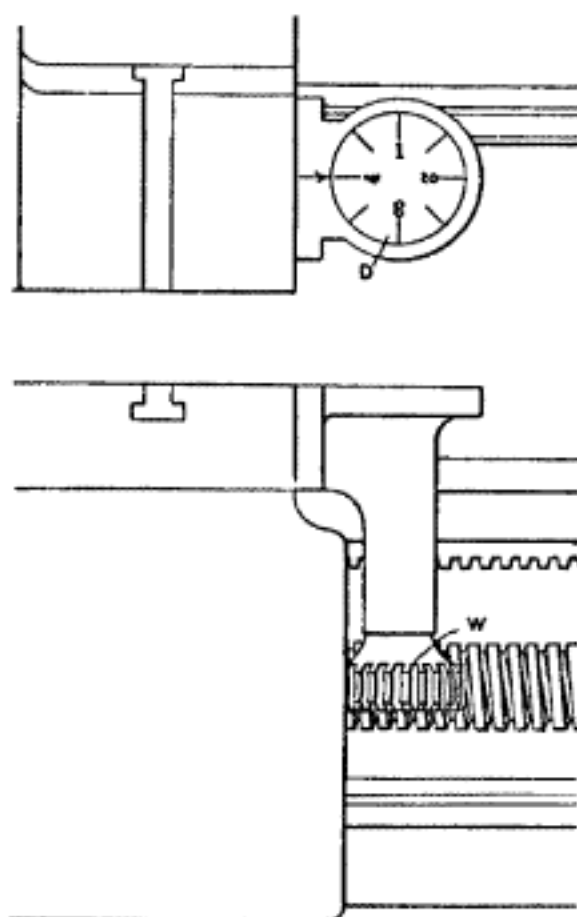


Fig. 13-13. Thread chasing dial.

gear has 24 teeth and the lead screw has 6 threads per inch, the dial would have four divisions—each representing an inch of carriage travel. Then, if the carriage should be moved one inch with the lead screw stationary, six teeth on the worm would be engaged by the six threads on the lead screw, thereby causing the worm and the connected dial to turn one-quarter of a turn ($6 \text{ teeth} \div 24 \text{ teeth} = \frac{1}{4} \text{ turn}$) which is equivalent to one mark or graduation on the dial since there are four equally spaced graduations around the dial. By subdividing the dial into eight equally spaced graduations, each line will correspond to a carriage movement of $\frac{1}{2}$ inch. The dial, therefore, will enable the carriage to be engaged with the lead screw at points equal to a travel of one-half inch.

To illustrate the significance of understanding the principle of the thread chasing dial, assume 13 threads per inch are being cut and that the work is stationary with the point of the tool in a thread. The lead screw is disengaged and the carriage is moved $\frac{1}{2}$ inch, or one mark on the dial. The point of the tool will be in the middle—between the sixth and seventh thread. It will *not* be in the thread *groove*. If the carriage were now moved two marks on the dial, or one inch, the tool could enter a thread groove again. Thus, when the number of threads to be cut is odd, the carriage can only be engaged when one of the four graduations, representing an

inch of travel, is opposite the reference mark. Suppose that ten threads per inch are being cut, with the work stationary and the point of the tool in a thread. When the lead screw is disengaged, the carriage is moved $\frac{1}{2}$ inch and the tool will be in a thread groove since there are 5 even threads in the $\frac{1}{2}$ -inch length of thread. Obviously, the tool will be in a thread groove when the carriage has moved one inch. Thus, it can be concluded that the carriage can be engaged when any mark on the dial comes opposite the index. If $11\frac{1}{2}$ threads per inch are to be cut, the dial must be engaged on graduations representing a two-inch movement of the carriage, because in this instance, there will be a whole number of threads — in this case 23. Thus, the split nut must be engaged at every fourth graduation. Since every second graduation on the dial is numbered, this would mean that the split nut can be engaged at numbers 1 and 3, or 2 and 4 when cutting threads such as $11\frac{1}{2}$, $12\frac{1}{2}$, $15\frac{1}{2}$, etc.

There are differences in design in thread chasing dials; however, in principle they are all the same. To determine how to use a particular dial, simply move the carriage, longitudinally, a distance of $\frac{1}{2}$ inch with the spindle stationary and note if the dial has turned one mark. Next, move the carriage one inch and note which marks on the dial represent the one-inch movement. Remember that *even* number threads per inch are cut by engaging the carriage at the $\frac{1}{2}$ -inch mark; *odd* number threads per inch are cut by engaging the carriage at the one-inch mark; and *fractional pitches*, such as $11\frac{1}{2}$ threads per inch, are cut by engaging the carriage at two-inch marks.

Thread Chasing Dial-Metric Lead Screw Lathes

The operation of a metric thread chasing dial on a lathe having a metric lead screw is identical to operating a thread chasing dial on a lathe having thread-per-inch lead screw. The dial gives a visual signal to the operator to engage the split nut to the lead screw when a predetermined graduation on the face of the dial is opposite the index mark. When the lead screw is engaged in this manner, the thread cutting tool will re-enter the thread groove as it moves along the workpiece. Metric thread chasing dials differ because they require a choice of one of several gears to obtain the necessary relationship between the thread on the workpiece and the thread on the lead screw, whereas only one gear is required on thread chasing dials used with inch lead screw lathes. With the metric thread chasing dial it is necessary to find the combination of the dial graduation and the specific gear that will give the desired signal. On most metric lathes this combination is given on a chart attached to the lathe. In instances where no chart is provided or the thread pitch to be cut is not on the chart, the thread chasing dial setting can be calculated. Depending on the design of the particular dial, there may be some thread pitches for which the dial cannot be used.

Although in principle all metric thread chasing dials are the same, they

may be different in the details of their construction. The principle of the metric thread chasing dial is illustrated in Fig. 13-14. The dial is attached to a shaft to which a cluster of gears is also attached. Any one of the five gears shown may be positioned to engage the lead screw. The combination of a particular gear in the cluster, lead-screw pitch, and dial graduation provides a base pitch.

The base pitch is the distance in millimeters that a carriage will travel when the thread chasing dial rotates a certain amount—as determined by the graduations on the dial—while the lead screw is not rotating. Assuming that a lathe has a lead screw with a particular metric pitch, the base pitch determines if a combination of gear size (in terms of number of teeth) and dial graduations can be used when cutting a given thread. When the quotient of the base pitch divided by the pitch of the required thread is a whole number, the thread chasing dial can be used for cutting the required thread. For many metric thread pitches several thread chasing dial settings are possible. The following formula can be used to calculate the base pitch in millimeters:

$$\text{Base Pitch} = \frac{\text{No. teeth on gear} \times \text{pitch of thread on lead screw}}{\times \text{no. of turns of dial}} \quad (13-17)$$

In the formula, the number of turns of the thread chasing dial is determined by the graduations on its face in the following manner, using as an example, the dial in Fig. 13-14:

Using any of the eight graduations is equivalent to $\frac{1}{8}$ turn of the dial.

Using only the graduations numbered 1, 2, 3, and 4 is equivalent to $\frac{1}{4}$ turn of the dial.

Using only the graduations numbered 1 and 3, or 2 and 4, is equivalent to $\frac{1}{2}$ turn of the dial.

Using only one graduation (1, 2, 3, or 4) and no other graduation is equivalent to one turn of the dial.

Calculating the combination of the dial graduations to use and the gear size is somewhat of a trial and error process, until a combination is found that will work. Moreover, it is often advisable to make several calculations until the best combination is found, as will be shown in the following example.

Example 13-8:

Using the thread chasing dial in Fig. 13-14, calculate the gear and the dial graduations that can be used when cutting an M24 $\times$ 3 thread. The pitch of the lead screw is 6 mm.

1. Try the 14 tooth gear using any graduation on the dial.

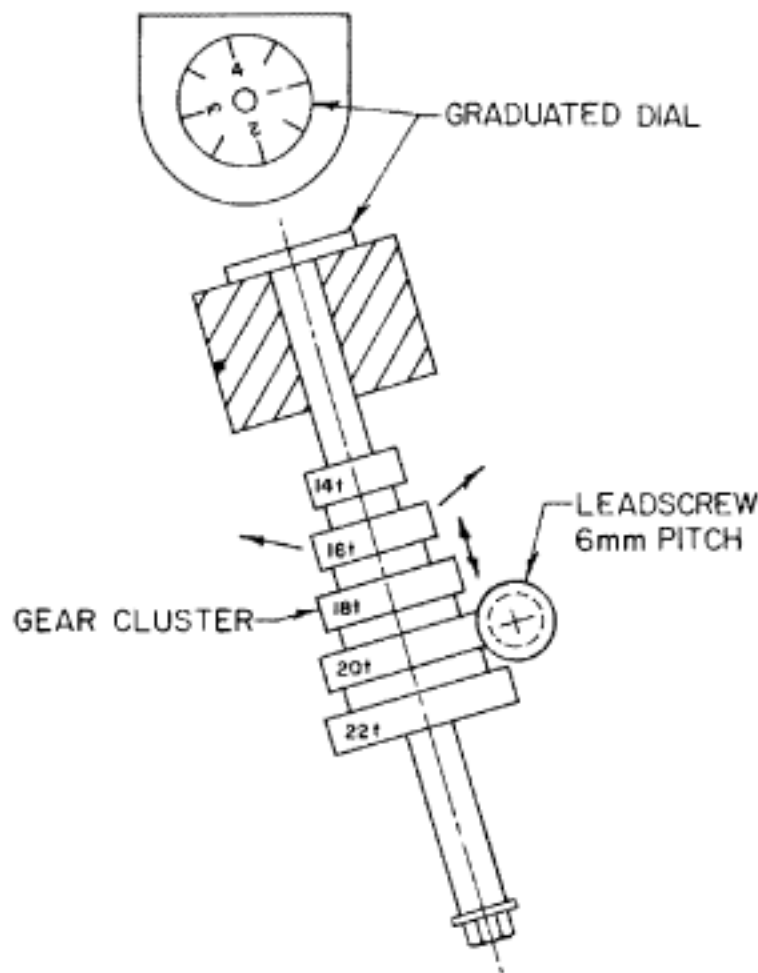


Fig. 13-14. Schematic drawing of a thread-chasing dial for a lathe equipped with a metric lead screw.

$$\text{Base Pitch} = \text{No. teeth on gear} \times \text{pitch of thread on lead screw} \\ \times \text{no. turns on dial} = 14 \times 6 \times \frac{1}{8} = 10.5 \text{ mm}$$

$$\frac{\text{Base pitch}}{\text{Pitch of thread required}} = \frac{10.5}{3} = 3.5$$

This combination cannot be used because the quotient of the base pitch divided by the pitch of the thread required is not a whole number.

2. Try the 14 tooth gear using only the graduations numbered 1, 2, 3, and 4.

$$\text{Base pitch} = 14 \times 6 \times \frac{1}{4} = 21 \text{ mm}$$

$$\frac{\text{Base pitch}}{\text{Pitch of thread required}} = \frac{21}{3} = 7$$

This combination can be used since the quotient is a whole number.

3. Try the 16 tooth gear using any graduation on the dial.

$$\text{Base pitch} = 16 \times 6 \times \frac{1}{8} = 12 \text{ mm}$$

$$\frac{\text{Base pitch}}{\text{Pitch of thread required}} = \frac{12}{3} = 4$$

This combination is preferred because the lathe operator can use any graduation on the thread chasing dial.

The significance of the base pitch becomes apparent in the above example. With the lathe stopped and a particular gear engaging the lead screw, the base pitch is the distance that the carriage will move when the thread chasing dial turns a certain amount, as indicated by the graduations. In the example, using the 16-tooth gear, when the carriage is moved 12 mm (one mark on the dial) the thread cutting tool will re-enter a thread groove. In operation, the threads on the workpiece and on the lead screw are rotating. Whenever the lead screw has made two complete revolutions ($2 \times 6 \text{ mm} = 12 \text{ mm}$), the gears on the lathe cause the thread groove on the workpiece to make four complete revolutions ($4 \times 3 \text{ mm} = 12 \text{ mm}$); therefore, the thread groove will be aligned to accept the thread cutting tool.

Setting the Thread-Cutting Tool

The thread-cutting tool must be set accurately, otherwise the form of the thread that is cut will be incorrect even if the tool is correctly shaped. It should be set at the height of the lathe centers. The angular relationship of the thread is obtained by holding a center gage against the work, or against any convenient cylindrical surface that is parallel to the axis of the lathe, and comparing the sides of the threading tool with the sides of the 60-degree notch on the side of the center gage as shown in Fig. 13-15. The tool should not be jammed tightly in the 60-degree notch. Enough space should be left free to allow the center gage to be moved sideways so that each side can be checked independently of the other. It will be helpful to place a piece of white paper that will reflect light underneath the center gage. Thread-boring tools are set in a similar fashion as is seen in Fig. 13-23. Acme thread-cutting tools are also set in a similar manner except that an Acme thread gage is used in place of the center gage. For ultra-precision work, especially on smaller threads, the tool setting is sometimes given a final inspection by cutting a thread in a piece of brass with the tool. The thread is then inspected on an optical comparator or under a toolmakers microscope. Any errors in the position of the cutting tool can be seen on the thread and corrected. The procedure is repeated until the thread form is correct. For most ordinary classes of work using the center gage, or the thread gage, as described, is sufficient when done with care.

Cutting the Thread

Two typical thread cutting operations on an engine lathe are shown in Figs. 13-16 and 13-17. The procedure for cutting the thread is shown in Figs. 13-18 and 13-19. As shown schematically in Fig. 13-18, the thread is cut by taking a number of cuts across the workpiece with a single point thread cutting tool that has been ground to the shape of the thread. Each successive cut enlarges the thread groove until it reaches the finished size. Before starting each cut, the tool is fed inward a small amount with the compound rest which establishes the equivalent of the feed rate, or the chip thickness, although this is usually called the depth of cut when cutting threads. With the compound rest positioned at the half-angle of the thread as previously explained, a single chip will form on the leading edge of the thread cutting tool. A chip would form on both sides of the tool if it were fed straight in with the cross slide to establish the depth of cut. In passing off, the two chips will collide or interfere with each other, which can cause the surface finish on the thread to be rough.

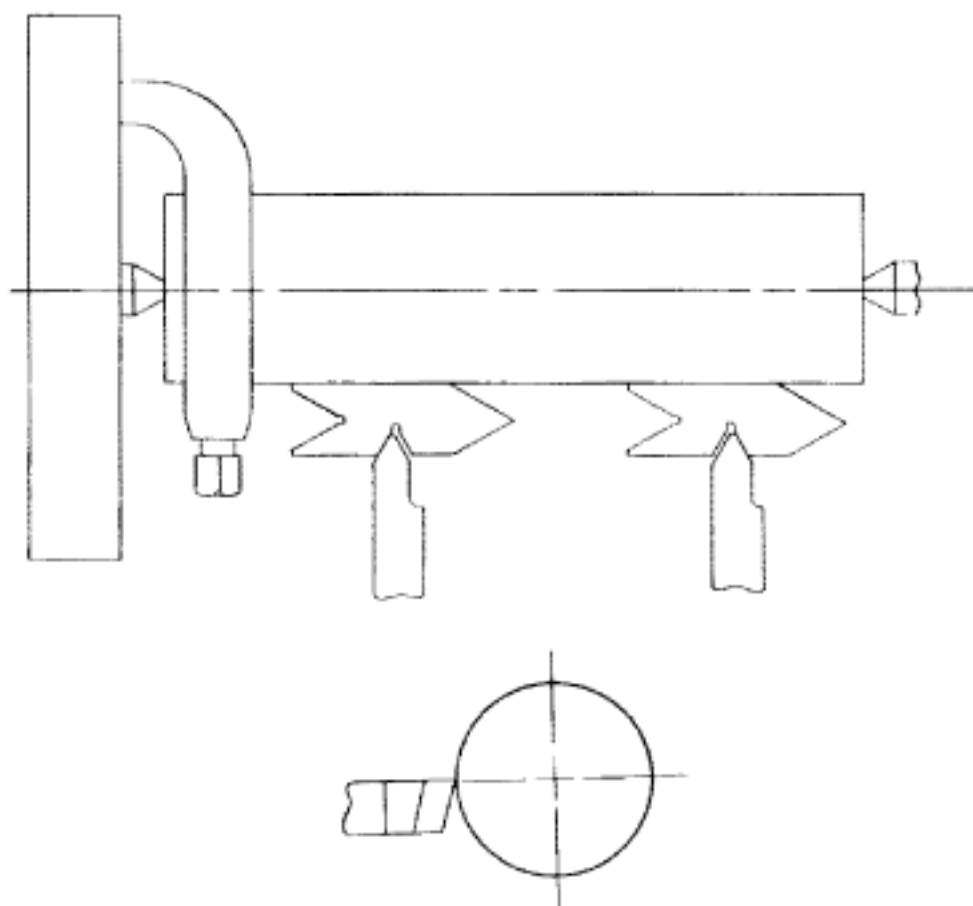


Fig. 13-15. Setting a thread-cutting tool with a center gage for cutting an American Standard Unified Thread Form.



Fig. 13-16. Cutting an American Standard Unified thread with the workpiece held between two centers.

No schedule which would fit each case can be given for the number and the size of cuts required to cut a thread to its finished size. The number of cuts and the depth of each cut will depend on the following factors: 1. the type, size, and class of the thread to be cut; 2. the diameter and length of the thread; 3. the diameter and length of the workpiece; and 4. the workpiece material. Often the first cut is a scratch cut only .001 inch to .003 inch (0.03 mm to 0.08 mm) deep, which is cut to check if the correct threads per inch are being cut. With this cut the workpiece will not be ruined if it is found that the required threads per inch have not been cut, and corrective measures can be taken. After the scratch cut, several fairly deep cuts are taken. Depending on the factors listed above, these cuts may be from .005 inch (0.13 mm) to .020 inch (0.50 mm) deep. As the thread groove gets larger the size of the cuts is reduced, with the final cuts being .001 inch to .0005 inch (0.03 mm to 0.013 mm) deep. For example, a typical schedule of cuts that might be used to cut a $\frac{1}{2}$ -13 UNC-2A thread would be: 2 cuts, .010 in.; 3 cuts, .005 in.; 4 cuts, .003 in.; .001 in. and .0005 in. cuts until the thread is finished to size. This schedule is based on moving the compound rest, positioned at 30 degrees, to set the tool for the depth of cut. Of course, under other circumstances this schedule may be different.

The procedure for cutting a right-hand American Standard Unified thread with a single point thread cutting tool on an engine lathe is shown in Fig. 13-19 and described further on in this section. In this illustration, the numbers refer to the positions of the thread cutting tool; the arrows



Courtesy of Hardinge Brothers Inc.

Fig. 13-17. Cutting an American Standard Unified thread on a precision toolmakers' lathe with work held in a collet chuck and on a center.

indicate the direction in which this tool is moving; and, the letters indicate the method by which the tool is moved as follows; L.S., carriage movement by lead screw and split nut; C.S., cross slide movement; M.L., manual longitudinal movement of carriage; and C.R., compound rest slide movement.

Referring to Fig. 13-19:

1. Start the lathe spindle. With the thread cutting tool just left of Position 1, move the cross slide inward until the tool just touches the workpiece. This position of the cross slide is the zero reference position. Before each cut through the thread groove, the cross slide must be at the zero reference position. To accomplish this, adjust the cross feed micrometer dial to read zero. If this cannot be done without moving the cross slide from the zero reference position, the micrometer dial reading must be memorized or recorded.
2. Move the thread cutting tool to Position 1 to be clear of the end

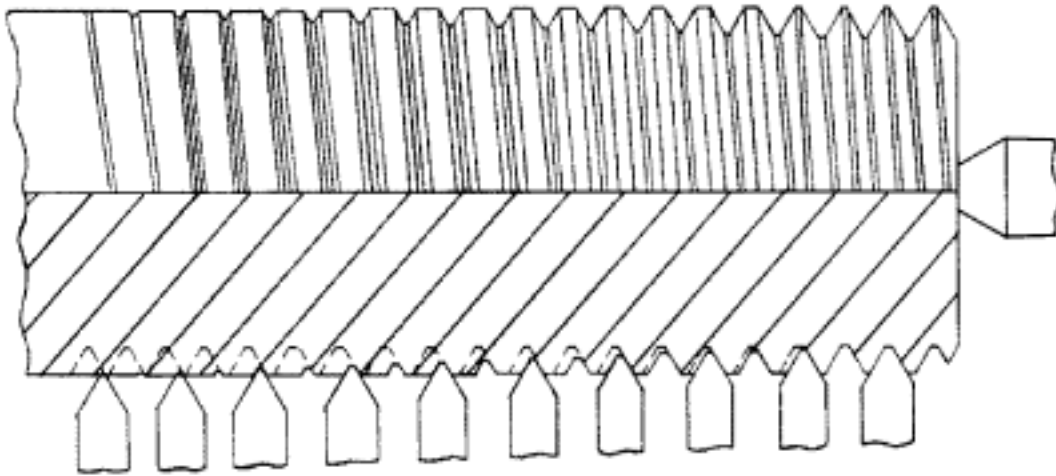


Fig. 13-18. Schematic drawing showing how succeeding cuts (starting from the left) cut the thread from start to finish.

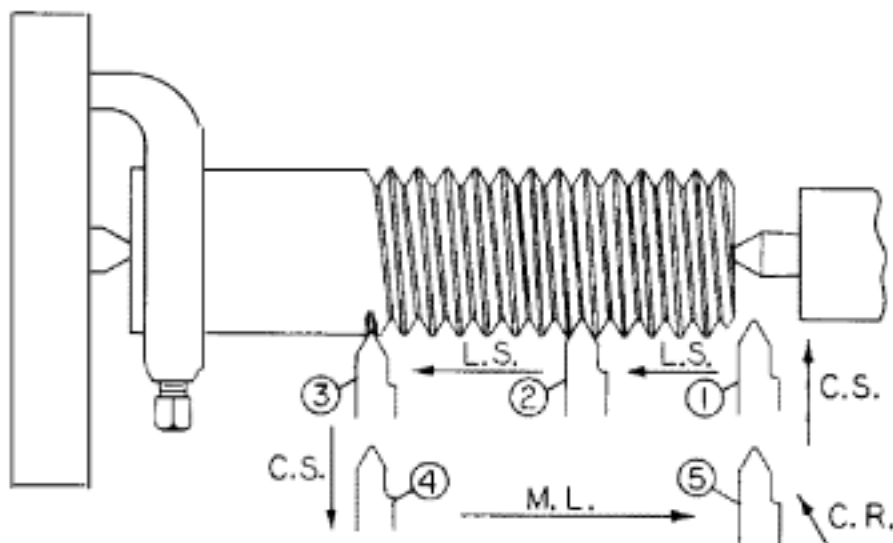


Fig. 13-19. Procedure for cutting an American Standard Unified thread. Arrows indicate direction of motion: L.S. Feed by lead screw; C.S. Feed by cross slide; M.L. Manual longitudinal feed; C.R. Compound-rest slide feed.

of the workpiece. Establish the depth of cut by moving the thread cutting tool inward the required distance with the compound rest. To obtain a precise depth of cut setting, the compound rest micrometer dial should be used. For the first scratch cut, the depth of cut should be .001 inch to .003 inch (0.03 mm to 0.08 mm).

3. The thread cutting tool is now in Position 1 and prepared to take a cut. Place the left hand on the cross feed crank handle and the right hand on the split nut lever. These positions should be maintained while the tool is cutting. Watching the thread chasing dial,

quickly engage the split nut lever when an appropriate graduation is opposite the reference mark.

4. The carriage advances the tool into the work to cut the thread groove, as shown by Position 2. Be prepared to withdraw the tool from the thread groove and to disengage the split nut at the end of the cut.
5. At the end of the cut, Position 3, quickly turn the cross feed crank handle to withdraw the thread cutting tool with the cross slide. When the tool is just clear of the thread groove, quickly disengage the split nut lever to stop the carriage movement. This order can be reversed when the thread ends in a groove or neck cut into the workpiece to provide clearance for the tool; i.e., first stop the carriage and then withdraw the tool. When this step has been completed, the tool should be in Position 4, clear of the workpiece.
6. Move the carriage manually from Position 4 to Position 5. Some lathes have a rapid traverse carriage feed which is used to move the carriage to Position 5 when long threads are cut. However, the operator must be certain that the rapid traverse will move the carriage toward the tailstock.
7. When the tool is in Position 5, stop the lathe spindle and check to see if the required threads per inch are being cut. Place a rule against the thread grooves and count the numbers of grooves cut in one inch. If available, a thread pitch gage (Fig. 13-21) may be used. If necessary, correct the quick-change gearbox setting. This step is performed only on the first cut; it is not repeated on the following cuts.
8. At Position 5, set the depth of cut for the next cut by moving the thread cutting tool inward with the compound rest slide. Use the micrometer dial to obtain a precise setting.
9. Move the cross slide inward to the zero reference position (Position 1) as determined by the reading on the cross feed micrometer dial. To eliminate the effect of the clearance between the cross feed screw and the nut, called backlash, the movement to the zero reference position must be made with the cross slide moving inward toward the lathe axis, or toward the workpiece. If by accident the cross slide is moved beyond the zero reference position, move the cross slide outward toward Position 5 until the backlash has been taken up, and then inward again to the zero reference position.
10. The thread cutting tool is now prepared to take the next cut. Except for Step 7, which is not repeated, repeat Steps 3 through 9 until the thread is finished to size. The lathe spindle needs to be stopped only to measure or gage the size of the thread.

The size of the thread is not usually measured directly when it is in the process of being turned in a lathe. The procedure of bringing it to size is a cut-and-try procedure. The appearance of the thread will give an indication when it is getting close to the finish size. When it is judged to be close to the finish size, it is gaged with the ring gage, snap gage, or mating part, after each cut, until the gage fits properly as described in Chapter 12. Essentially the same procedure is used when measuring the threads by the three-wire method or with thread micrometers.

On some engine lathes the longitudinal-feed handwheel on the carriage can be disengaged. This is done to avoid the influence of the unbalanced force on this handwheel as it rotates. This force can cause the carriage to feed unevenly when taking very light cuts such as when cutting threads. The uneven motion of the carriage results in errors appearing in the lead of the thread that is being cut; therefore, where possible, the handwheel should be disengaged when the carriage is engaged to the lead screw.

When cutting threads in steel and similar metals, a good grade of cutting fluid should be used to obtain a good surface finish on the threads. There are a good many proprietary cutting fluids. Generally lard oil or a heavy, sulfurized mineral oil will give good results. Threads should be cut in gray cast-iron without using any cutting fluid.

When the infeed of the tool is made with the compound rest set at 29 degrees, the right side-cutting edge rubs against, or burnishes, the "back" side of the thread, thereby improving its surface finish. Regardless of the angle of infeed, the "back" side of the thread is sometimes rough and must be cut with the right side-cutting edge of the tool in order to "clean up." The procedure for "cleaning up" the back side of the thread is shown schematically in Fig. 13-20. This must be done before the thread is to size because it is necessary to take several cuts through the thread groove, thereby enlarging it. Position 1, Fig. 13-20 shows the incompleted thread groove which has been cut using a 29- or 30-degree infeed of the tool. The tool is moved to cut with the right side-cutting edge by backing out

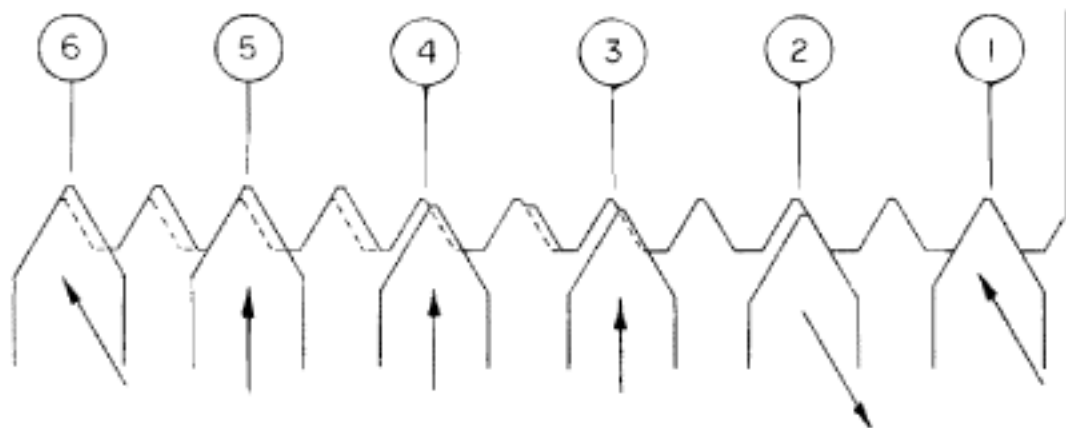


Fig. 13-20. Schematic illustration of procedure for "cleaning up" the back side of a thread.

the compound rest a distance of approximately .010 to .015 inch. Use the micrometer compound-rest feed dial to ascertain this distance and be sure to allow for the lost motion in the compound-rest feed screw. This is shown schematically in position 2, which shows the effect of this movement in relation to the thread. Actually, the movement is made before the cut is started, when the tool is clear of the thread. For the next several cuts the infeed of the tool is made with the cross slide which must be moved past the zero reference position. Positions 3, 4, and 5 show how the right side-cutting edge cuts the "back side" of the thread as a result of feeding with the cross slide. When the left side-cutting edge of the tool again touches the "front side" of the thread, as shown at position 5, a new zero reference position for the cross slide is obtained by adjusting the micrometer cross-feed dial to read zero or by memorizing the reading. The thread is then finished to size by infeeding the cutting tool with the compound rest as before (position 6).

It is sometimes necessary to align a thread-cutting tool with an existing thread groove. For example, this must be done if the tool has to be replaced before the thread is finished; or when the thread is rough cut with a roughing tool and finished with a finishing tool which is sometimes the procedure used for cutting large threads. The new tool must be set to a center gage and to the height of the centers as before. It is aligned with the thread by engaging the lead screw and turning the lathe forward, until the carriage has moved the tool opposite the thread grooves. The spindle is then stopped, but the lead screw is kept in engagement. This motion of the carriage takes up any lost motion in both the gearing and the split nut. The tool is then positioned to fit into a thread groove by moving the cross slide and the compound rest slide until the threading tool, as determined visually, is positioned to a perfect fit in the thread groove. The movement of the compound rest slide, being at an angle, moves the tool both longitudinally and perpendicularly to the lathe axis. Errors caused by the perpendicular movement of the compound rest can be corrected by the movement of the cross slide until the tool fits in the thread groove. When the tool is accurately positioned in the thread groove, the reading of the cross-feed dial determines the new zero reference position for cutting the thread, or it can be set to read zero. The tool is then withdrawn from the thread. The carriage is disengaged from the lead screw and again is brought to the starting position (position 1, Fig. 13-19) to cut the thread.

Cutting Left-Hand Threads

Left-hand threads are cut by reversing the direction of rotation of the lead screw so that the carriage will move from left to right. The compound rest is positioned at 29 or 30 degrees; however, the position is not the same as that for right-hand threads. It is positioned so that the right side-cutting edge of the threading tool will form the chip. The left-hand thread-cutting tool should have a 10- to 17-degree relief angle on the flank below the right side-cutting edge, and an 8- to 12-degree relief angle

below the left-hand cutting edge. The procedure for cutting the left-hand thread is essentially the same as for cutting the right-hand thread except for the differences explained below. If possible, the part with the left-hand thread should be designed with a neck, or groove, ahead of the cut. This provides a space in which the tool can be positioned in starting the cut. If a neck cannot be cut ahead of the thread, the best procedure is to start cutting the thread manually. The zero reference position of the cross slide is established in the same manner as for right-hand threads. The infeed of the tool is made with the compound-rest slide before the start of each cut. The cut is started by turning the spindle by hand, with the carriage engaged to the lead screw. When the tool approaches the start of the thread it is carefully moved into the thread groove, with the cross slide. The cross slide is moved in until the micrometer dial reaches the zero reference position, while at the same time slowly continuing to turn the spindle manually. When the zero reference position has been reached, the spindle is then started, and the remainder of the thread is finish cut by power.

Cutting Metric Threads

The procedure used to cut metric threads on lathes having an *inch lead screw* is different from the procedure used to cut inch threads because the split nut must be engaged continuously from start to finish. It cannot be disengaged at the end of each pass because it would not be possible to pick up the thread again, unless the procedure previously described to align a thread cutting tool in an existing thread groove is used. This procedure is not intended to be used during the normal operation of thread cutting.

When cutting a thread with a split nut continuously engaged, the spindle speed must be slow enough to allow the operator to have complete control over all of the lathe movements at all times. Thus, the spindle speed used depends upon the judgment of the operator. He should consider his experience in cutting metric threads on inch lead-screw machines, his experience on a particular machine, the ease with which a particular machine can be manipulated, and the size and type of thread being cut. All other machine settings made in preparation for cutting threads are the same as before, except that the 127-tooth translating gear must be used in the gear train.

The procedure for cutting a right-hand metric thread on an inch lead-screw lathe is given by the following steps. Refer again to Fig. 13-19, in which the numbers refer to the positions of the thread cutting tool and the arrows to the direction of tool travel; however, *the letter symbols should be disregarded*.

1. Start the lathe spindle. With the thread cutting tool just left of Position 1, move the cross slide inward until the tool just touches

the workpiece. The position of the cross slide is the zero reference position. Before each cut is started, the cross slide must be at the zero reference position. To accomplish this, adjust the cross feed micrometer dial to read zero. If this cannot be accomplished without moving the cross slide from the zero reference position, the micrometer dial reading must be memorized or recorded.

2. Stop the lathe spindle and move the thread cutting tool to Position 1. Establish the depth of cut by moving the tool inward the required distance with the compound rest. To obtain a precise depth of cut setting, the compound rest micrometer dial should be used. For the first scratch cut, the depth of cut should be .001 inch to .003 inch (0.03 mm to 0.08 mm).
3. With the thread cutting tool in Position 1, slowly turn the lathe spindle by hand and engage the split nut.
4. The thread cutting tool is now prepared to take a cut. Place one hand on the lathe spindle starting lever (or button) and the other on the cross feed crank handle. These positions should be maintained while the tool is cutting. Then start the lathe spindle.
5. The carriage advances the tool into the work to cut the thread groove, as shown by Position 2. Be prepared to withdraw the tool from the thread groove and to stop the spindle at the end of the cut.
6. At the end of the cut, Position 3, quickly turn the cross feed crank handle to withdraw the thread cutting tool with the cross slide. When the tool is just clear of the thread groove, quickly stop the lathe spindle. Because the spindle may coast before coming to a complete stop, the thread cutting tool must always be withdrawn first when cutting threads by keeping the split nut continuously engaged. When this step is completed, the tool should be in Position 4, clear of the workpiece.
7. Start the spindle rotating in the reverse direction of rotation. This will also reverse the direction of rotation of the leadscrew causing the carriage to move toward the tailstock. By this method, carefully move the carriage from Position 4 to Position 5. Then stop the spindle. Do not just reverse the lead screw and run the spindle in the forward direction to move the carriage. Some operators will switch directly from forward to reverse spindle rotation. With practice this can be done, but caution must be used. This procedure can be hard on the lathe mechanism; and on some lathes it can cause the motor to overheat when done repeatedly.
8. With the spindle stopped and the thread cutting tool in Position 5, check to see if the correct thread pitch is being cut using a metric thread pitch gage, as seen in Fig. 13-21. It is not convenient to

Internal Threads

Internal threads are cut with a single point cutting tool ground to the shape of the thread, or with a tap. The single point internal thread cutting tool may be a cemented carbide insert or a high speed steel tool bit. These are clamped onto a boring bar as in Figs. 7-12 and 13-23. When the threads to be cut do not penetrate into the bore too deeply, an internal threading tool may be made from a piece of solid high speed steel, which also acts as a boring bar. A typical internal thread cutting tool of this type made from a high speed steel tool bit is shown in Fig. 13-22. The relief angles on internal cutting tools must be large enough to clear the curvature of the bore, as well as the sides of the thread.

The first step in producing an internal thread is to drill or bore a hole to a size equal to the minor diameter of the thread. Then the lathe can be set up to cut the thread. First the compound rest is turned and clamped in the position shown in Fig. 13-23, or 30 degrees with respect to the cross slide. It may also be positioned 180 degrees from this position. However, this may result in interference between the compound rest and the workpiece or the chuck, unless the boring bar is extended longer than necessary. The position shown in Fig. 13-23 is the preferred position. With the thread cutting tool in place, the height of the cutting edge is positioned exactly at the height of the lathe centers, which may be done by comparing the height of the cutting edge and the height of the tailstock center with a surface gage. Using a center gage as shown in the illustration, the thread cutting tool is positioned to cut the correct thread angle on the workpiece and then it is clamped in place. To complete the setup, the lathe is adjusted to cut the required number of threads per inch or pitch, whichever the case may be.

The method of cutting the thread is the same as for cutting an external thread. Using the micrometer cross-feed dial as a reference, the cross slide is set at a zero reference position. After each cut, the cross slide is moved *forward* to clear the tool. After the tool is withdrawn from the bore it is returned to the zero reference position, as established by the cross-feed

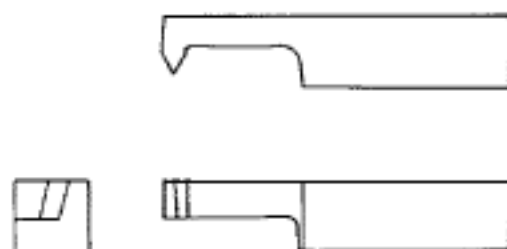


Fig. 13-22. Typical internal thread-cutting tool made from a high-speed steel tool bit which also acts as a boring bar.

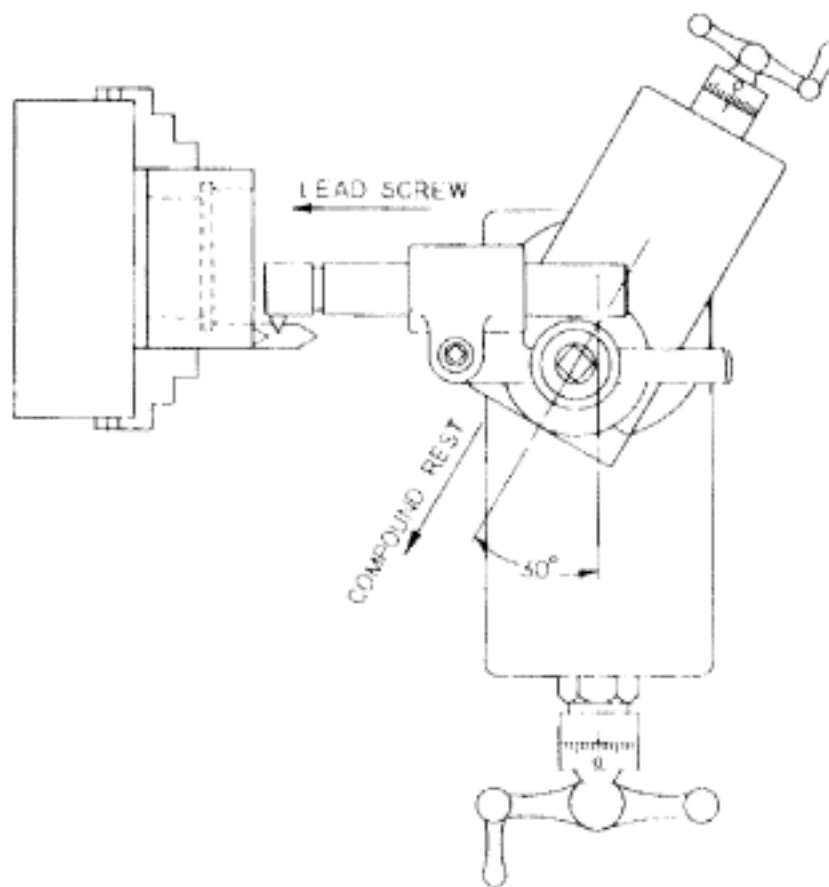


Fig. 13-23. Setup for boring thread and method of setting thread boring tool.

dial, in preparation for the following cut. The infeed of the tool to set the depth of cut for each pass is made with the compound-rest slide. The size of the thread is determined by the fit of a thread plug-gage or by a mating part. Small-diameter holes that have threads cut deep into the hole are sometimes finished by running a tap through the hole.

It is sometimes difficult to see the end of the cut when boring threads in a hole, and it is difficult to tell exactly when to actuate the split-nut lever to disengage the carriage from the lead screw. This becomes unnecessary if the lead screw feed can be used in feeding the tool out of the hole, rather than into the hole, in order to cut the thread. Left-hand threads can be bored in this manner with the setup shown in Fig. 13-23; however, right-hand threads must be bored feeding *into* the hole with this setup. On lathes equipped with a long taper, or cam-lock spindle nose, right-hand threads also can be cut with the lead screw feed feeding the tool out of the hole by cutting the threads with the lathe spindle turning in the reverse, or clockwise, direction. When this procedure is used, the lathe must be set up as shown in Fig. 13-24, in order to bore the thread. The thread is actually cut on the "back side" of the bore, as shown. The thread-cutting tool is positioned in the internal groove to start the cut with the cross slide in the zero reference position, as determined by the cross-feed

dial. With the lathe spindle running in reverse, the split nut is engaged at the proper time (determined by the threading dial), and the carriage is moved from left to right to bring the tool out of the bore. The infeed of the tool is made with the compound rest, and the cross slide is backed off to allow the tool to clear the threads as it reenters the hole. The cross slide is then positioned to the zero reference position in preparation for the next cut. This procedure is repeated until the thread is cut to size. Since the tool is clear of the work at the end of each cut, no special care is required when actuating the split-nut lever to disengage the carriage from the feed screw. Higher spindle speeds may be used because there is less danger of the threading tool feeding too far into the bore. This procedure is not recommended for lathes having a threaded spindle nose because of the possibility of the chuck or faceplate unscrewing itself from the spindle. Likewise, boring bars with threaded ends cannot be used.

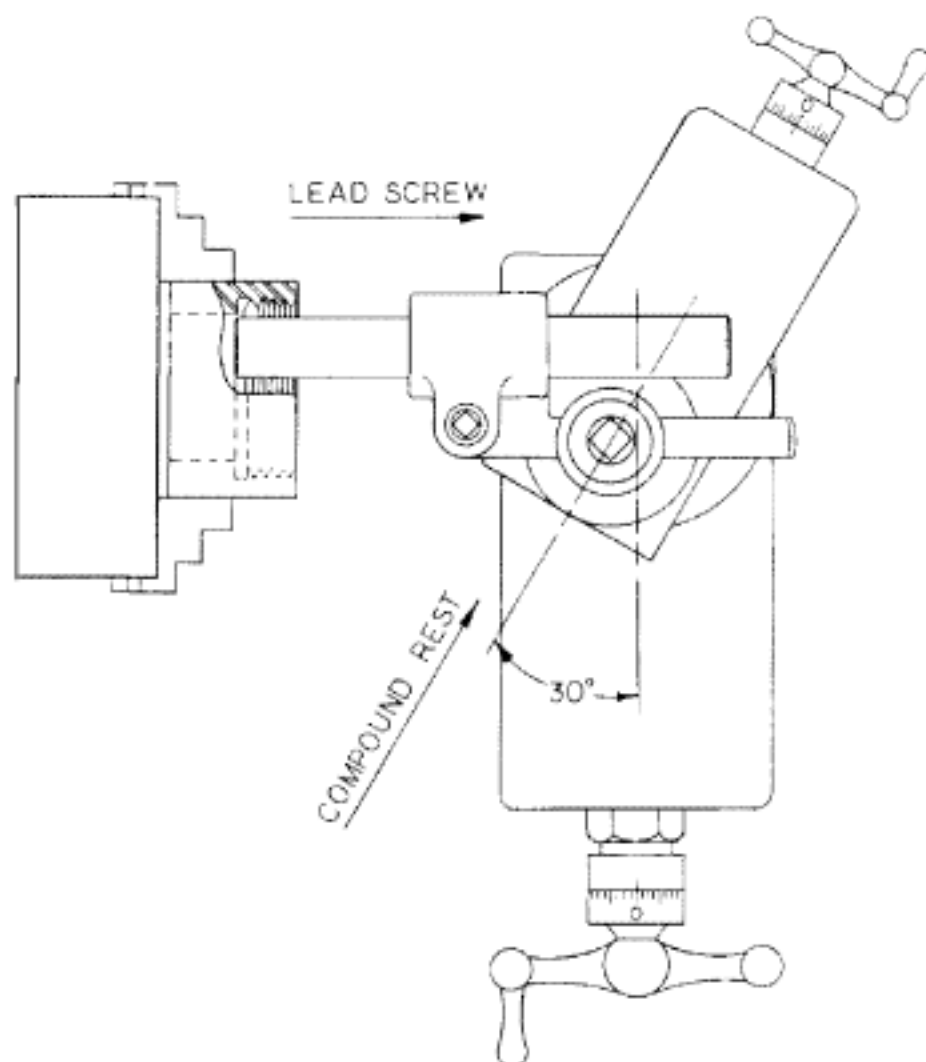


Fig. 13-24. Alternate method of boring a right-hand thread on lathe equipped with tapered, or cam-lock, spindle nose.

Tapping

Internal threads can also be cut in a lathe with a tap, as shown in Fig. 13-25. Usually, but not always, threads that are approximately $\frac{3}{4}$ inch (19 mm) and less in diameter are cut with a tap, although on many occasions larger threads are also cut with a tap. It is first necessary to drill a hole having the correct size for the tap. The thread may then be cut complete with the tap. Sometimes the thread is cut undersize with a single point tool and finished with a tap. This procedure produces a uniform sized thread throughout the hole and reduces the torque, or effort, required to cut the thread with the tap.

Figure 13-25 illustrates the setup and the procedure for tapping a thread on an engine lathe. First the tap is clamped in a long-handle type tap wrench. The threaded end of the tap is then placed in the hole in the workpiece and the center hole in the shank end of the tap is placed on the tailstock center, with the handle of the tap wrench resting on the compound rest. Before starting, and from time to time when tapping, a suitable grade of cutting oil is applied to the tap and inside of the hole, except when tapping cast iron, which is cut dry. The tapping operation is performed by manually turning the lathe chuck with one hand while at the same time turning the tailstock handwheel with the other hand to cause the tailstock center to follow the tap through the hole. The tailstock center must always be seated in the center hole of the tap as the tap progresses through the hole. A chuck wrench may be used as shown in Fig. 13-25 to provide added leverage, when required, in turning the chuck. While cutting, the tap is prevented from turning by the tap wrench handle resting on the compound rest. When the thread is finished the tap is removed by manually turning the lathe chuck in the opposite direction. As the tap backs out of the hole, the shank end must be continually supported by the tailstock center. The tap is prevented from turning by the other handle on the tap wrench which is made to rest on the cross slide.

Cutting Acme Threads

The general procedure for cutting an Acme thread is the same as for cutting a 60-degree American Standard Unified and National thread. An Acme threading tool with a 29-degree included angle must be used. It is set with an Acme thread gage after the compound rest has been positioned at $14\frac{1}{2}$ degrees, with respect to the cross slide. The micrometer cross-feed dial is used to establish the zero reference position of the cross slide and the cross slide is returned to this position before starting each cut. The infeed of the cutting tool, in order to obtain the depth for each cut across the work, is made with the compound rest so that the tool will cut with

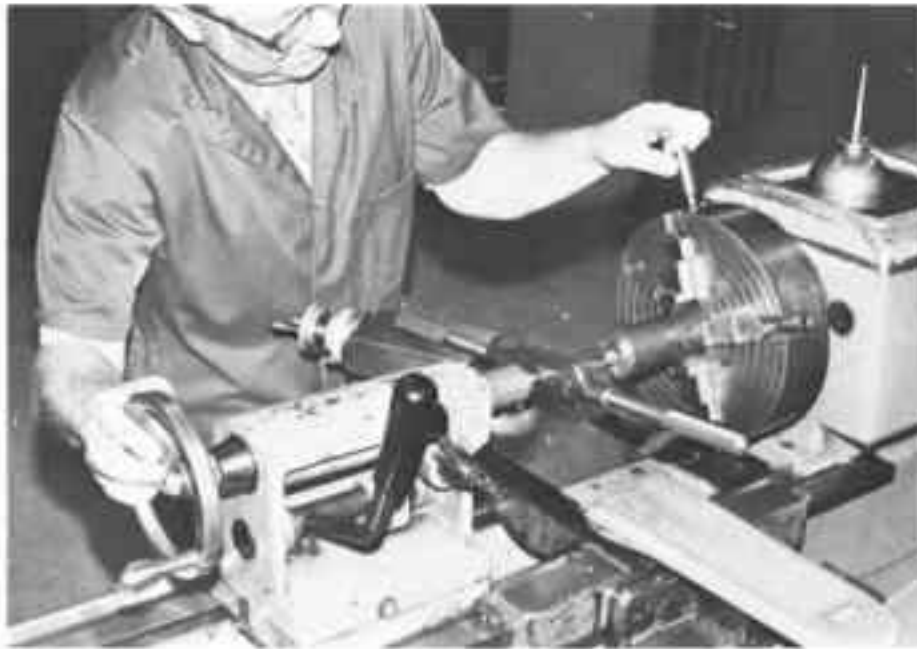


Fig. 13-25: Setup and procedure for tapping a thread on an engine lathe.

only two cutting edges, namely, the right side-cutting edge and the end-cutting edge. If required, the "back side" of the thread can be cleaned up using the same procedure previously described. Many Acme threads are very long, as shown in Fig. 13-26, and the work must be given support to prevent it from being deflected by the cutting forces. A steady rest cannot be used when cutting the thread as in the illustration, because it would interfere with the movement of the carriage; therefore, a follower rest must be used to support the work.

Taper Threads

Taper threads should be cut with a taper attachment if at all possible. When the required taper is obtained by setting the tailstock off-center, the thread will not advance at a uniform rate, nor form a true helix. This "drunken thread," or error, is caused by the angularity between the driving dog and the driven plate, which causes the rotating speed of the work to vary during each revolution. The bearing surface between the lathe centers and the work centers (when the tailstock is offset) is another cause of inaccuracy, because as the work centers wear rapidly due to the poor bearing surfaces, the angle of the taper is changed as the tailstock spindle is tightened. The amount of the error depends upon the angle of the taper and the distance that the centers must be offset.

The cutting tool must be set having the included angle of the thread bisected by a line that is perpendicular to the axis of the work, as shown at A, Fig. 13-27. It cannot be set by holding the center gage against the tapered surface as shown at B, which would cause the form of the thread



Courtesy of the Clausing Division of the Atlas Press Company

Fig. 13-26. Cutting an Acme thread for a lead screw.

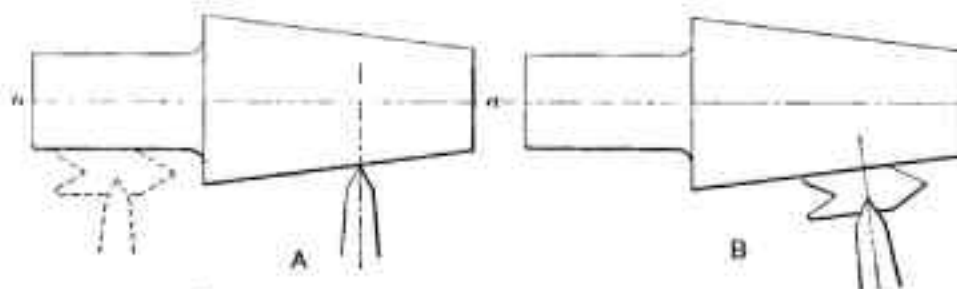


Fig. 13-27. Correct and incorrect positions of the tool for taper thread cutting.

that is cut to be incorrect. The taper attachment can be adjusted to the correct taper by placing a taper gage (not threaded) of the required taper between centers. A dial test indicator is fastened in the toolholder and the contact point of the indicator is placed against the gage at the height of the lathe centers. The taper attachment is adjusted until the hand of the indicator does not move when it is traversed from one end of the gage to the other.

Cutting Multiple Threads

The general procedure for cutting multiple threads is the same as for cutting single threads. Figure 13-28 shows a multiple thread and the method used to cut the thread. The first thread is cut as though a single thread were being cut, leaving the work as shown at A. When this cut is finished the work must be indexed one-half of a revolution (for a double thread) without disturbing the position of the lead screw or car-

riage. This will place the tool in a position to cut the second thread groove, centrally, between the grooves of the first thread, as indicated by the tool shown in dotted lines. The second groove is then cut producing a double thread, as shown at B.

The lead, which is the distance that the thread will advance in one turn, is not equal to the pitch—for multiple threads. For example, in Fig. 13-28, the pitch of the double square thread is $\frac{1}{4}$ inch and the lead is $\frac{1}{2}$ inch. If this had been a triple square thread the lead would have been $\frac{3}{4}$ inch. The cutting tool must travel a distance equal to the lead per revolution of the work. In the case of multiple threads, the lathe must be set to cut an equivalent number of threads per inch, which is dependent upon the lead of the thread, and which is not equal to the actual number of threads per inch. The equivalent number of threads per inch can be calculated by the following formula:

$$\text{Equivalent threads per inch} = \frac{1}{\text{Lead}} \quad (13-18)$$

For the threads in Fig. 13-28 the equivalent threads per inch would be:

$$\text{Equivalent threads per inch} = \frac{1}{\text{Lead}} = \frac{1}{\frac{1}{2}} = 2$$

The actual number of threads per inch in this case is four.

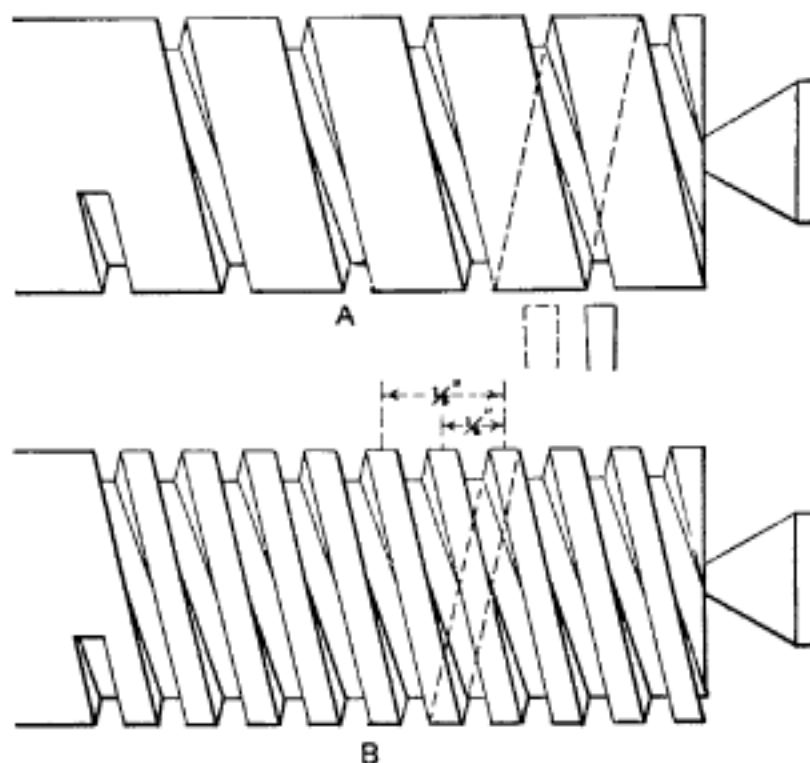


Fig. 13-28. View showing how a double square-thread is cut.

The indexing, or spacing, of multiple threads can be done in several different ways. One method is to disengage the gear on the stud (Figs. 13-10 and 13-12) from the idler, or quadrant gear, and turn the stud shaft a predetermined amount while the idler, or quadrant gears remain stationary. The stud gear is then reassembled to the stud, in engagement with the idler, or quadrant, gear. The stud shaft is turned the predetermined amount rotating the stud gear a given number of teeth. Before starting to turn the stud shaft a stud-gear tooth and the mating tooth-space on the idler, or quadrant, gear are marked with chalk. If the number of teeth to be indexed is 16, then the sixteenth tooth from the marked tooth on the stud is placed in the marked tooth space in the idler, or quadrant, gear.

The number of teeth on the stud gear that must be indexed is determined as follows:

1. Count the number of teeth through which the stud gear is turned when the spindle is turned one revolution.
2. Divide this number by the number of threads that are to be cut.
3. The answer must be a whole number, or the stud gear must be changed.

Example 13-5:

A double thread is to be cut on a lathe having a 24-tooth stud gear.

When the spindle is turned one revolution it is found that the stud gear turned through 18 teeth.

The number of teeth which the stud gear must be indexed is:

$$\frac{18}{2} = 9 \text{ teeth}$$

One method that can sometimes be used to obtain the spacing for multiple threads is to engage the carriage and the lead screw and move it forward just far enough to eliminate the lost motion in the split nut and the gearing. Attach a dial test indicator to the lathe bed of the carriage and place a stack of precision-gage blocks between the indicator and the carriage. The length of the stack of gage blocks is made equal to the pitch (not the lead) of the thread. Zero the indicator to the gage blocks and then remove the blocks. Disengage the feed-thread change handle or the quadrant gear (Fig. 13-12) to disconnect the lead screw from the spindle. Turn the lead screw manually, with the carriage engaged, until it comes against the dial test indicator making it read zero. Reengage the feed-thread change handle or the quadrant gear. The carriage can also be moved the required distance, using the micrometer carriage positioning attachment, Fig. 6-23. When multiple threads must be cut frequently, a special indexing faceplate, or a special tool block that accurately spaces the cutting tool can be used.

The thread-chasing dial can also be used to space double threads whenever the equivalent threads per inch are an odd numbered thread as, for

example, 5, 7, 11, etc. The first thread is cut using the graduations on the thread-chasing dial that represent one-inch movements of the carriage, while the second thread is cut by the alternate graduations which represent $\frac{1}{2}$ -inch movements of the carriage. If the equivalent threads per inch end in a " $\frac{1}{2}$," such as $5\frac{1}{2}$, $7\frac{1}{2}$, $11\frac{1}{2}$, etc., the first thread is cut by any one graduation, say, number 1, and the second thread is cut by the graduation that represents a two-inch movement of the carriage. In the case of the dial shown in Fig. 13-13, this would be graduation number 3. Graduations 2 and 4 could also be used. The use of the thread-chasing dial as a means of spacing double threads is recommended whenever possible. The procedure to use when the thread dial is employed to space the threads, is to alternately cut one thread groove and then the other so that both thread grooves are finished at nearly the same time.

Turret Lathes, Production Lathes, and Vertical Lathes

While engine lathes are very versatile and are easily adapted to perform a wide range of different operations, generally they do not meet the demand for turning parts in large quantities and at high production rates. This demand is met by other types of high production turning machines, such as turret lathes, single and multiple spindle automaties and, increasingly, numerically controlled lathes. These lathes may be designed with a conventional horizontal spindle or with the axis of the spindle vertical. They may be controlled manually, such as conventional turret lathes, or they may be controlled automatically by mechanical mechanisms or by electronic controls.

There is a class of medium sized to very large vertical spindle lathes that are not intended for high production manufacturing. These machine tools, usually called vertical boring mills, are designed to perform turning type operations on workpieces that are more convenient to lay down and to hold on a horizontal faceplate surface than on the vertical faceplate surface of a conventional lathe. Many medium and large sized parts are machined more conveniently in this manner. Often these machine tools are required to bore large, deep holes, in which case the boring bars or long tool holding slides are attached in a vertical position. Held in a horizontal position, these bars or slides would bend or sag appreciably under their own weight, which would make it difficult to bore an accurate hole. Many of these machines are operated by numerical control, which enables them to machine complex contoured surfaces in addition to performing other more conventional operations.

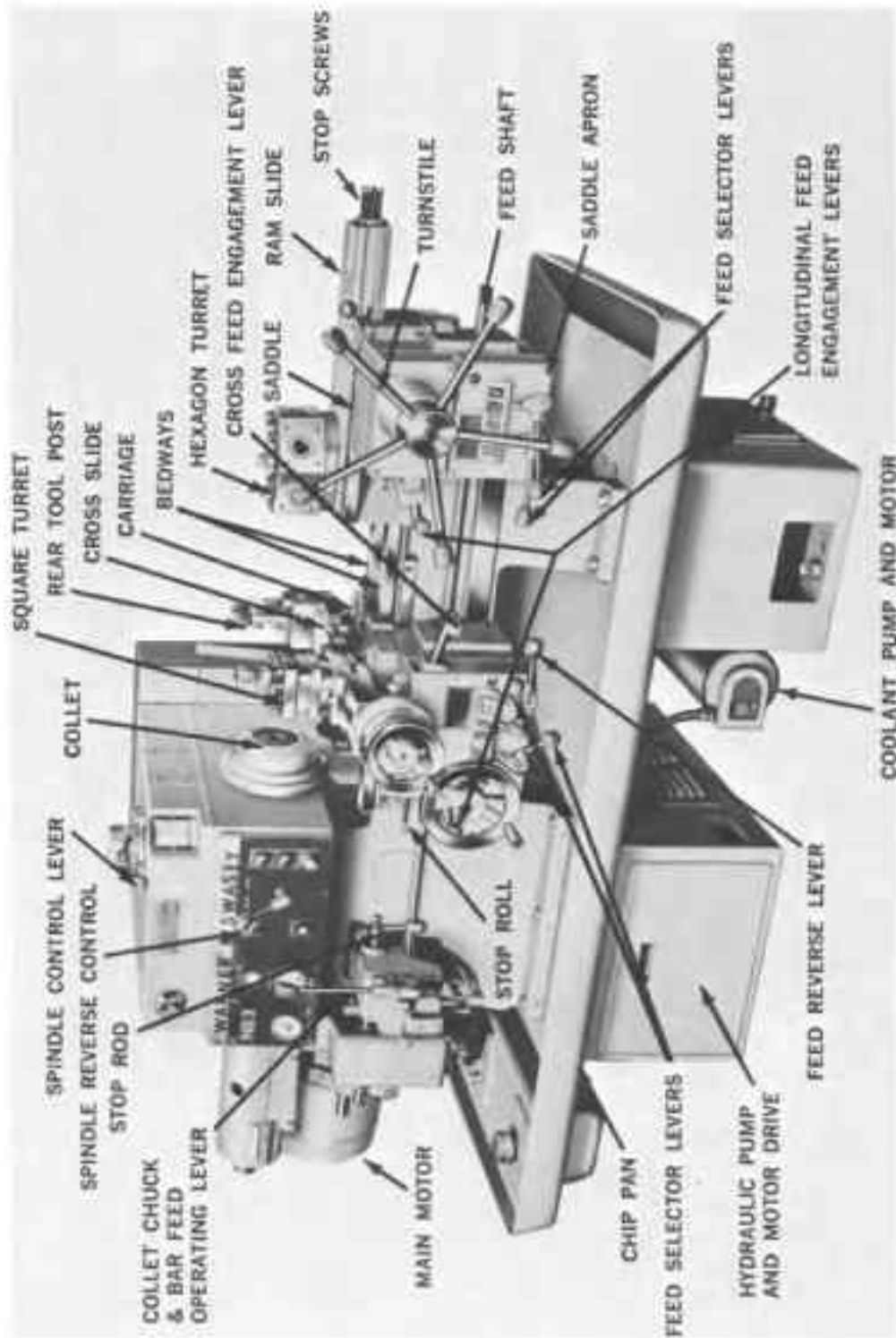
Turret Lathes

The characteristic feature of a turret lathe is the turret which is located at the rear of the bed in the position occupied by the tailstock on engine lathes. The turret has six faces, on each of which a variety of cutting tools can be mounted. The turret can be indexed to bring the cutting tools on one of the turret faces into a position to perform cutting operations on the part, while the tools on the other turrets are held con-

veniently out of the way. The turret lathe also has a carriage which, like the engine-lathe carriage, provides a cross-feed and a longitudinal-feed motion to the tools that are mounted on it. Turret lathes usually have two aprons, one for the carriage and the other for the saddle upon which the turret is mounted. Both of these aprons contain selective gear trains which provide the available feed rates. The necessary handwheels, handles, and levers for actuating these feeds are mounted outside of the apron. Normally, turret lathes do not have a compound rest, although one can be mounted on the cross slide, in some cases, as an attachment. The headstock contains the spindle and the gears for making the spindle turn at the selected revolutions per minute.

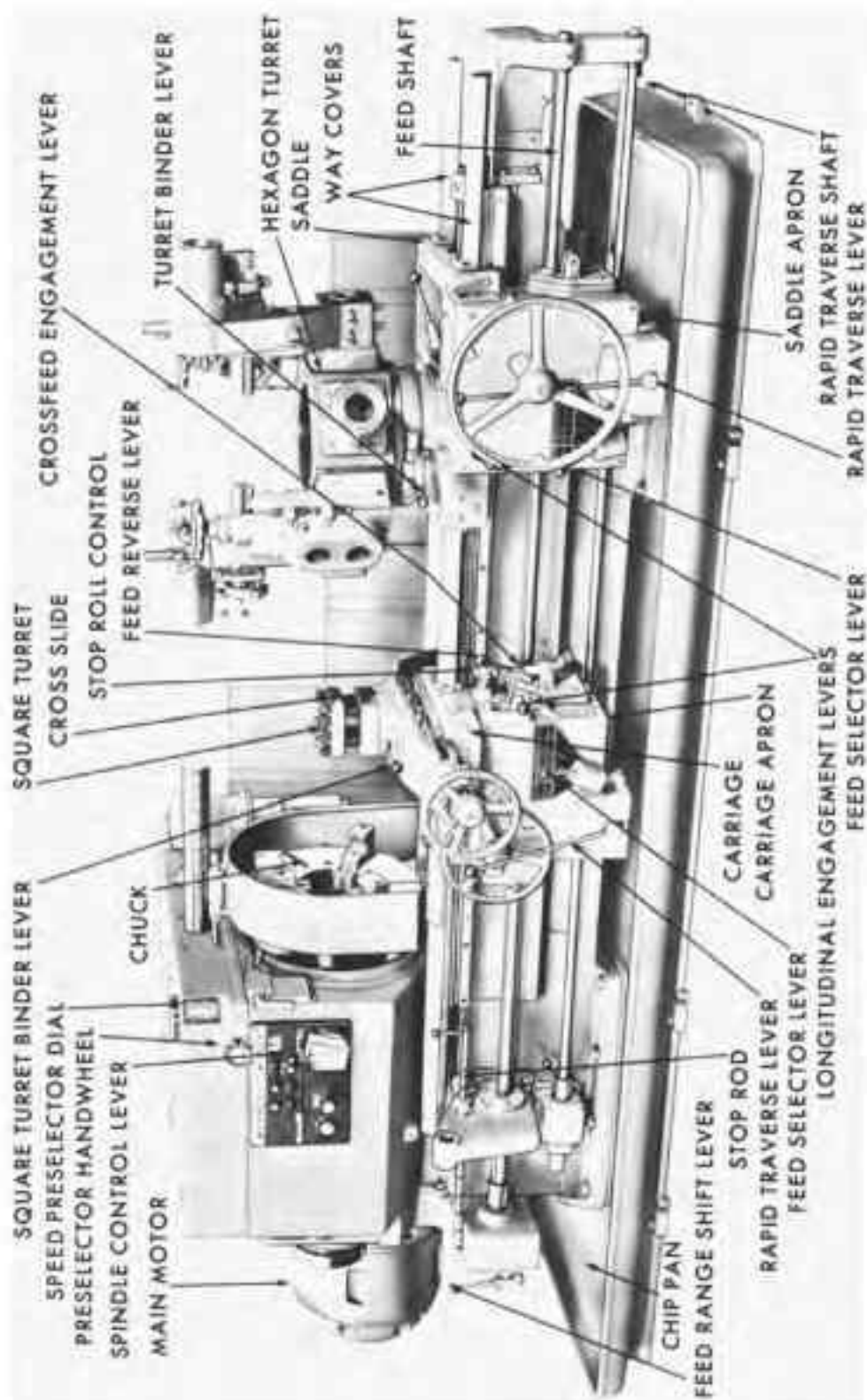
There are two basic types of turret lathe constructions, the ram type shown in Fig. 14-1 and the saddle type shown in Fig. 14-2. The hexagon turret of the ram type turret lathe is mounted on a ram slide which can be rapidly moved toward or away from the spindle, by the turnstile. It can also be fed by an automatic power feed toward the spindle. Stop screws provide a positive stop and an automatic disengagement of the longitudinal feed. When the ram is brought back, the turret is automatically indexed to bring the next face into the operating position. The ram slide moves easily and can be quickly handled by the operator. It slides on a saddle that can be fastened to the lathe bed at different distances from the spindle. When the turret lathe is operating, however, the saddle is fastened to the bed and remains stationary. The carriage is provided with both forward and reverse longitudinal and cross feeds. The longitudinal feed is provided with a positive stop which also disengages the automatic feed. The cross slide has trip dogs that disengage the automatic cross feed; however, these dogs can be passed over manually and the tool can be positioned for taking a cut by reading the graduations on the micrometer cross-feed dial. The cross slide is a bridge type which, like the cross slide on an engine lathe, bridges the opening in the lathe bed. It is finish machined on the top and sides. The top has a T-slot which provides an anchor for the bolts used to clamp tool blocks in place. Two standard tool blocks that satisfy many of the requirements are the rear tool post for cutting from the back side of the cross slide and the square turret which is mounted in the front. The square turret can hold four different cutting tools and it can be indexed to bring the appropriate cutting tool into position for taking a cut. The ram type turret lathes are usually smaller than the saddle type turret lathes and are generally used to machine smaller parts.

The saddle type turret lathe, Fig. 14-2, is usually a heavier and larger machine that is used to machine comparatively large parts. On the saddle type turret lathe the turret is mounted directly on the saddle which is moved longitudinally back and forth by the handwheel. It can also be moved back and forth by means of a power rapid-traverse. Automatic power feeds are used to feed the saddle toward the headstock when taking cuts. A stop roll, located inside of the bed, provides a positive stop and disengages the feed of the saddle. It can be passed over by swinging a master stop out of position. The turret on saddle type turret lathes must



Courtesy of The Wornat & Sweeney Company

Fig. 14-1 Principal parts of a ram type turret lathe.



Courtesy of The Warner & Swasey Company

Fig. 14-2. Principal parts of a saddle type turret lathe.

be indexed by hand. It is rigidly supported by the saddle at all times, in contrast to the ram on ram type turret lathes, which may well overhang the saddle a considerable distance when taking cuts. For the lighter work generally done on ram type turret lathes this is usually no disadvantage whereas on saddle type turret lathes, which usually take heavier cuts, the rigid support of the saddle is a very decided advantage. Some saddle type turret lathes are equipped with a cross-sliding saddle. This provides a cross-feed direction to the entire turret which can be used for internal recessing and facing operations. Standard turning and boring heads are shown mounted on the faces of the turret in Fig. 14-2. When these are removed, the face of the turret appears the same as in Fig. 14-1. The carriage of the saddle type turret lathe, Fig. 14-2, is generally a side-mounted or "side-hung" type, although a bridge type cross slide can be provided. The bridge, however, places a limit on the diameter of the work that can be placed over it. The advantage of the side-mounted carriage is that the cutting tool held on this carriage is rigidly supported when turning larger diameters. The side-mounted carriage is supported by one of the ways on the top of the bed and by the way that is machined on the front side of the bed. It can be moved in two directions longitudinally by the apron handwheel, by rapid traverse, and by automatic power feed. A stop rod which engages a micrometer stop-roll on the carriage provides an accurate means of positioning the carriage in the longitudinal direction. It will also automatically disengage the power longitudinal feed when the carriage contacts the stop roll. The automatic cross feed is disengaged by stop dogs which can be passed over by the manual cross feed. The micrometer cross-feed dial is used to accurately position the tool for establishing the depth of cut.

Turret lathes are also classified as bar and chucking machines. Bar type turret lathes are used to machine parts directly from raw bar stock. The bar is placed through the spindle of the lathe. One end of the bar is supported by a bar chuck that is held on a bar feed rack which is attached to the end of the headstock. The other end of the bar is held in a collet chuck which is located inside the spindle nose. When the collet chuck is opened, the bar stock is fed out of the spindle by the bar feed, against a positive stop located on one of the turret faces. The collet chuck is then closed and the part is made from that portion of the bar stock protruding from the nose of the spindle. As a final operation, it is cut off with a cut-off tool. Chucking type turret lathes are used to hold the work in a three- or four-jaw chuck, on a faceplate, or in a fixture that is held on the faceplate while performing the various turning and boring operations. Most turret lathes are designed to be readily converted from one type to the other. Ram type turret lathes, particularly, can be readily converted for either bar work or chucking work. The work done on ram type turret lathes is probably equally divided between bar work and chucking work. Saddle type turret lathes are more frequently used to do chucking work.

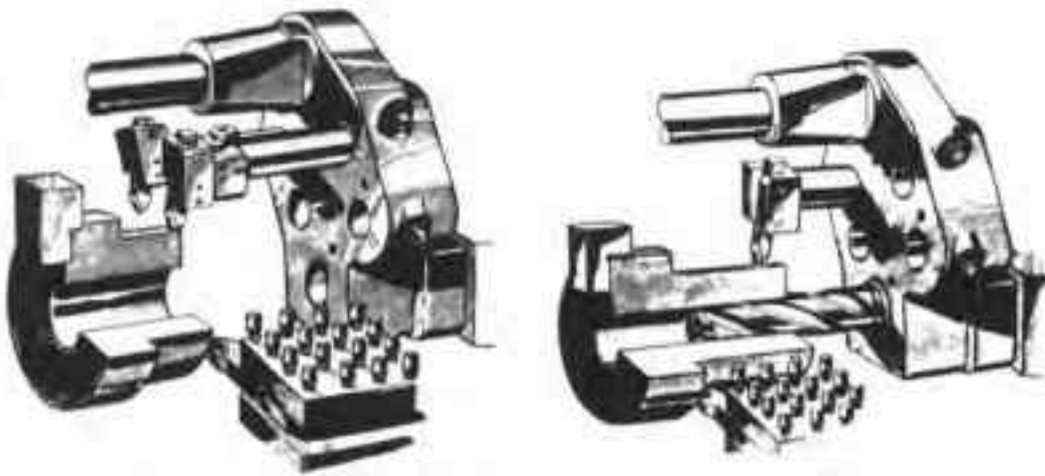
Turret-Lathe Tooling and Machining Methods

The turret lathe is capable of performing all of the operations that can be performed on an engine lathe. The method by which these operations are performed on the turret lathe sometimes differs from the method used on an engine lathe because of the tooling that is available for use on the turret lathe. Most of the turret-lathe tooling is standard tooling that can be used to do a variety of work. Some special tooling is also used, if required by the complexity of the workpiece, or if the quantities to be produced are large enough to warrant the extra expense necessary to obtain faster production.

Standard turret-lathe tools are classified as chucking tools, cross-slide tools, and bar tools. Chucking tools are held on the turret and are used to machine parts that are held in three- or four-jaw chucks. Typical of the parts made in this way are large gear blanks, large collars, large bushings, pulleys and sheaves. Bar tools are used to make parts from bar stock such as small gear blanks, small bushings, shafts, studs, screws, and bolts. Cross-slide tools can be used for both classes of work. One additional category of tools should be mentioned and this is work-holding devices. These include collet chucks, three-jaw chucks, four-jaw chucks, faceplates, and special faceplate work-holding fixtures. The cutting tools are the same, or similar, to those used on engine lathes. The cutting speed is calculated the same as for engine-lathe work and the feed rates are also selected on the same basis. The possible ways in which these tools can be set up is almost without limit. Single-point cutting tools that are essentially the same as those described in Chapter 7 are used on the cross slide; therefore, they will not be treated in this chapter. Attention will be focused on some of the basic chucking and bar tools that are attached to the turret. These tools are introduced by referring to the appropriate illustration which should be studied along with the descriptions that follow.

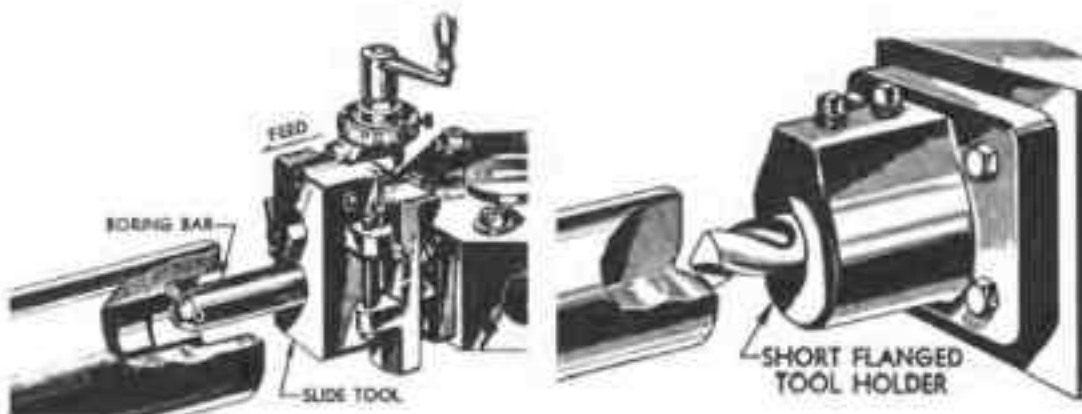
Figure 14-3, Overhead Turning. This operation consists of turning one or more surfaces on the outside diameter of the workpiece with cutting tools that are positioned above the workpiece and are fed by the turret. The cutting tools are held by *cutter holders*, one of which is plain, while the other is adjustable. The cutter holders are held by a *multiple turning head* which is mounted on a face of the turret. Several cutter holders can be mounted on the multiple turning head. In Fig. 14-3, two cuts are taken on the outside diameter, while the end of the part is faced with a tool that is held by a square turret mounted on the cross slide.

Figure 14-4, Multiple Turning and Drilling. The multiple turning head can be used to turn, while simultaneously machining the inside of the hole with a drill, reamer, or boring bar. In the illustration two turning cuts are being taken simultaneously; one from the square turret and the other by tooling mounted on the turret face. The drill shown is a core drill, used



Courtesy of The Warner & Swasey Company

Fig. 14-3. (Left) Overhead turning. Fig. 14-4. (Right) Multiple turning and drilling.



Courtesy of The Warner & Swasey Company

Fig. 14-5. (Left) Boring. Fig. 14-6. (Right) Start drilling.

to enlarge a cored hole in a casting. It has more cutting edges than a twist drill, thereby enabling a faster feed to be used; however, it cannot be used to open a hole from solid metal as the cutting edges do not come to a point.

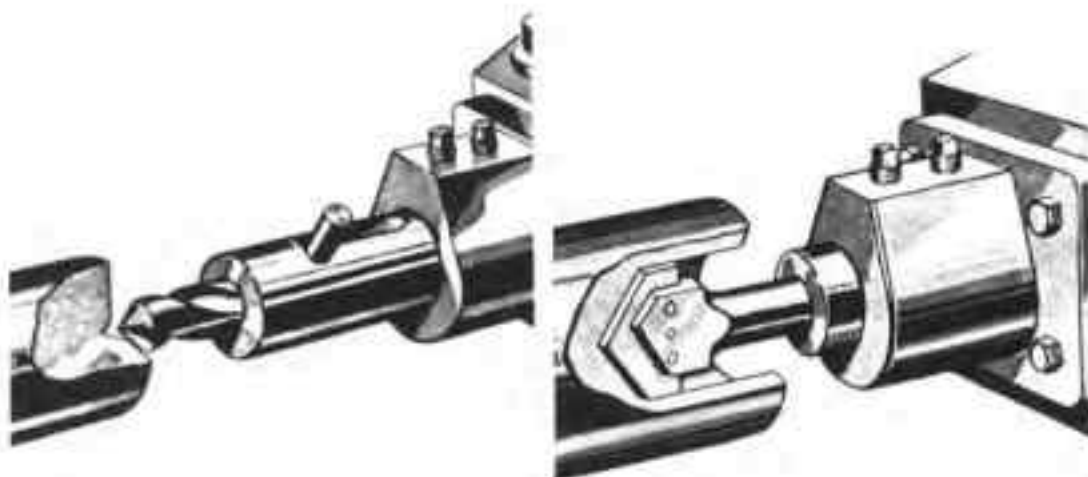
Figure 14-5, Boring. Boring operations are frequently performed on turret lathes. One method is to hold the boring bar in a *slide tool* that is mounted on the face of the turret. The slide tool has a micrometer that is used to adjust the boring tool to obtain the required size of the bore quickly and accurately. Stops are provided on the slide tool so that the boring tool, when it is being withdrawn, can be backed away from the bore to prevent the tool from marking the finished surface. Boring operations can also be performed on turret lathes with the boring tool held by the square turret which is mounted on the cross slide.

Figure 14-6, Start Drilling. The purpose of the start drilling operation is to spot a true cone on the end of the workpiece which will act as a guide for the subsequent drilling operation. The start drill is short and stiff. It will not deflect when starting the hole, and the coned spot will be concentric with the rotation of the lathe. The drill which follows will be guided by this cone and will drill a hole which is also concentric. Although special start drills are frequently used, combination drill and counter-sinks, or center drills, are also used for this purpose.

Figure 14-7, Combination Stock-Stop and Start-Drill. The first operation when doing bar work is generally called stock stop. This operation consists of feeding the bar stock through the spindle until it contacts a stop that is held in the turret. This stop allows just the right length of bar stock to protrude from the face of the collet chuck in order to make the part. The collet chuck is closed when the bar stock comes against the stock stop. The stock stop shown is combined with a start drill which is withdrawn during the stock-stop operation.

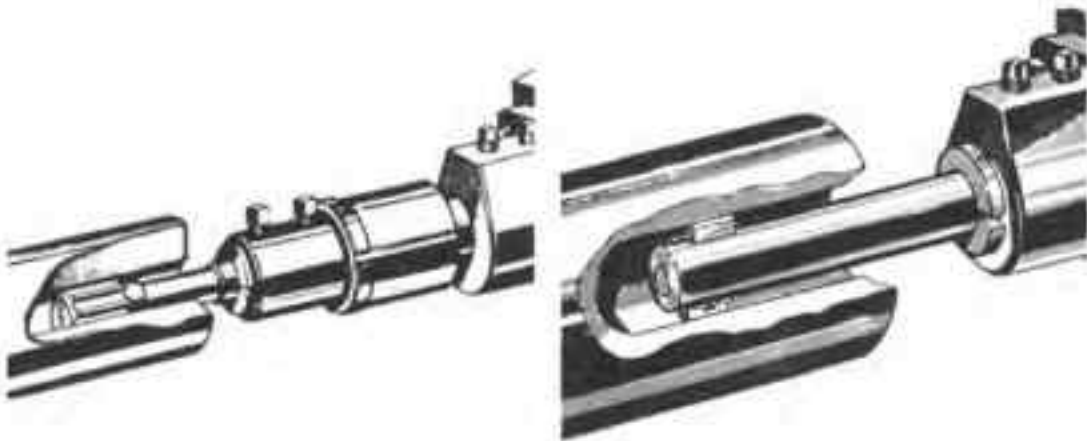
Figure 14-8, Spade Drilling. A spade drill is made from a flat piece of tool steel that is clamped to a bar-like spade-drill holder. The point of the spade drill is similar to the point of a twist drill, providing it with the ability to open holes in solid metal. Spade drills are often used to drill the large holes that are over one inch in diameter. Twist drills of this size are too long and overhang the turret excessively, whereas the overhang of spade drills is at a minimum. The blade of the spade drill can be replaced, when required, without the necessity of purchasing a new spade-drill holder. The setup for drilling a hole with a twist drill is similar to Fig. 14-8, and therefore will not be illustrated.

Figure 14-9, Reaming. Holes are finished by reaming from the turret, as shown. The reamer should be held in a *floating reamer holder* to assure



Courtesy of The Warner & Swasey Company

Fig. 14-7. (Left) Combination stock-stop and start-drill. Fig. 14-8. (Right) Spade drilling.



Courtesy of The Warner & Swasey Company

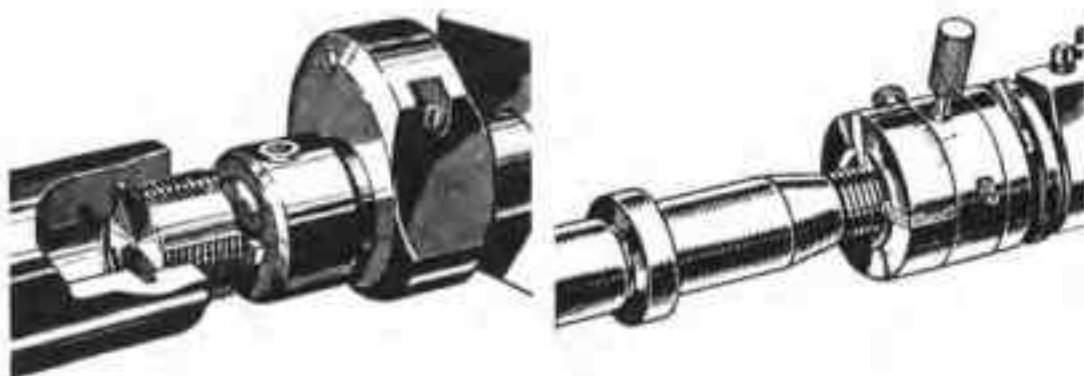
Fig. 14-9. (Left) Reaming. Fig. 14-10. (Right) Floating blade reamer.

proper alignment of the reamer with the drilled hole, thereby assuring that the reamed hole will be straight—not tapered.

Figure 14-10, Floating-Blade Reamers. Larger holes are often reamed with a floating-blade reamer. The two cutting blades of this reamer are allowed to float a certain amount, which provides the necessary alignment to obtain a straight hole.

Figure 14-11, Tapping. Tapping can be done on a turret lathe to produce internal threads. The tap must be held in a *releasing tap holder*. To tap the hole the turret is advanced toward the rotating workpiece, using sufficient force at the beginning in order to make the tap start to cut the threads. After the threads have been started the rotation of the workpiece and the lead of the threads will combine to pull the tap into the hole. As the tap feeds through the hole the turnstile is turned by hand just enough to keep the turret and the tap holder moving at the same rate as the tap. When the forward progress of the turret is stopped, the tap holder will release the tap, allowing it to revolve with the workpiece. The tap is withdrawn by reversing the direction of the spindle rotation.

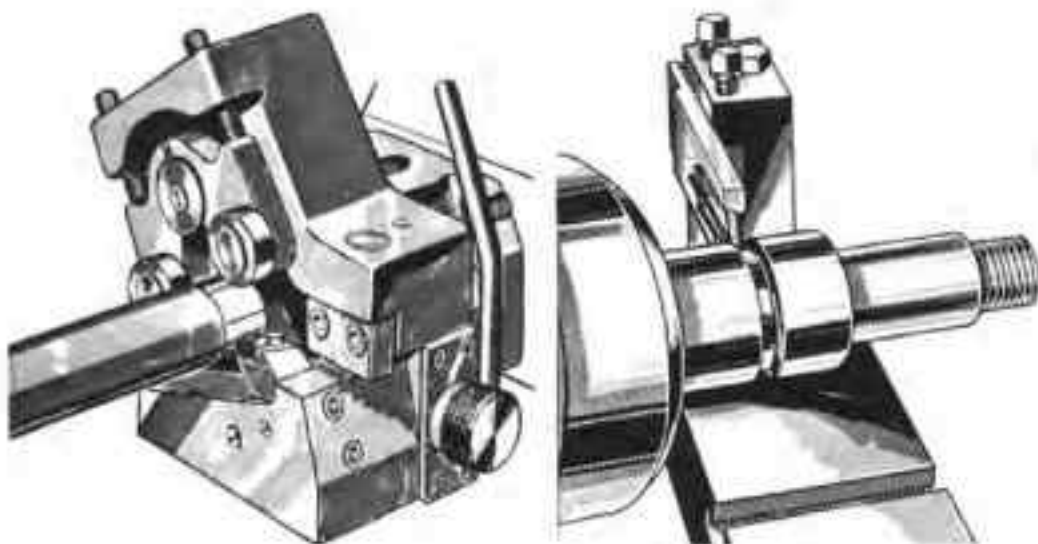
Figure 14-12, External Threading. External threads are generally cut on turret lathes with self-opening die heads. As with tapping, the die head must be started to cut the threads. After the die has started, a light feeding force is used on the turnstile in order to keep the turret moving with the die head. When the forward progress of the turret is stopped the die head will snap open, thereby clearing the dies from the threads that have been cut. After the die head has been backed away from the threads it is closed by a movement of the handle in preparation for cutting the next thread. Die heads are available for cutting a wide range of thread sizes.



Courtesy of The Warner & Swasey Company

Fig. 14-11. (Left) Tapping. Fig. 14-12. (Right) External threading.

Figure 14-13. Roller-Bar Turners. Roller-bar turners are used to turn long bar work which overhangs the collet. The rollers support the bar stock to keep it from deflecting. The rollers may be mounted ahead of the cutting tool in order to obtain concentricity between the previously turned diameter and the diameter being turned. The rollers are set behind the cutting tool, as shown, when turning the first diameter on rough-bar stock. When the rollers are set behind the cutting tool they burnish the turned surface thereby improving its appearance. At the end of the cut the cutting tool is withdrawn from the work by moving the cutter relieving lever. This prevents the cutting tool from marking the turned surface as the roller turner head is backed off the work by the turret.



Courtesy of The Warner & Swasey Company

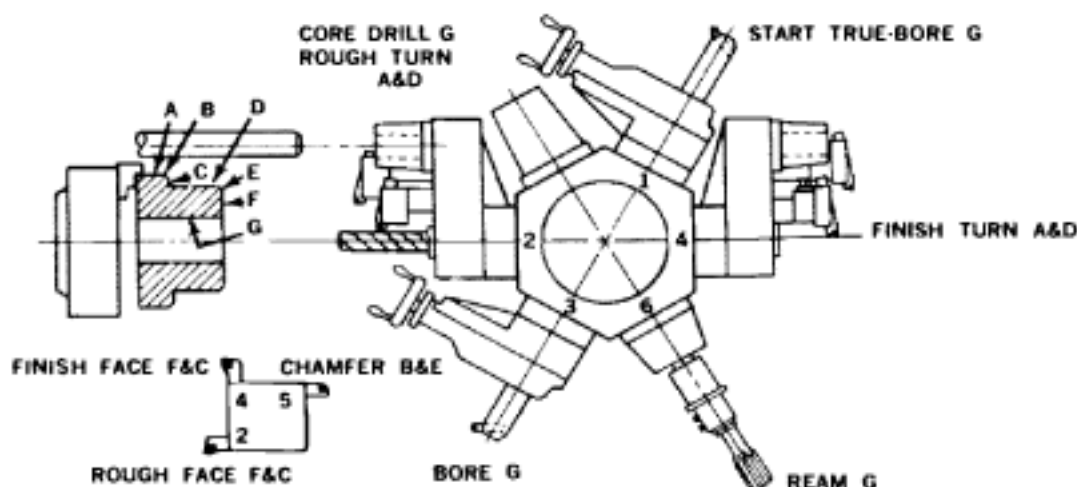
Fig. 14-13. (Left) Roller bar turner. Fig. 14-14. (Right) Cut-off operation.

Figure 14-15, Cutting Off. After a bar part has been completely machined it is cut off with a cut-off tool. A convenient place to mount this tool is in the rear cutter-block. When held by the rear cutter-block the cut-off tool must be held in an upside-down position. When the space is available, the cut-off tool can also be held by the square turret.

Most work done on a turret lathe is planned in advance. Often, tooling layouts are made which give the operator, or the set-up man, a pictorial representation of the way that the machine is to be set up. Figure 14-15 illustrates the tooling layout for machining a flange. The following sequence of operations is used to finish the part:

1. Start to true-bore hole *G*, deep enough to start core drill true.
2. Core drill hole *G* and rough-turn diameters *A* and *D* from turret. At the same time, rough-face *F* and *C* from square turret. Face *F* first and *C* last.
3. Bore hole *G*, to make sure that it is concentric.
4. Ream hole *G* to size.
5. Finish-turn diameters *A* and *D*. At the same time, finish faces *F* and *C*.
6. Chamfer *B* and *E*.

Figure 14-16 shows a typical ram type turret-lathe setup for doing bar work. A saddle type turret lathe equipped with a cross-sliding turret is shown machining steel gate-valve bodies in Fig. 14-17. The gate valve is held in a special indexing fixture that permits three ends of the valve body to be machined in a single chucking. The versatility of the turret lathe for doing a wide variety of turning jobs is demonstrated by these illustrations.



Courtesy of The Warner & Swasey Company

Fig. 14-15. Tooling layout for machining a flange.



Courtesy of The Warner & Swasey Company

Fig. 14-16, Ram type turret lathe setup for bar work.

Chucking and Bar Automatics

Chucking and bar automatics are high production, yet rather versatile lathes, inasmuch as they can readily be set up to machine different parts. The controls on these machines are set manually. However, in operation they are automatic. These lathes may be equipped with numerical control. Some may be changed over to do either chucking or bar work and some are equipped to turn between centers. A characteristic of chucking and bar automatics is the pentagon turret, which on some lathes is above the spindle and on others is behind the spindle. A single spindle chucking automatic is shown in Fig. 14-18. This machine has an overhead pentagon turret and two slides. The turret is an integral part of a turret bar which indexes the turret and provides the motion parallel to the spindle axis. A variety of standard tooling blocks and tool holders can be mounted on each of the five turret positions to perform turning, drilling, reaming, boring, and thread cutting operations. The two slides move perpendicular to

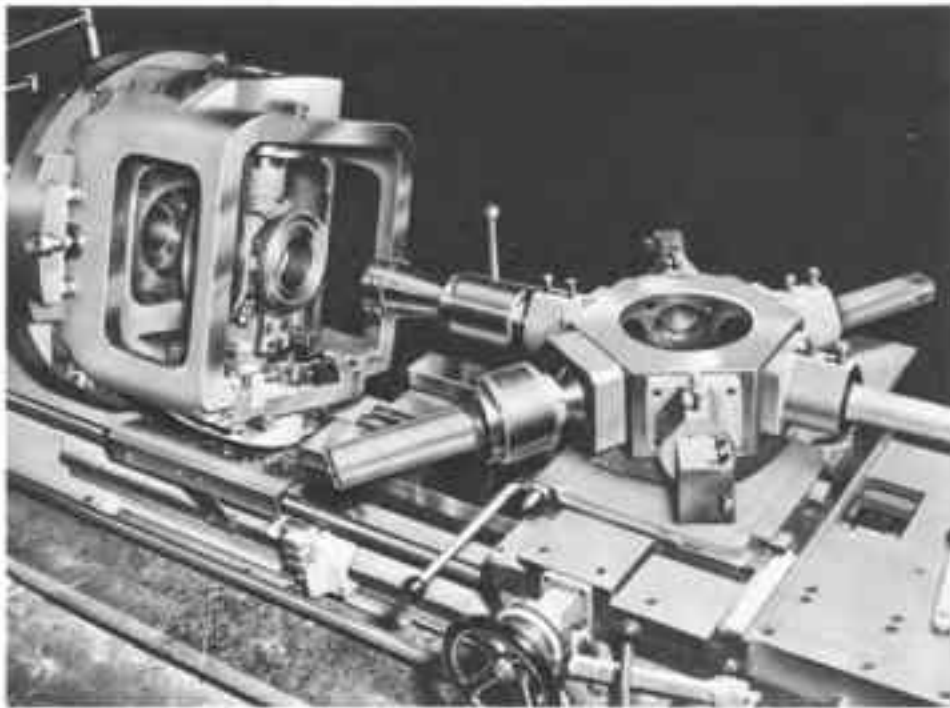


Fig. 14-17. Machining steel gate-valve bodies held in an indexing fixture on a saddle type turret lathe having a cross-sliding turret.



Courtesy of the Turning Machine Div. of Warner & Swasey

Fig. 14-18. A single-spindle automatic chucking machine.

the spindle axis to actuate cutting tools that perform grooving, cut-off, facing and forming operations. All of the tooling is within easy reach of the operator, as can be seen in Fig. 4-19. The arrangement of the tools on the machines is such that there is a minimum of interference between the tools, making it possible to take simultaneous cuts with tools that are attached to the turret and to the slides. The tooling blocks that are mounted on the turret have provisions that allow several tools to cut simultaneously;



Courtesy of the Turning Machine Div. of Warner & Swasey

Fig. 4-19. Changing a tool in an automatic chucker.

e.g., it is possible for several turning tools to be cutting different diameters while at the same time drilling, reaming, or boring a hole. A typical operation on a chucking automatic is illustrated in Fig. 14-20, where the large steel castings shown in the foreground are being machined.

A single spindle bar automatic is shown in Fig. 14-21. This machine has three slides, the additional slide being used to actuate a cut-off tool. Bar stock from which the part is made is contained inside of the spindle and held by a collet chuck. In operation the stock is fed out against a stock stop and clamped by a collet-type chuck. The protruding portion of the bar is machined to the part configuration and then it is cut off, after which the finished part falls into a chute.

Much of the tooling on these machines, such as chucks, tooling blocks, and toolholders, is standard tooling that can be obtained from the



Courtesy of the Turning Machine Div. of Warner & Swasey

Fig. 14-20. Machining heavy castings in an automatic chucker.



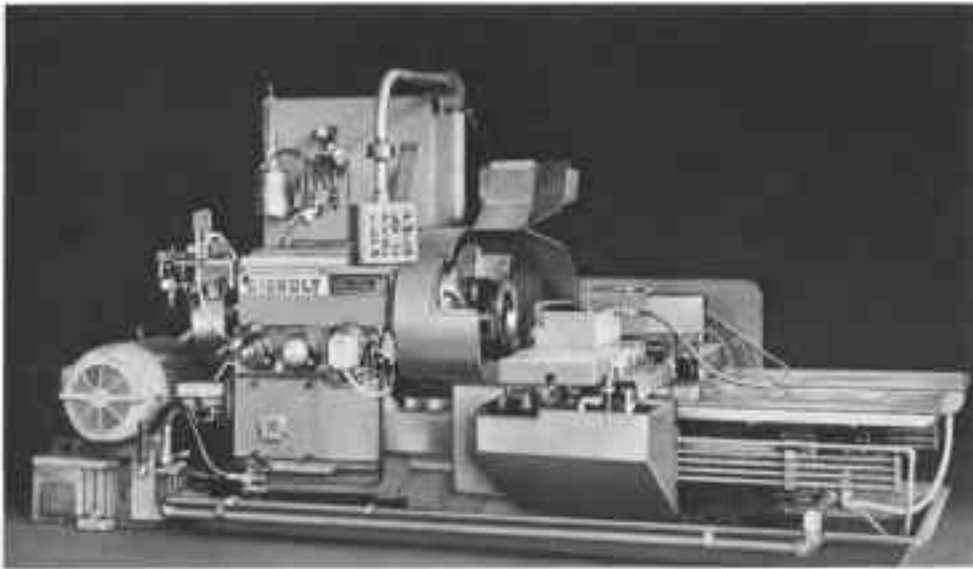
Courtesy of the Turning Machine Div. of Warner & Swasey

Fig. 14-21. A single-spindle automatic bar machine.

builder. Rather frequently the standard three-jaw universal chucks require special jaws to grip the workpiece. For some work special work-holding fixtures must be designed and built.

Production Lathes

A typical automatic production lathe is shown in Fig. 14-22. These machines are used for high-volume repetitive work such as is encountered in the automotive industry. The tooling used on these machines is specially designed to do one particular job with great efficiency. A typical setup which is used to machine front and rear automotive brake drums is shown in Fig. 14-23. A special back-facing attachment is located inside the work-holding fixture for facing the mounting face of the brake drum. The front tool-slide turns and bores the other end of the drum. All turning and boring operations are completed on this machine in one setup. No great amount of skill is required to operate an automatic production lathe; however, the planning of the operations, the layout of the tooling,



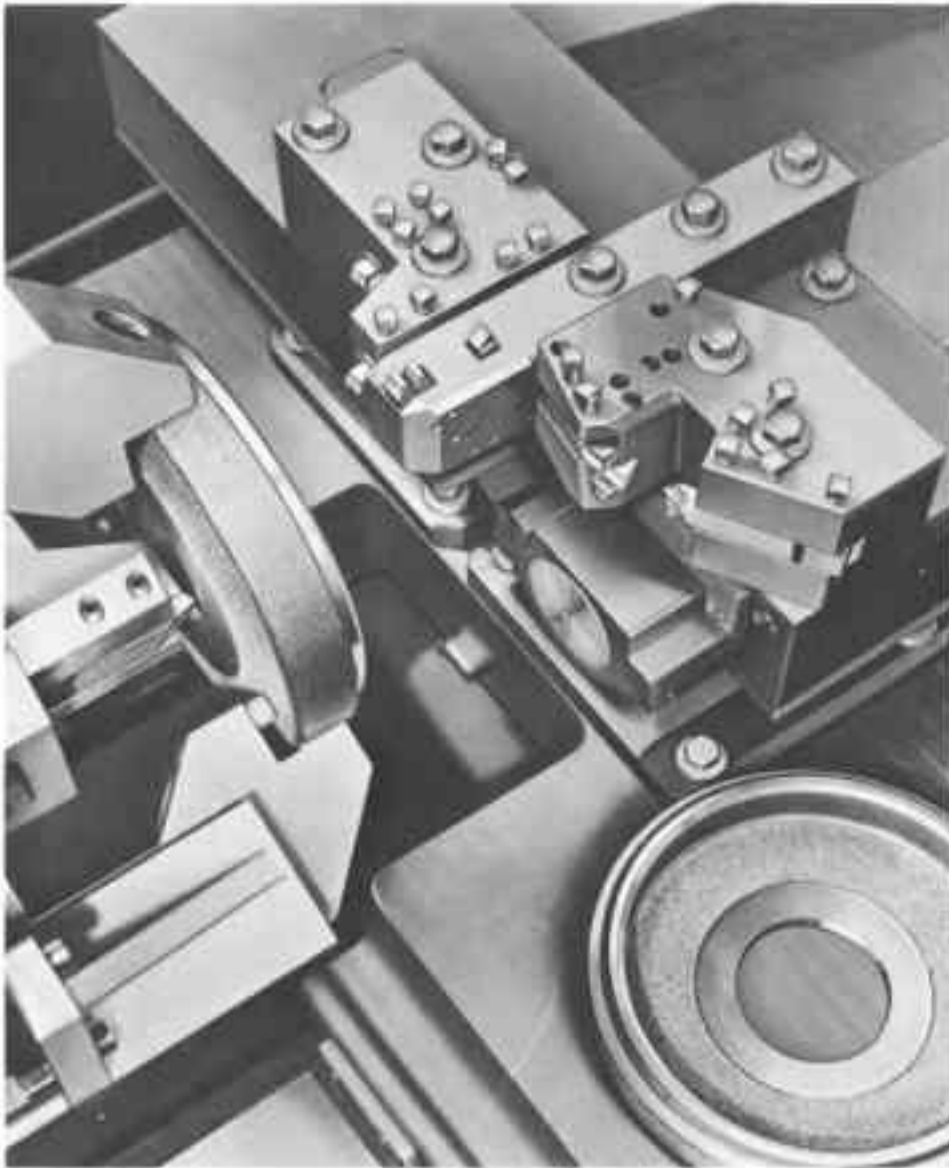
Courtesy of the Gisholt Machine Company

Fig. 14-22. Automatic production lathe.

and the design of the tooling do require skill and experience of the highest order.

Vertical Lathes

The vertical lathe, commonly referred to as a *vertical boring mill*, has a circular table which revolves about a vertical axis so that the work-holding surface is horizontal, thus making it comparatively easy to place and hold large circular parts in position (See Fig. 14-24.) The two large, vertical housings are connected together at the top by a bridge, to provide a rigid support. Ways are machined on the face of the housing on which the cross rail is mounted. The cross rail can be adjusted to any desired height above the top of the circular table. It is not, however, set in motion to take a cut. Two saddles, mounted on the cross rail, can be moved along the length of the cross rail. Tool bars, which have a vertical movement, are mounted on each saddle. They can also be swiveled on the saddle to provide an angular movement. Tool blocks for holding cutting tools are located at the ends of the tool bar. The vertical orientation of the tool bars allows them to be made small enough to do boring operations without deflecting due to their own weight. Many vertical boring mills are equipped with a pentagon turret, in which case they are usually called *vertical turret lathes*. A vertical turret lathe is shown in Fig. 14-25 machining a large fan exit-case of a jet engine. The pentagon turret provides five tooling stations on each of which one or more cutting tools can be mounted. Both vertical boring mills and vertical turret lathes can be equipped with a side head which can be seen in Fig. 14-25. The side head can move in a vertical direction by sliding over the ways of the right-hand column. The side-head arm moves in a horizontal direction.



Courtesy of the Gisholt Machine Company

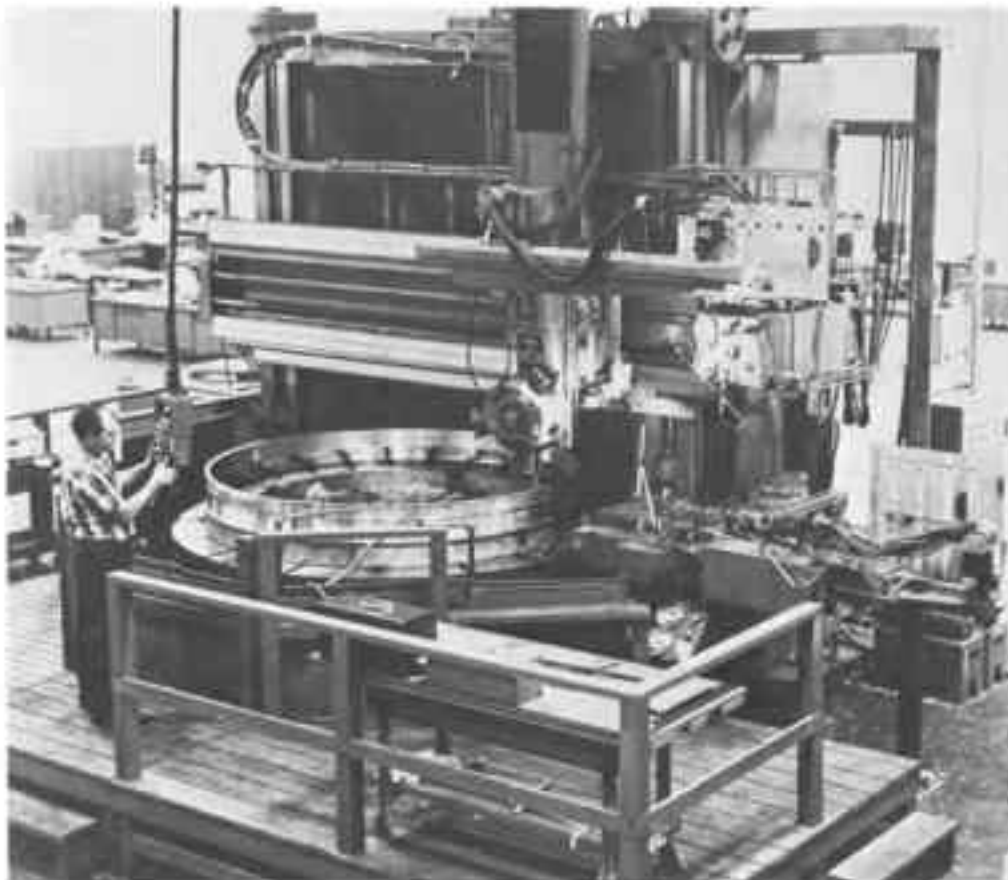
Fig. 14-23. Automatic production lathe tooled to produce automotive brake drums.

The cutting tools mounted on the side head can, therefore, move parallel and perpendicular to the surface of the faceplate.

Workpieces can be bolted directly to the faceplate or they can be held by chuck jaws. The chuck jaws operate independently of each other in a manner similar to four-jaw chucks. Figure 14-26 shows a cable hoist drum being machined on a vertical turret lathe. The workpiece is held by four-chuck jaws while simultaneous cuts are taken by tools held by the pentagon turret and the side head. Special work-holding fixtures, such as shown in Fig. 14-27, are sometimes used to hold the work. This workpiece is very thin and is apt to distort and vibrate when being cut.



Fig. 14-24. Vertical boring mill.



Courtesy of The Bullard Company

Fig. 14-25. Vertical turret lathe machining jet-engine part.

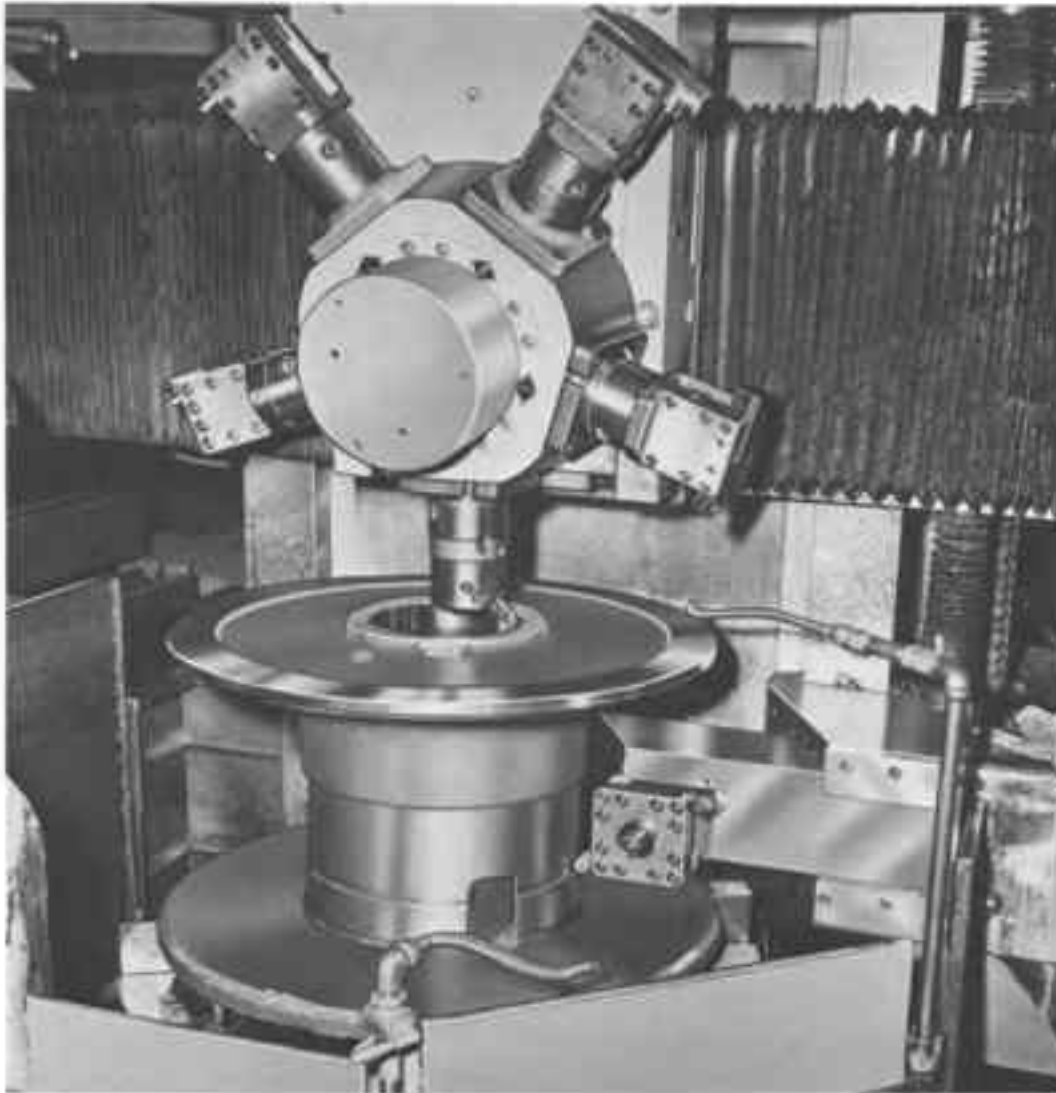


Fig. 14-26. Cable hoist drum being machined on a vertical turret lathe.

The fixture must also hold the part without distorting it. The part is firmly clamped to the base of the fixture. The snubbers at the top help to support the work against distortion due to the cutting forces, and they help to prevent vibration or chatter from occurring.

Vertical lathes are used to hold workpieces that can be more conveniently clamped to a horizontal surface rather than to a vertical surface. They are used to machine surfaces that require the use of a long, extended bar to hold a cutting tool that cannot be supported at two ends. The advantage of the vertical lathe in such situations is that the long bar—say, a boring bar—does not deflect under its own weight when it is hung in a vertical position. The vertical lathe is also often used in preference to other machine tools when large, internal, facing cuts are required; when very large diameter holes are to be bored; and when both large-diameter



Courtesy of The Bullard Company

Fig. 14-27. Special work-holding fixture and tooling blocks on vertical turret lathe.

boring and facing cuts are required on the same part. Of course, cylindrical turning and facing cuts on outside surfaces can also be conveniently taken on these machines. Many vertical lathes are equipped with numerical controls or with tracer controls which add to their versatility. The vertical turret-lathe in Fig. 14-25 is equipped with tracer controls on the turret and on the side heads, enabling it to cut angles and complex contours. The machine in Fig. 14-26 is equipped with a numerical-control system.

Precision Hole Location — The Jig Borer

The problem of precision hole location is encountered by all craftsmen who are engaged in precision machine-tool work. Many holes in precision machine and engine parts, jigs, fixtures, and metal stamping dies must be machined to very close tolerances with respect to their diameters and their location on the workpiece. The hole-locating methods previously treated in this book satisfy the accuracy requirements for most ordinary holes; however, when the location of a hole must be very precise, other methods must be used.

Precision hole-location methods have been developed over the years by skilled craftsmen such as clockmakers, instrument makers, toolmakers, and machinists. Some of these methods are still used upon occasion and will be described later, in this chapter. An engineering solution to the precision hole-locating problem was the development of the jig borer which occurred at the end of World War I, independently, both in the United States and Switzerland. This machine tool is designed to machine holes that are precise, with respect to diameter and location. A bank of jig borers at work is shown in Fig. 15-1. Just prior to World War II, a further step was taken by The Moore Special Tool Company, when they developed the jig-grinding machine which is used to locate and grind holes in hardened metals to extremely close tolerances. More recently, coordinate measuring machines have been developed to accurately measure the location of holes.

Coordinate Dimensions

The coordinate system of dimensioning is used on drawings for jig-borer work. This system of dimensioning is designed to be compatible with the coordinate axes of movement available on the jig borer. Other machines, particularly numerically controlled machines, are also designed to have table movement that is compatible with the coordinate axes of the coordinate dimensioning system. These dimensions are normally placed on the drawing by the draftsman; however, the machinist or the toolmaker is occasionally required to do this at the machine. It is worthwhile to learn the principles involved.

In the coordinate dimension system all dimensions are given from two reference axes that are perpendicular to each other. The position of these axes sometimes corresponds to the side of the workpiece, as at B, Fig.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-1. Jig borers and jig grinders in a temperature-controlled toolroom.

15-2; although this is not always the case. At A, Fig. 15-2, the dimensions are drawn in the conventional manner, while at B, they are drawn using the coordinate dimension system. Having the top edge and the left edge, or the upper left-hand corner, act as a reference surface, agrees with the positions of the scales on many jig borers, although there are some differences in this respect. The dimensions for the coordinate system for parts to be jig bored is usually given to .0001 inch. Tolerances are usually not specified, for it is understood that the jig-borer operator will locate each hole to the limit of accuracy of the machine. This is the accepted practice for tool and die work. The dimensions can be transferred from the conventional system of dimensioning to the coordinate system by simply adding or subtracting, as shown in Fig. 15-2. A discussion of how tolerances are affected when transferring from the conventional to the coordinate system of dimensioning is given at the end of this chapter.

Boring

Because of its importance in precision hole location, a review of the boring process is helpful at this time. In Fig. 15-3, the boring tool shown is rotating about the axis of the spindle of a jig-boring machine. The point of the cutting tool is always the same distance from the axis of rotation of the spindle; therefore, the cylindrical surface of the hole that is generated must be concentric with the axis of the spindle. For this reason, single-point boring is the most accurate method of finishing holes in metals that can be cut with single-point cutting tools.

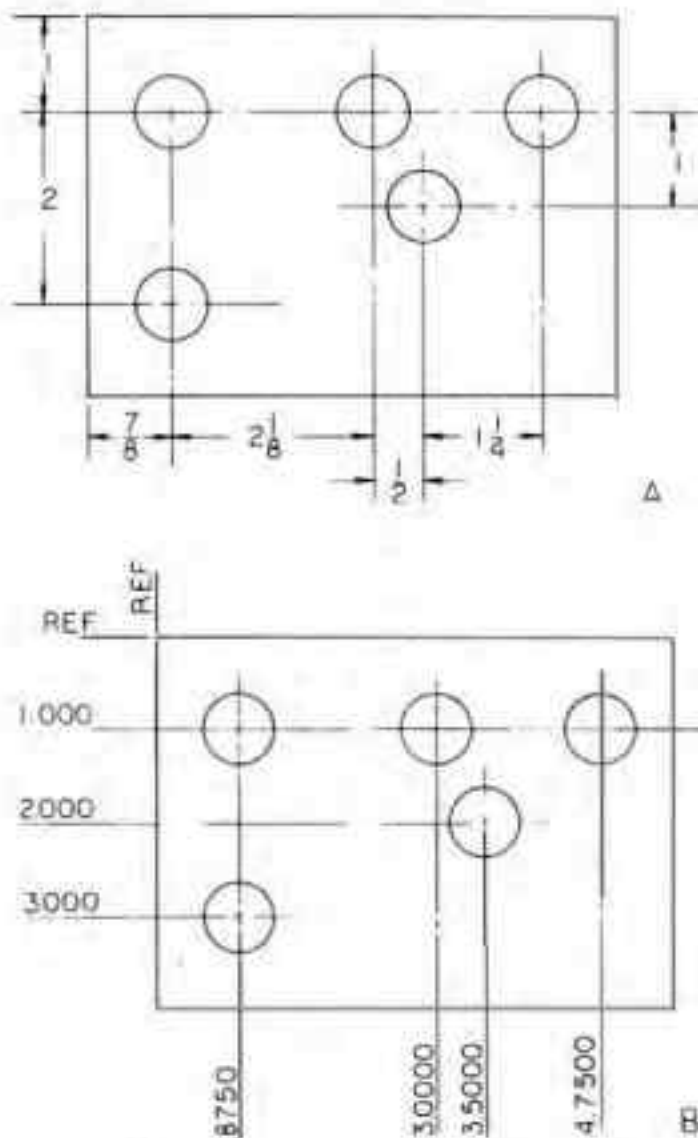


Fig. 15-2. (Top) A. Conventional dimensions. (Bottom) B. Coordinate dimensions.

The boring tool shown in Fig. 15-3 is held in an offset boring-head which can be very accurately adjusted, radially toward or away from the center of rotation, in order to bore holes of different diameters. With normal care it is not difficult to bore a hole to a tolerance in the order of "tenths," (.0001 in. or 0.002 mm) with a precision boring-head.

If the hole that was originally drilled is not concentric with the axis of the spindle, the boring tool will take a heavier cut on one side of the hole than on the other side. The unequal cutting force created by this cut will deflect the bar more on the side where the heavier cut is taken, thereby causing the bored hole to be eccentric with respect to the axis of the spindle. This effect can be eliminated by grinding the boring tool correctly, by using a sharp tool, and by taking light cuts when finishing the hole to size.

In the situation just described, the boring tool was rotating. The same



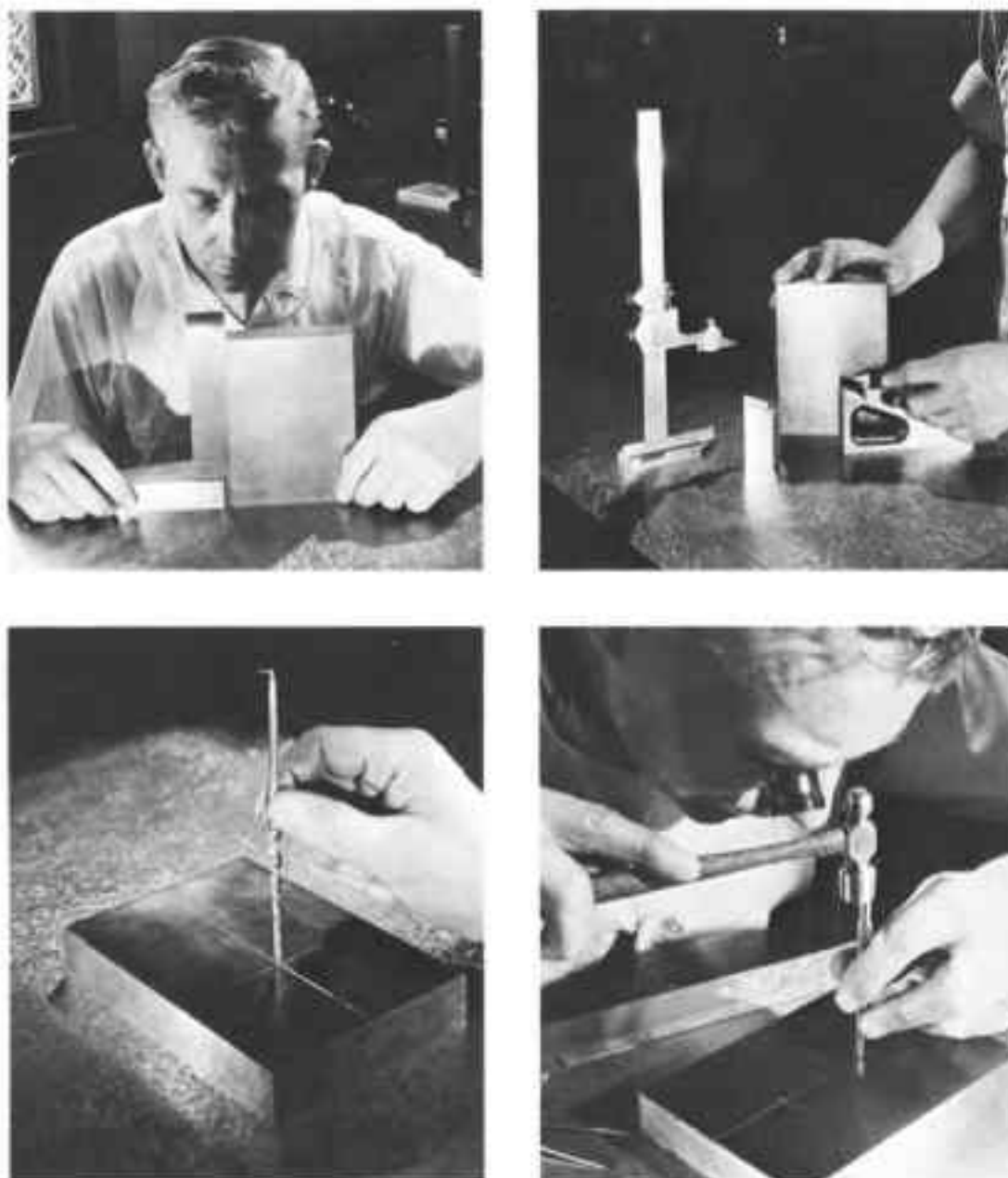
Courtesy of The Moore Special Tool Company

Fig. 15-3. Single-point boring—the most precise hole-locating method. The surface is generated by rotation about the hole axis. Either the workpiece or tool may rotate.

end-result will also occur when the work is rotating, as in a lathe. The point of the cutting tool will always be at the same distance from the axis of rotation of the workpiece, which is determined by the axis of rotation of the lathe spindle. The cylindrical surface that is generated as the tool feeds through the bore must then be concentric with the axis of rotation of the spindle.

Precision Hole Location

The shapes and sizes of the workpiece in which precision holes are located are almost infinite. Rectangular-shape plates, such as shown in Fig. 15-4, are frequently encountered in tool and die work. Because of the availability and the ease with which this shape can be worked, it is used to construct components for stamping dies, jigs, fixtures, and gages. This shape, therefore, is used to illustrate and explain the methods of precision hole location. Four proven and practical methods of hole location will be explained in the following pages; in addition, two methods of locating holes in hardened workpieces will be discussed. It should be emphasized that the use of the jig borer, when available, is not only the preferred, but the superior method of precision hole location.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-4. (Upper left) A. Checking the squareness of the sides prior to laying out. At least two sides must be square to each other and to the top and bottom faces. (Upper right) B. Scribing two perpendicular sides to locate the hole, using the squared sides as reference surfaces. (Lower left) C. Locating the center of the hole by twirling the hand prick-punch at the line intersections. (Lower right) D. Carefully prick punching the hole center.

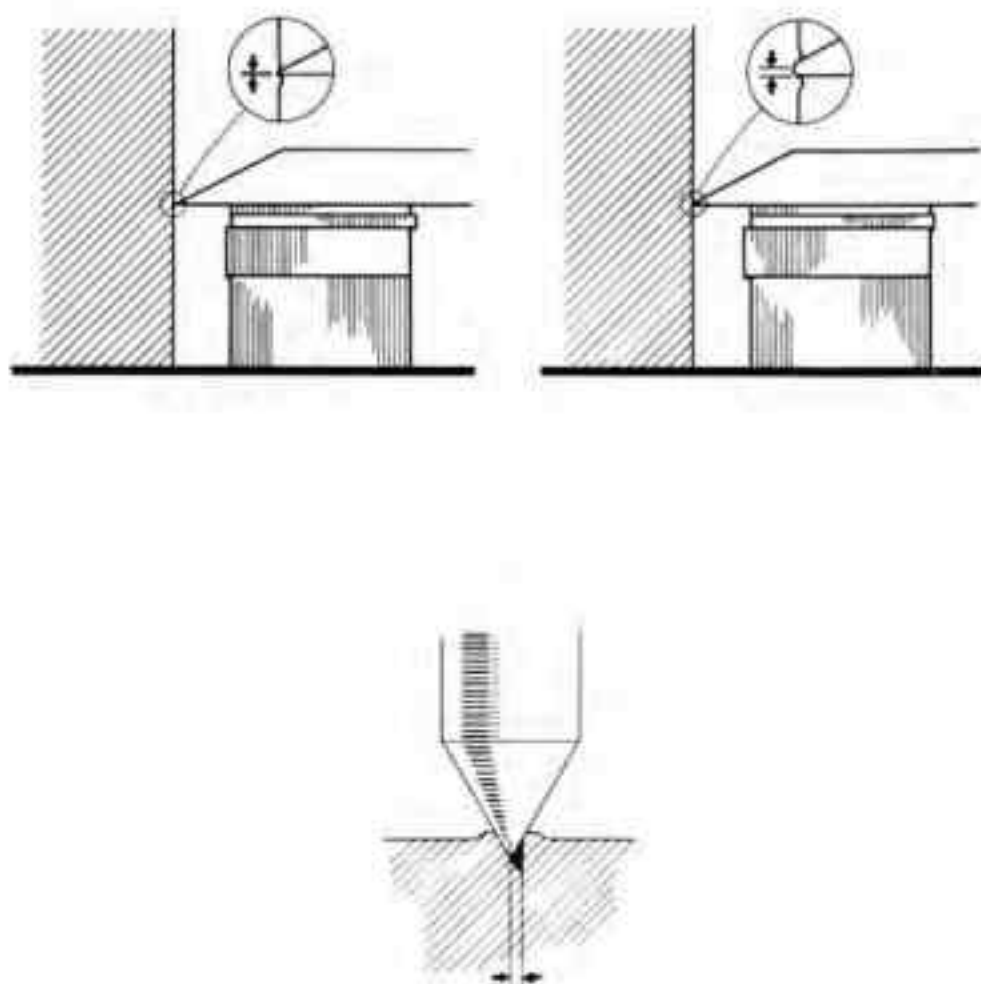
Precision Layout and Drilling Procedure

Prepare the workpiece. The workpiece is machined so that the two faces are parallel to each other and "square" with respect to all sides, or edges, as they are often called in the toolroom. Two of the edges are

machined to be exactly square with each other; these two edges will serve as reference surfaces. Sometimes it is convenient to machine all four edges square to each other. The machining can be done on a milling machine, a shaper, or on a surface grinder. It is often more convenient to rough machine on a milling machine or on a shaper, and then to finish grind the part on a surface grinder. A metal-cutting band saw will sometimes be best for rough machining this part.

Make a precision layout. The face of the workpiece is painted with a blue layout dye and the lines are scribed with precision layout tools on the required locations. The precision layout tools are shown at B, Fig. 15-4. Shown are a vernier height-gage, precision gage-blocks, and a universal precision gage, or "planer" gage, as it is also called. The vernier height-gage has a scribe attached to the movable arm which can be set at any desired distance above the face of the surface plate. With care, layout lines can be scribed to an accuracy of .001 inch with the vernier height-gage. The slider of the planer gage can be accurately positioned anywhere along the incline. The top of the slider is perfectly parallel to its base and serves as a reference surface. It is positioned to the desired height above the surface plate by measuring across the top of the slider and the base with a micrometer caliper. A more accurate method is to transfer the height of a stack of precision gage-blocks to the top of the slider. This is done by holding a dial test indicator in a fixed vertical position on an indicator holder or on the movable arm of the height gage. The indicator is positioned over the gage blocks and set to read zero. It is then positioned over the planer gage which is adjusted until the indicator again reads zero. A wedge-shape scribe, which has had its bottom surface lapped perfectly flat, is placed on the planer gage to scribe the layout lines. The wedge-shape scribe can also be placed directly on the stack of gage blocks to scribe the line if an accuracy to "tenths" is required and a planer gage is not available. This method, however, is hard on the gage blocks and should be used sparingly. Lines are thus scribed by whatever precision method is most suitable. The coordinate dimension layout for the holes is obtained by resting one of the two squared reference edges on a precision surface plate and scribing all of the lines that are parallel to this edge. The second reference edge is then placed against the surface plate and all of the layout lines parallel to this edge are scribed. The scribed lines should be kept as shallow as possible.

Prick-punch marks are carefully made at the intersection of the layout lines where the holes are desired. When an ordinary prick punch is placed in a line that has been scribed by a wedge type scribe a small error will result, as shown in Fig. 15-5. In order to minimize this error, the prick-punch mark should be started with a hand prick-punch as shown at C, Fig. 15-4. A hand prick-punch has a more acute point than a regular prick punch. The punch is made from a piece of high-carbon tool steel that is twisted as shown, and hardened at the point. The hand prick-punch is carefully placed at the intersection of the layout lines, using the sense of touch to feel the intersection with the point of the punch. When the

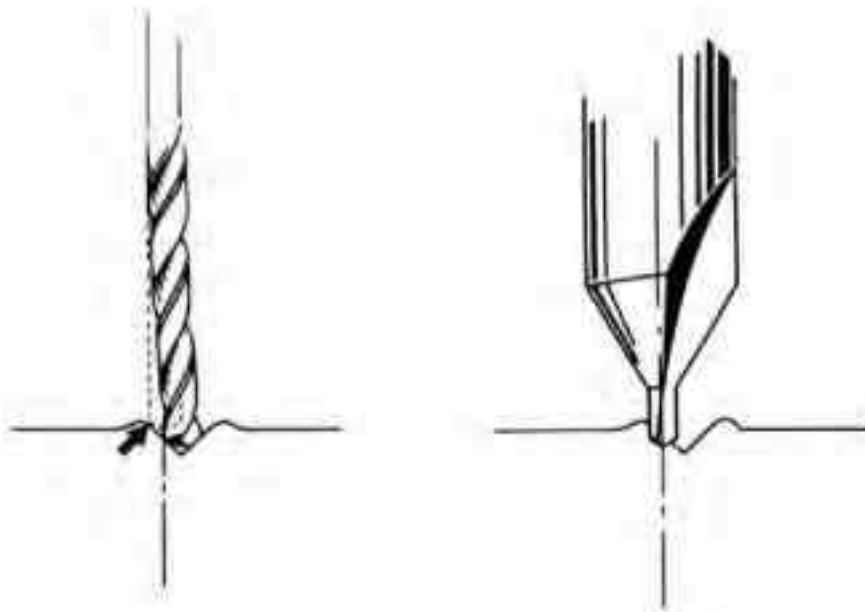


Courtesy of The Moore Special Tool Company

Fig. 15-5. Error introduced by shape of scribed line. This error is reduced by keeping the scribed line shallow and by using a rotating, hand prick-punch with an acute-point angle prior to using a regular prick-punch.

hand prick punch is in position, it is rotated by twirling back and forth between the thumb and fingers until a small mark is made. The prick punch mark is enlarged with a standard prick-punch and hammer as shown at D, Fig. 15-4. This must be done carefully, using a magnifying glass to be sure that the original position is not disturbed. Also, the prick punch must be accurately ground to have a point in the center of the punch. The hammer blow should be light, making a prick-punch that is approximately $\frac{1}{64}$ inch in diameter. A circle may now be scribed with dividers although this is not always necessary, especially with smaller holes. The circle is not used to locate the hole. When a circle is scribed on precision layouts, the witness marks are usually omitted. On die and tool work they may mar a surface that must be finished smooth or they may even form the nucleus of a crack if the part is to be heat treated.

The layout is now completed. When done with skill and care, accurate results can be expected.

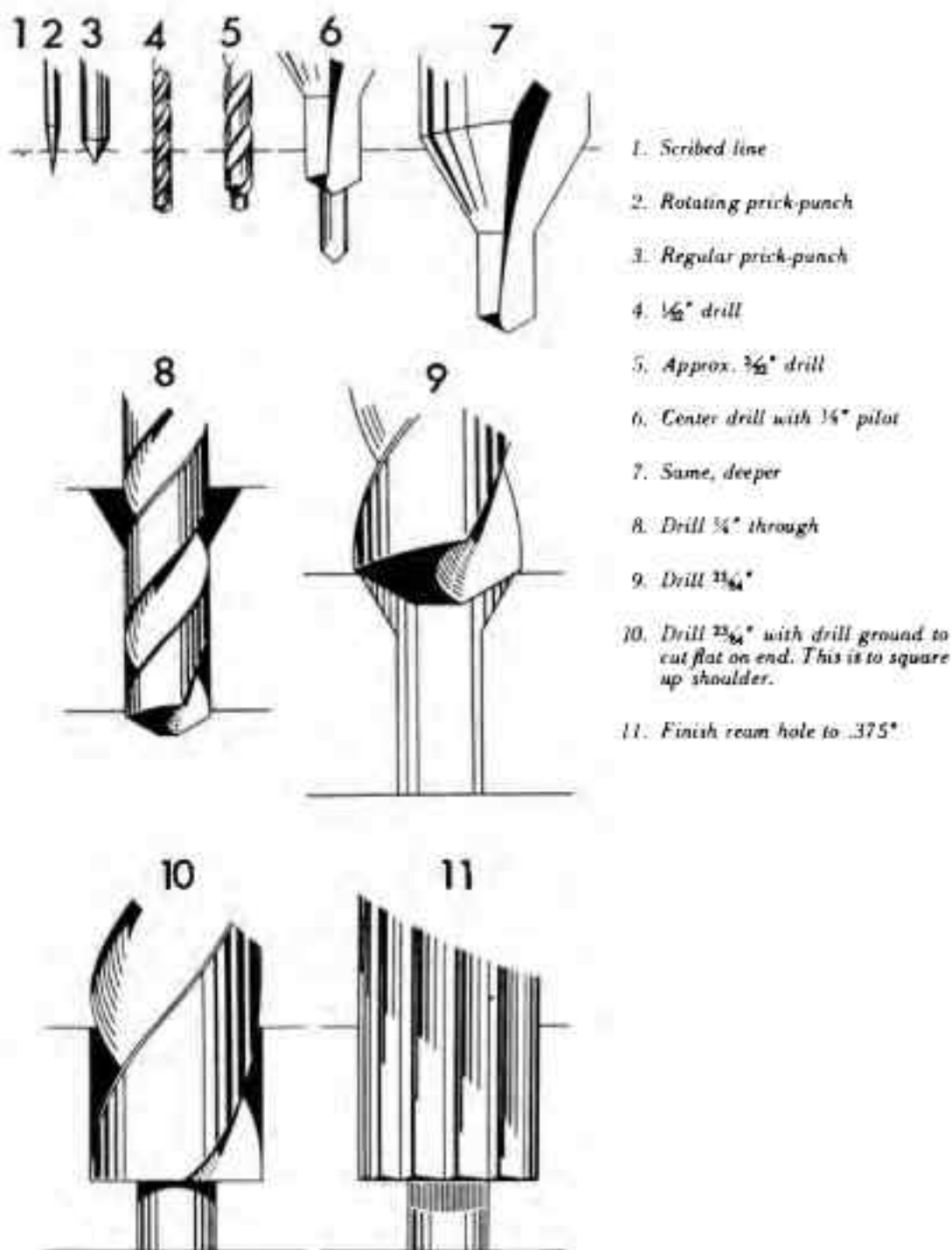


Courtesy of The Moore Special Tool Company

Fig. 15-5. Starting the hole with a small slender drill. Slender drill will bend and follow prick-punch mark. Center drill will cut into the side of prick-punch mark instead of following it.

Drill and counterbore the hole from the layout. A large drill should never be started in a small prick-punch mark. The prick-punch mark will be quickly obliterated and the drill will drift into another location. Also, a center drill should not be used. The center drill is rigid and a slight misalignment between the center drill and the prick-punch mark will cause the drill to start the hole at the side of the prick-punch mark, as in Fig. 15-6, with a consequent error in the hole location. The best start is made when the prick punch entirely surrounds the drill. An ordinary twist drill, having a diameter of about $\frac{1}{64}$ or $\frac{1}{32}$ inch, should be used to start the hole. The position of the hole is determined by the prick-punch mark and any misalignment of the drill will be compensated for by the flexibility of the drill, as shown in Fig. 15-6. The slender twist drill will bend and center itself in the prick-punch mark, whereas the rigid center drill will not. Since the slender starting drill will bend, it will not drill a vertical hole, and it should not be permitted to go too deep. The entire sequence of laying out and drilling the hole is shown in Fig. 15-7.

The starting drill should be followed by another, slightly larger and stiffer drill, say $\frac{3}{32}$ diameter, which is followed by a center drill having a $\frac{1}{8}$ -inch pilot. The center drill is drilled to the depth as shown at 7, Fig. 15-7. A $\frac{1}{4}$ -inch drill is then used to drill through the part. The hole is further enlarged with a $\frac{23}{64}$ -inch drill which is followed by a $\frac{25}{64}$ -inch drill that has been ground to cut flat on the end in order to square the shoulder of the counterbore. This operation is omitted if the hole is a through hole. The final operation is to finish ream the hole to size.



Courtesy of The Moore Special Tool Company

Fig. 15-7. Procedure for drilling hole from layout.

Precision Layout, Drilling, and Boring Procedure

1. Prepare the workpiece.
2. Make a precision layout. (Steps 1 and 2 are done as previously described and will, therefore, not be repeated.)

3. Mount the workpiece on a lathe faceplate. The workpiece is clamped on the faceplate, Fig. 15-8, in preparation for drilling and boring. On parts of this size it is usually most convenient to remove the faceplate to mount the work as shown in Fig. 15-37. A sheet of paper placed between the faceplate and the workpiece serves to reduce the tendency of the part to move, particularly when it is subjected to large cutting forces. Soft metal shims placed between the strap clamps and the work will prevent the clamps from marring the surface of the work.
4. Align the workpiece in the lathe. The point of a pump center, Fig. 15-9, is placed in the prick-punch mark corresponding to the hole to be drilled, as shown in Fig. 15-8. The other end is placed



Courtesy of The Moore Special Tool Company

Fig. 15-8. Aligning the work with the axis of rotation of the lathe using a pump center and an indicator.

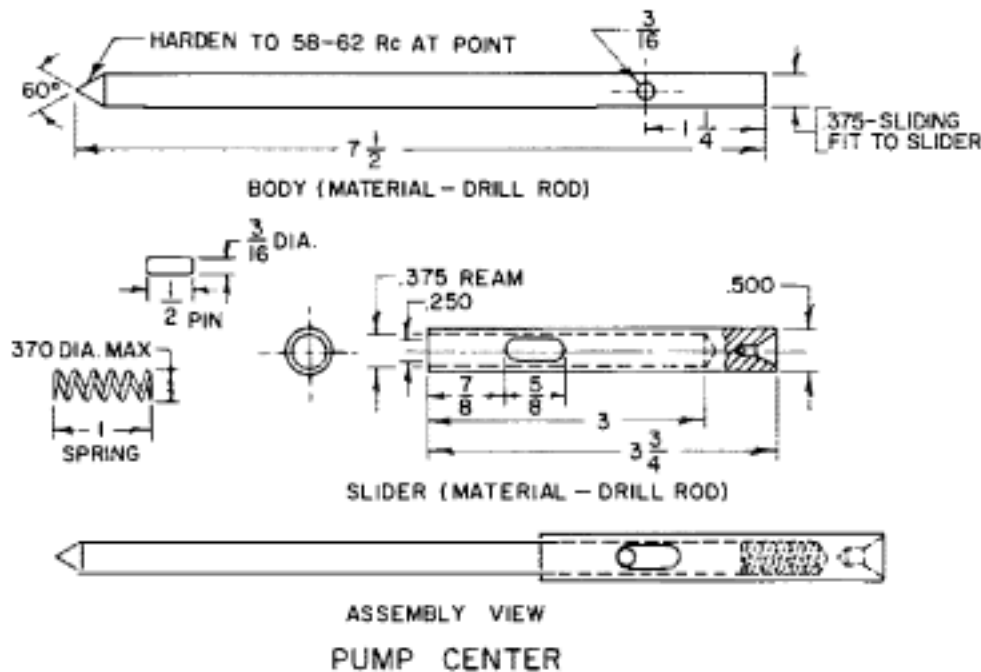


Fig. 15-9. Design drawing of pump center.

on the tailstock center of the lathe and the tailstock is adjusted so that the pump center is held in position with a light spring tension. The contact point of a dial test indicator is positioned against the pump center close to the point. When the lathe spindle is slowly turned by hand, the movement of the indicator hand will show whether or not the prick-punch mark is in the center of rotation. The work is tapped with a hammer and a soft metal bar, as shown, until the indicator hand does not move when the work is turned. The work is then tightly clamped to the faceplate.

5. Drill the hole. The prick-punch mark has served its purpose—that of aligning the part on the faceplate. It is no longer essential to the job. The hole should be started with a center drill. This should be followed by a $\frac{1}{4}$ - or $\frac{5}{16}$ -inch drill which is used to drill the hole through the workpiece. The hole is enlarged by drilling with successively larger drills until it reaches the size for boring. Too much metal should not be removed with any one drill in order to keep the cutting forces from being large enough to move the piece on the faceplate. Sufficient stock should be left for boring to allow several boring cuts to be taken. This can vary from $\frac{1}{8}$ to $\frac{1}{4}$ inch or more.
6. Bore the hole to size. Figure 15-10 illustrates the hole being bored to size. The trial cut procedure is used to bore the hole to size. A trial cut is taken in the bore and the diameter is carefully measured. The boring tool is then advanced into the workpiece the correct amount, and another trial cut is taken. If the measurement



Courtesy of The Moore Special Tool Company

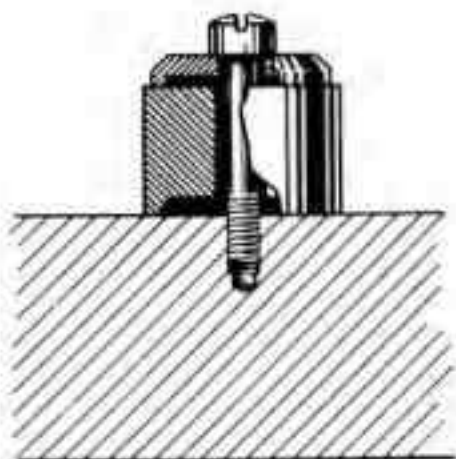
Fig. 15-10. Boring the hole in the lathe with the work mounted on a faceplate.

of the second trial cut shows the diameter to be correct, the cut is completed; if not, a further adjustment is made before completing the cut. The final finishing cut should not be started until the trial cut shows that the diameter being cut is correct.

Several light boring cuts should be taken through the workpiece to make certain that the hole will be concentric with the axis of rotation of the lathe. When using this procedure the hole should be bored to size; it should not be finished by reaming because inaccuracy may result due to any slight misalignment between the spindle axes of the headstock and the tailstock.

Toolmakers Buttons, or the Buttoning Procedure

Toolmakers buttons, Fig. 15-11, are hardened, ground, and lapped collars, or buttons, with the ends ground perfectly square with the cylindrical surface. Thus, when the buttons are clamped in place with the



Courtesy of The Moore Special Tool Company

Fig. 15-11. Toolmakers buttons. The cylindrical surface is hardened, ground, and lapped. The end faces must be exactly perpendicular to the cylindrical surface.

$\frac{1}{8}$ -40 machine screw, the cylindrical surface will be perpendicular to the workpiece. The outside diameter of the buttons is made in several sizes, namely, in .300, .400, and .500 inch. It is not good practice to mix the diameters of the buttons in any one setup as this can result in errors. The inside diameter of the buttons is slightly larger than the screw in order that the button can be moved around the screw.

The buttoning procedure is a good procedure that is frequently used when a jig borer is not available. This procedure is generally more reliable than those previously described. A number of buttons can be attached to the workpiece, simultaneously, in order to locate holes in the workpiece. For simplicity, the description to follow will concern itself with only one hole:

1. Prepare the workpiece. It is machined square and parallel all over, using methods previously mentioned.
2. Lay out the part. When buttoning, a precision layout is not required although the precision layout methods described are often convenient to use.
3. Drill and tap the button screw holes. Each hole is drilled with a No. 38 drill, using ordinary care. All holes are then tapped with a $\frac{1}{8}$ -40 tap.
4. Attach the buttons to the workpiece. The buttons should be clamped lightly but firmly so that they can be moved with a light tap of a hammer.
5. Locate each button on the workpiece. The part is placed on the surface with one of the reference edges down. (See Fig. 15-12.) Attach a dial test indicator to the vernier height-gage, as shown. With the sliding arm of the height gage adjusted so that the vernier scale reads zero, position the contact points of the indicator against the surface plate and adjust the indicator to also read

zero. The top of the surface plate is now the zero reference plane for the height gage and the workpiece. The height gage is set to read the distance from the reference edge to the center of the hole plus one-half of the diameter of the toolmakers button. The indicator contact point is now run over the top of the button and the maximum reading is noted. The button is positioned by tapping it lightly with a hammer and a piece of soft metal, as shown in Fig. 15-12, until the maximum reading of the dial indicator is zero when it is passed over the button. The button is now located with respect to one reference edge and the procedure is repeated with respect to the other reference edge. The toolmakers button is located when the indicator reads zero over the button from each reference edge with the height gage correctly set for each position. It should now be tightened securely in position.

An alternate method of locating the buttons is to use a stack of precision gage blocks. The length of the stack should be equal to the distance over the toolmakers button as explained in the last paragraph. A dial test indicator is held by a height gage and zeroed over the precision gage blocks which are resting on the surface plate. The indicator is run over the top of the toolmakers button and the button tapped into position until the maximum indicator reading is zero, as before. Again, this must be done from



Courtesy of The Moore Special Tool Company

Fig. 15-12. Locating the button on the workpiece. This must be done in two directions from two reference edges.



Courtesy of The Moore Special Tool Company

Fig. 15-13. Aligning the workpiece in the lathe with a toolmakers button.

the two reference surfaces. The use of the precision gage-blocks makes it possible to locate the buttons to within "tenths" when this procedure is carried out with skill and care.

6. Mount the work on the faceplate. This is done as previously described. A counterweight, shown in Fig. 15-14, is used to balance the part and prevent unwanted vibrations from occurring.
7. Align the work on the faceplate. This operation is illustrated in Fig. 15-14. The dial test indicator is placed in the lathe as shown and its contact point is positioned against the toolmakers button. The faceplate is slowly turned by hand and the movement of the indicator hand is observed. The workpiece is adjusted by tapping it with a hammer and a piece of soft metal until the indicator hand does not move when the faceplate is turned at least one complete turn.
8. Remove the toolmakers button.
9. Center drill the hole. Use a large center drill that is consistent, of course, with the final size of the hole. The large center drill, being stiff, will tend to correct any eccentricity in the tapped holes used to clamp the toolmakers buttons.
10. Drill the hole. The drilling operation is done as previously described for the faceplate-mounted workpiece.
11. Bore the hole to size. This is also done as previously described.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-14. Moore precision jig-borer having ultra-precision lead screws to make precise coordinate table positionings.

Toolmakers buttons are also used to perform precision hole location operations on a milling machine. The buttons are located and clamped on the workpiece as has been described for lathe work. The workpiece is aligned and clamped on the table of the milling machine so that the holes to be produced can be reached by tools held in the spindle of the machine. A dial test indicator is attached to the milling machine spindle. Each hole is located by placing the contact point of the indicator against the toolmakers button and slowly rotating the spindle by hand. When the button is located on the axis of the milling-machine spindle, as shown by the indicator reading, it is removed. The hole is then machined by drilling with a twist drill, and boring with a boring tool. For convenience, the boring tool should be held in an offset boring head, Fig. 15-29, which provides an accurate radial adjustment of the boring tool in order to bore the hole to size.

The Jig Borer

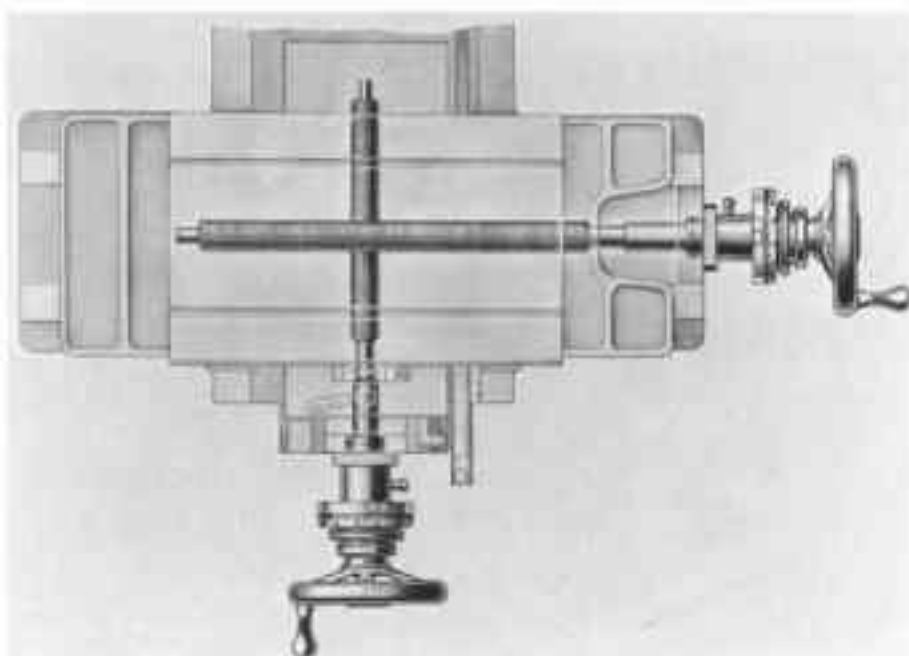
The jig borer is designed specifically for doing precision hole-location work. Its design is a combination of the application of sound engineering principles and a thorough knowledge of toolmaking practices. The highest order of craftsmanship is required in its construction.

A precision jig-boring machine is shown in Fig. 15-14. The bed of the jig borer rests on the floor and the column is bolted to a pad machined on the top of the bed. The column supports the driving motor, the spindle drive mechanism, the power-feed mechanism, the quill which contains the spindle, and the spindle housing. Highly accurate, machine-tool-grade anti-friction bearings are used to mount the spindle in the quill. These bearings are pre-loaded in order to reduce the tendency for chatter to occur when boring. Care is taken in the construction of the machine to make certain that the up and down movement of the spindle is perpendicular to the table.

The table is mounted on the top of a saddle and the saddle is mounted on the bed. The saddle moves toward and away from the column on a machined and hand-scraped slide. It carries the table with it in this motion which is called the *transverse* table movement. A machined and hand-scraped slide on top of the saddle allows the table to move in a direction perpendicular to the transverse direction and this movement is called the *longitudinal* table movement.

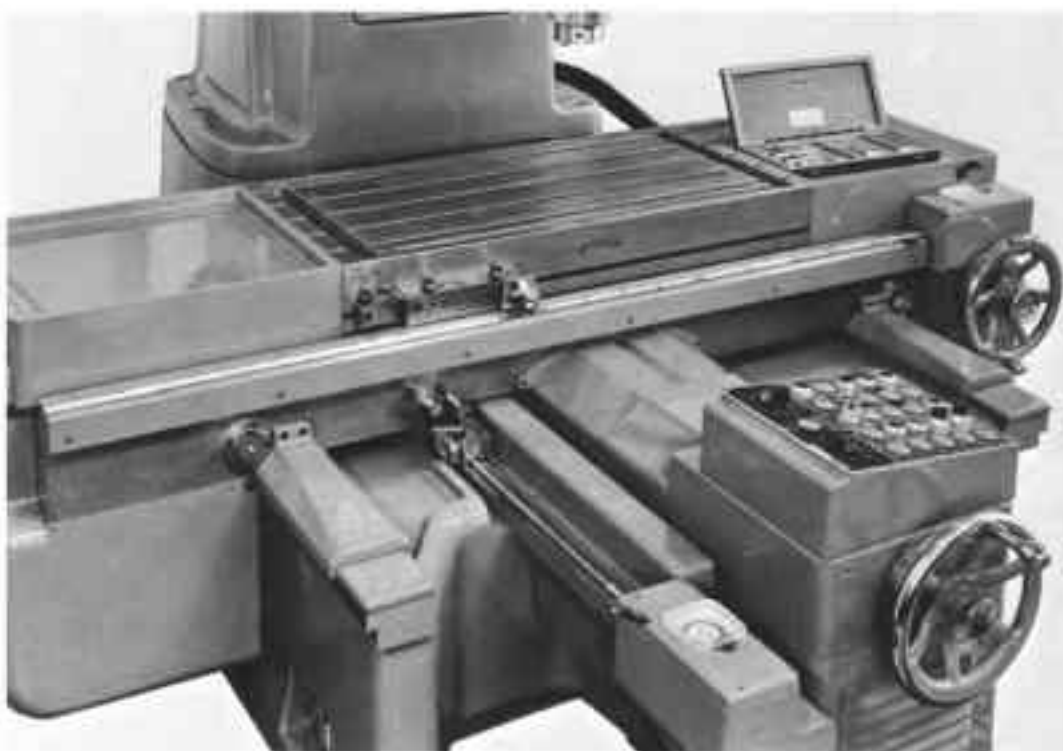
The coordinate measuring system of a jig borer is designed to provide longitudinal and transverse table movements that are accurate to one-ten-thousandths (.0001) inch. On the Moore jig borer, shown in Fig. 15-14, the coordinate measuring system is based on the two lead screws (see Fig. 15-15), that actuate the table movements. These screws are by no means ordinary screws. They are hardened and ground to an accuracy of .00009 inch in 18 inches. A micrometer dial attached to the end of each lead screw is graduated to read one-thousandth (.001) inch. One ten-thousandth (.0001)-inch readings are obtained by a vernier scale which is placed adjacent to the micrometer dial. Two steel rules are attached to the machine and are placed parallel to each of the two table movements. These rules are used to obtain the coarse table settings.

Another coordinate measuring system which is used on many jig borers is shown in Fig. 15-16. This system is based on precision end measuring rods and a micrometer head which are placed in a trough that is parallel to the table movement. There is a trough for each table movement and, therefore, there are also two sets of precision end measuring rods and micrometer heads. The precision end-measuring rods are made in different lengths with one-inch increments. The subdivisions of an inch are obtained by the micrometer head which has a vernier scale that enables it to be set to an accuracy of one-ten-thousandth (.0001) inch. An adjustable stop is placed at one end of each trough. The function of this stop is to move the stack, consisting of the precision end-measuring rods and the micrometer head, along in the trough until it touches the contact point of a dial test indicator which is in a fixed position at the other end of the



Courtesy of The Moore Special Tool Company

Fig. 15-15. System of measuring the coordinate movements of the jig-borer table using precision lead screws.



Courtesy of LeBlond Machine Tool Co.

Fig. 15-16. System of measuring the coordinate movements of the jig-borer table using precision end-measuring rods and micrometer heads. Arrows show dial test indicators.

trough. The only time that the lengthwise adjustment of the adjustable stop is used is after the initial alignment of the reference axes of the workpiece with respect to the axis of the spindle. At this time, only the micrometer head is placed in the trough and the adjustable stop is used to move it along in the trough until the dial test indicator reads zero. At other times, the adjustable stop moves along with the component to which it is attached—either the table or the saddle—and when the table settings are made it pushes the stack until the dial test indicator hand reads zero. The dial test indicator has one-ten-thousandths-inch graduations; however, these graduations should not be used to obtain one-ten-thousandth-inch table settings which are obtained by the setting of the micrometer head. The function of the dial test indicator is to place a uniform end-pressure on the end-measuring rods and the micrometer head, in order to hold them together and to provide a precise reference position for this stack in the trough.

Cleanliness and care must be observed when the precision end-measuring rods and the micrometer head are placed in the trough. The ends, where they contact each other, must be wiped clean. When a setting has been made, a check for cleanliness is made by slightly rotating each precision end-measuring rod and micrometer head in the trough, while observing the hand of the dial test indicator. If the hand moves, dirt is present. This dirt must be removed and the table setting made over again.

Several other coordinate measuring systems are used on different jig borers. It is not possible to describe all of these, in detail, within the scope of this book. The purpose of all these systems, however, is alike. It is to provide a method of aligning the spindle at the precise location on the workpiece where the hole is to be produced. The primary machining operations for which the jig borer is designed are drilling and boring. Operations such as reaming, counterboring, countersinking, and spot facing are also done on the jig borer. Light end-milling operations are sometimes performed on this machine although heavier cuts should be avoided. An air-driven, internal grinding attachment can be obtained for jig grinding. In addition to the production of holes in precise locations, the jig borer can be used to machine certain contours on dies and other tools.

Jig-Boring Practice

There are four basic steps which should be followed in sequence in order to locate and machine a hole on a jig borer. These steps are: 1. align and clamp the workpiece on the jig-borer table; 2. locate the two reference axes of the workpiece with respect to the jig-borer spindle; 3. locate the hole to be machined; and 4. drill and bore the hole to size. These steps will now be treated in detail.

Align and Clamp the Workpiece. The first consideration in placing the workpiece on the jig-borer table should be the relation of the coordinate measuring system of the jig borer to the coordinate dimensions on the drawing. Although the jig-borer table can be accurately positioned when

it is moved in any direction, the coordinate measuring system is designed so that the reading of the measurement is direct when the table is moved toward the left and toward the column. As shown in Fig. 15-17, moving the table forward toward the left has the same effect as moving the spindle toward the right. Likewise, moving the table toward the column has the same effect as moving the spindle away from the column, or down, when seen in the plan view. For this reason, the coordinate dimensions on drawings made for jig-borer work are made to read from left to right and from the top of the view toward the bottom. The reference axes, from which the coordinate dimensions are taken are, therefore, placed at the top and to the left in the view on the drawing which shows these dimensions, as shown in Fig. 15-2.

If the workpiece is placed on the jig-borer table so that one reference axis is toward the column and the other reference axis is toward the left, the coordinate dimensions on the workpiece will read in the same direction as the effective spindle movement when the coordinate measuring system on the jig borer reads direct. In other words, when the workpiece is placed in this position, the coordinate setting of the jig-borer table can be made to correspond exactly to the coordinate dimensions on the drawing in order to locate a hole. This avoids any error that can occur when calculations are required in order to make the table settings.

Frequently, the position of the reference axes corresponds with two of the edges on the workpiece. These edges should be machined perpendicular to each other before the workpiece is placed on the jig borer. They can then be used to align the workpiece on the jig-borer table and to locate it with respect to the spindle. Such workpieces should be placed on the table with the reference edges positioned to the left and toward the column of the jig borer.

If the holes to be bored pass through the bottom of the workpiece it must be placed on precision parallel bars (see Fig. 15-18) in order to provide clearance for the cutting tools and to prevent the table from being damaged. The method of clamping the workpiece will, to a large extent, depend upon the shape of the workpiece. If parallels are used they should be placed directly under the clamps to prevent the force exerted by the clamps from bending the workpiece. If this occurs, the workpiece will spring back when the clamps are released with the possible result that the work done on the jig borer will be inaccurate.

Before the workpiece is firmly clamped in place it must be accurately aligned with respect to the longitudinal and transverse table movements. The most effective method of aligning the workpiece is to indicate along an edge of the workpiece that is parallel to or, preferably, that corresponds to a coordinate dimension reference line. As an example, the workpiece shown in Fig. 15-18 is a die block on which the two reference edges have been machined perpendicular to each other. The reference edges are placed to the left and toward the column of the jig borer. The die block is shown being aligned by indicating along the reference edge. The dial test indicator is held in the spindle of the jig borer and the table is trav-

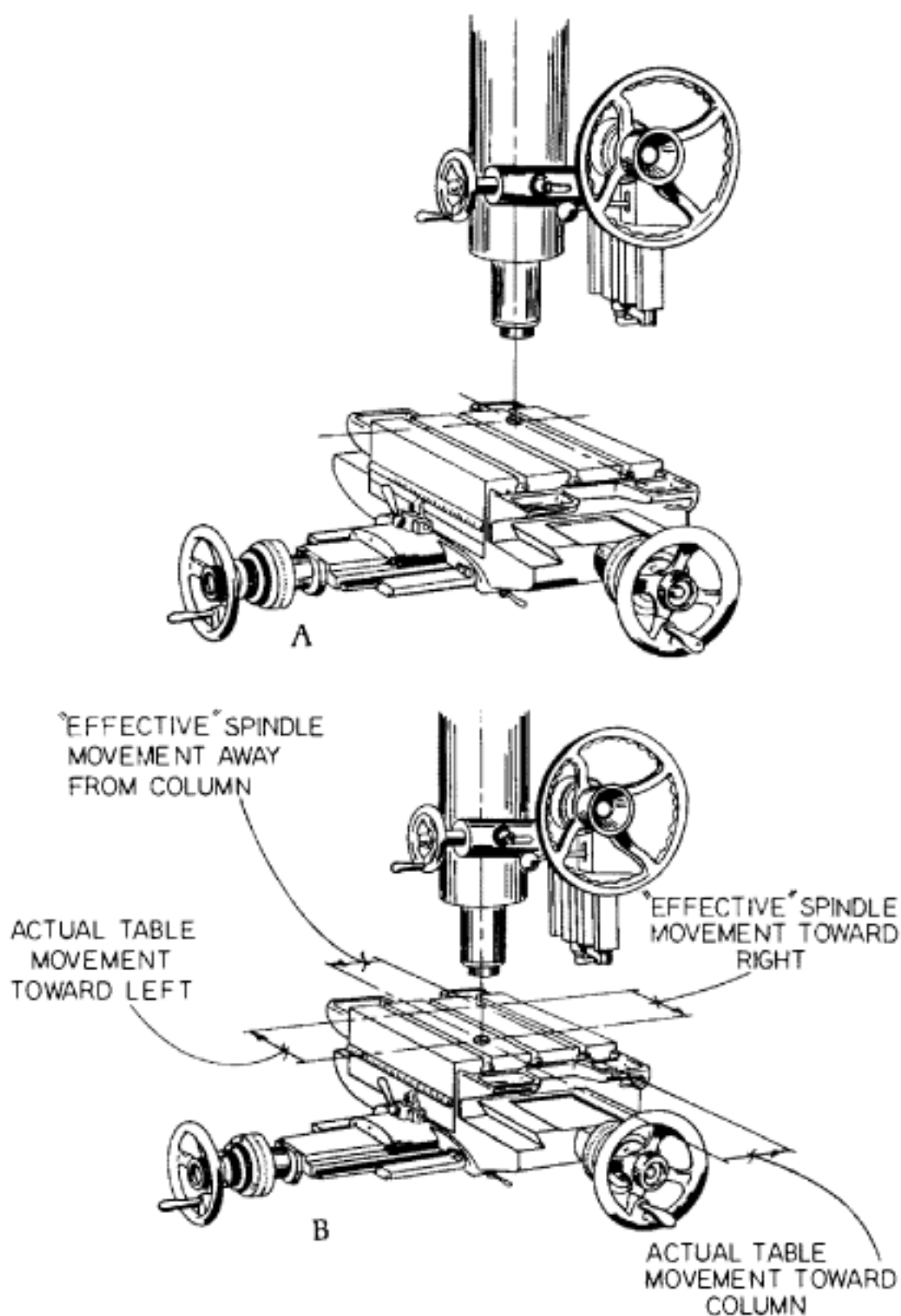


Fig. 15-17. A. Position of table relative to spindle before table movement. B. Position of table after moving in conventional direction by direct reading of coordinate measuring system. Note that the “effective” movement of the jig-borer spindle is opposite in direction from the actual table movement.



Courtesy of The Moore Special Tool Company

Fig. 15-18. Indicating the edge of a die block to align it with the longitudinal table movement.

ersed in the longitudinal direction. As the table is traversed the indicator reading is observed. The workpiece is adjusted until the indicator reading is the same for all positions.

Another method of aligning the workpiece is shown in Fig. 15-19. The jig borer in this case is provided with a straightedge which can be attached to the side of the table and which is parallel to the table travel. The workpiece can be set directly against the straightedge or spaced away by means of precision parallel blocks. For very precise work, this method can be used to obtain the first alignment after which it is checked with a dial test indicator, as shown in Fig. 15-18.

Locate the Two Reference Axes of the Workpiece with Respect to the Spindle. The jig borer table must now be moved to position the workpiece in a precise and known location from where it can be moved again to the location of the holes that are to be machined. Since all of the holes are dimensioned from the two reference lines or axes (see Fig. 15-2), the



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-19. Using straightedge attached to table and precision setup blocks to align workpiece parallel to the table travel.

most convenient position to start from is where the axis of the jig borer spindle and the two workpiece reference axes intersect each other. This is called the starting position, which is akin to a zero reference position. When in this position the longitudinal and the transverse measuring systems of the jig borer are set to read zero. From this position the table and workpiece together can be moved a distance equal to the coordinate dimensions of the hole to the location where the hole is to be machined. Moreover, the readings of the longitudinal and the transverse table position measuring systems will be the same as the coordinate dimensions of the hole when the table is at the hole position.

After the workpiece has been aligned and clamped on the jig borer table, the first step is then to move the jig borer table to the starting position; i.e., where the spindle axis and the reference axes all intersect. This is accomplished by picking up convenient reference surfaces on the workpiece. The term "picking up" means aligning in a special way. For example, a convenient reference surface may be the finished surface

of a hole. Picking up a hole means to indicate the hole with a dial test indicator held in the spindle and to adjust the position of the table until the axes of the spindle and the hole coincide. Picking up an edge means aligning the workpiece so that an imaginary plane will contain both the edge and the spindle axis.

When the workpiece reference axes intersect in the center of a hole that has been finish machined beforehand, it is only necessary to pick up the hole as described before to position the table in the starting position. If the hole has not been machined, other convenient reference surfaces must be picked up and the table moved to the position where the hole is located, which is also the starting position. When in this position, the longitudinal and transverse table position measuring systems are set to read zero.

Frequently the workpiece reference axes will coincide with two mutually perpendicular edges, or sides, which have been machined to precise tolerances, such as on the die block in Fig. 15-18. Although either edge can be picked up first, let us say that the edge closest to the jig borer column is picked up first. When this edge has been picked up the transverse table position measuring system is set to read zero. Next, the edge on the left side is picked up and the longitudinal table position measuring system is set to read zero. Then when the table is moved to the position where both measuring systems read zero, it will be at the starting position.

There are occasions when the measuring system is not set to read zero when the table is at the starting position. For example, the part drawing may show that the reference axes are actually located outside of the body of the workpiece, or the reference axes may not coincide with convenient surfaces on the workpiece that can be picked up. In this event a hole or two perpendicular reference surfaces that are convenient to pick up are selected. After they have been picked up, the table position measuring system is set to read the dimensions from these surfaces (or the hole axis) to the reference axes located outside the workpiece body.

The procedure for picking up an edge on the workpiece must be mastered in order to operate a jig boring machine. The most frequently used procedures will now be described. The first procedure requires no special tools, only a dial test indicator which is mounted in the spindle and a small precision parallel or gage block. This procedure is illustrated in Fig. 15-20 and described as it would be used on a Moore jig borer. The table position measuring system on this machine is based on two precise lead screws, shown in Fig. 15-15. In order to avoid errors caused by the backlash between the lead screw and the nut, the longitudinal table movements should be made toward the left and the transverse table movements should be made toward the column. Made in this manner the micrometer dial readings behind the handwheel will be direct. An arrow near the handwheel points to the directions that the handwheel should be turned. Referring again to Fig. 15-20:

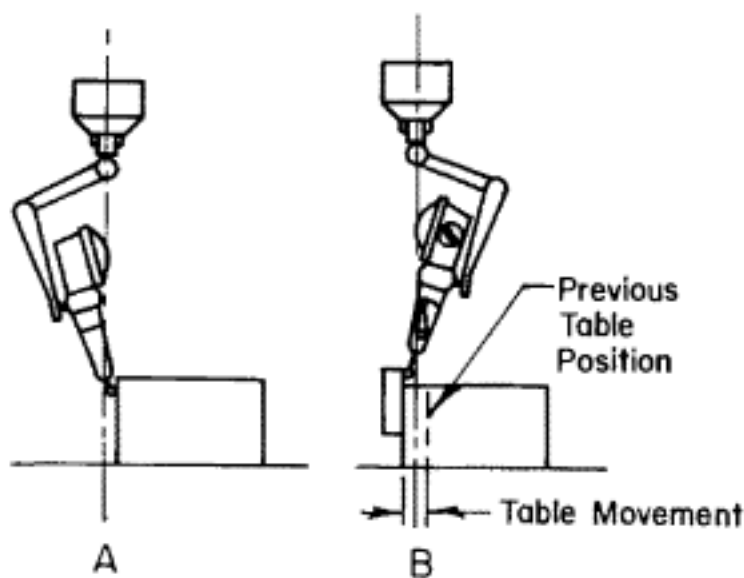


Fig. 15-20. Procedure for picking up the edge of a workpiece using a dial test indicator and a gage block.

1. Adjust the contact point of the dial test indicator to be beyond the spindle axis as shown in view A.
2. Move the table toward the left to a position where an indicator reading is obtained (view A).
3. Rotate the spindle back and forth slightly by hand and stop where the largest indicator reading is obtained.
4. Turn the face of the indicator to read zero; as an alternate the table may be moved to zero the indicator, making the movement to this position toward the left.
5. When in this position (Position A) set the lead screw micrometer dial to read zero.
6. Raise the spindle and rotate it approximately one-half of a turn.
7. Hold a small precision parallel or gage block against the edge of the workpiece (view B) and move the table toward the left to the position (Position B) where the indicator reading is again zero, when indicating the side of the parallel or gage block. This indicator reading is obtained by rotating the spindle back and forth, as before. The final table movement to this position must be made toward the left.
8. When at Position B, read the lead screw micrometer dial to find the distance moved from Position A.
9. The final table position, where the edge and the spindle axis are aligned, is one-half of the distance that the table has moved from Position A to Position B as determined by the lead screw micrometer dial readings.
10. Move the table to the final position, as determined by the lead

screw micrometer dial. The movement to this position must be made toward the left.

When the other edge is being picked up on a Moore jig borer the table movements should be made toward the column so that the readings of the table positions will be direct.

Another method of picking up an edge is shown in Fig. 15-21, which requires the use of two precision parallels. The dimension M of parallel P_1 is accurately measured beforehand. A dial test indicator is then mounted in the machine spindle and the parallel P_1 is placed against the edge of the workpiece. Holding the second parallel, P_2 , against the parallel P_1 as shown, the edge of the workpiece and then the side of the parallel P_2 are indicated by rotating the spindle by hand, and the table is moved to the position where the largest indicator readings are equal. When in this position the axis of the spindle is away from the edge a distance that is equal to one-half of the dimension M . Moving the table this distance will complete the alignment. Figure 15-21A illustrates how this procedure is used on a conventional jig borer, jig grinder, or vertical milling machine; view B illustrates how this procedure is used on a horizontal spindle milling machine or on a horizontal boring machine. While the methods of picking up an edge described in this section are used most frequently on a jig borer and on a jig grinder, it must not be overlooked that they can be adapted for use on other machine tools.

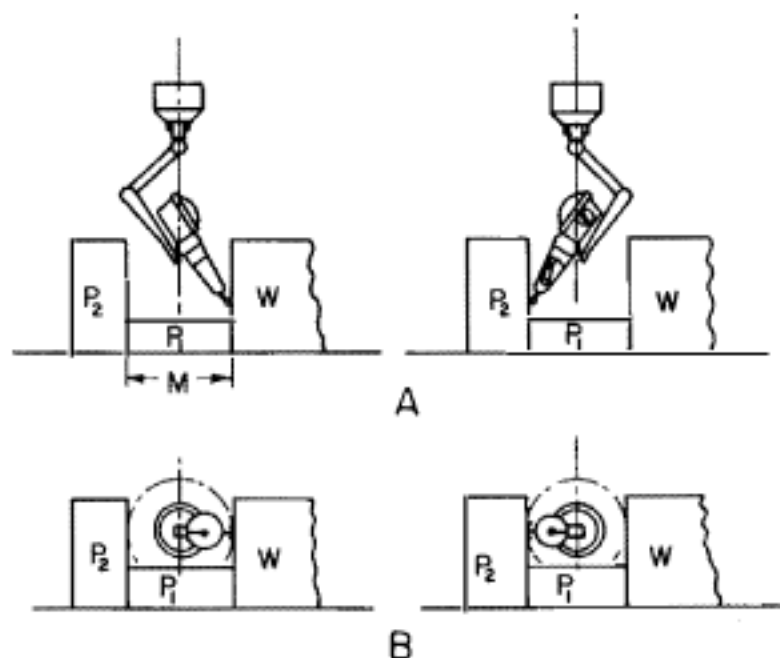


Fig. 15-21. Procedure for picking up an edge using two precision parallels. A. As used on jig borer, jig grinder, and vertical-spindle milling machine. B. As used on horizontal-spindle milling machine or horizontal boring machine.

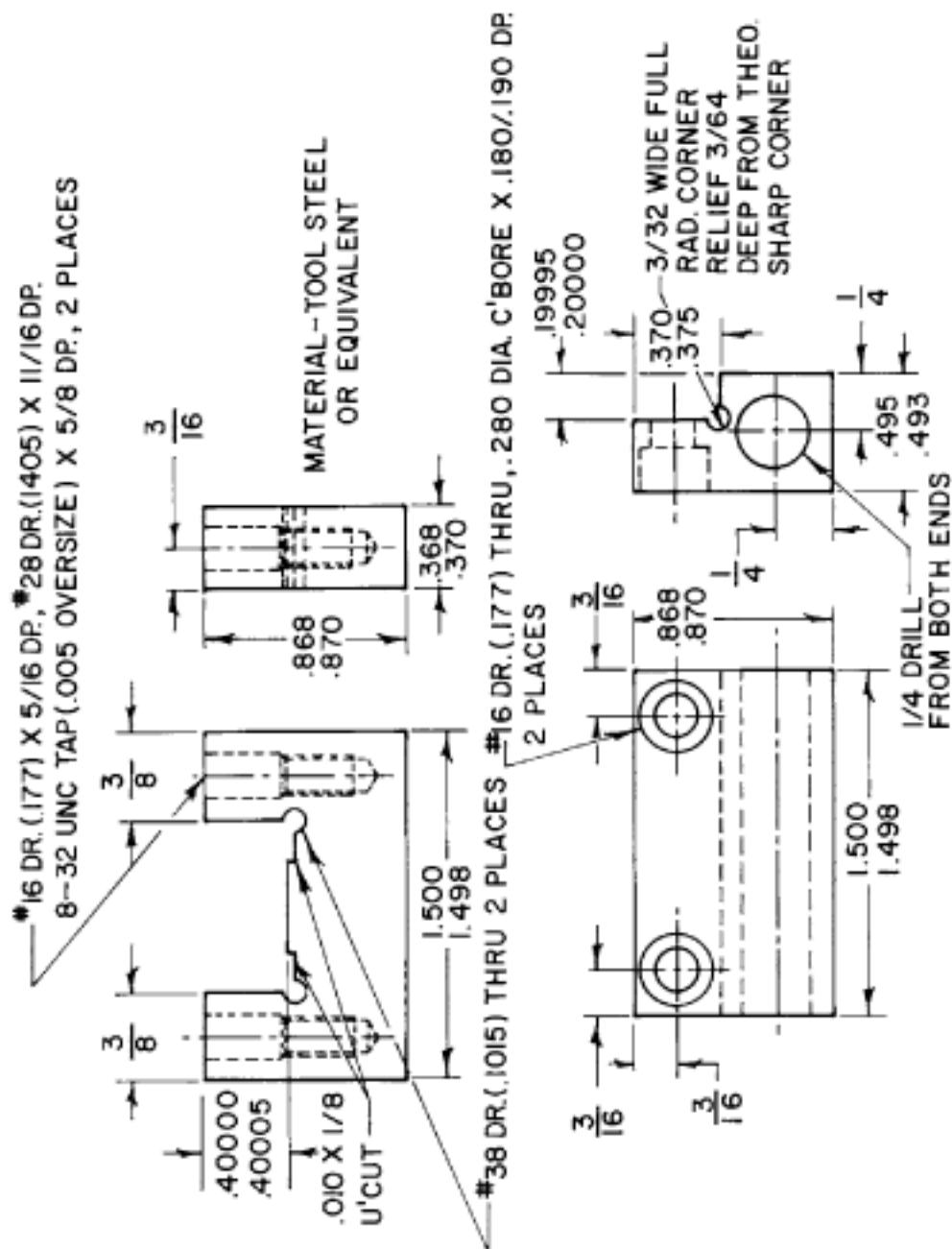
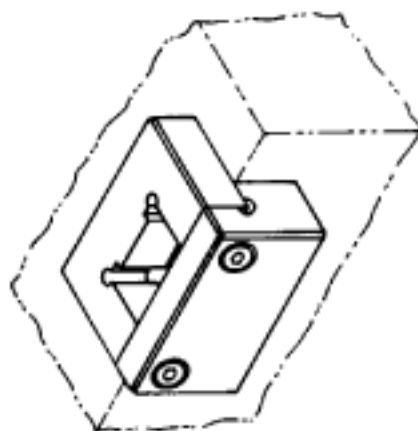


Fig. 15-22. Design drawing of edge finder developed by the Moore Special Tool Company.

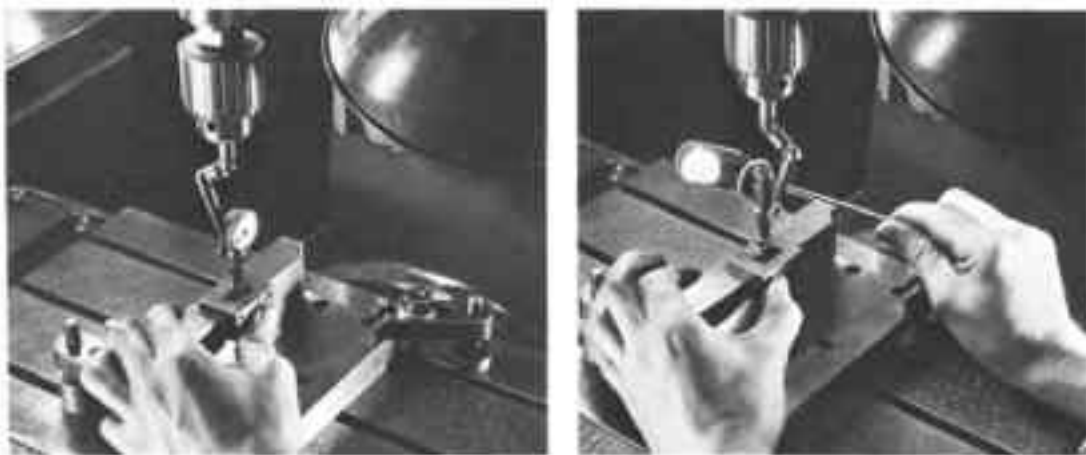
With permission of the Moore Special Tool Co., Inc.



A faster method of picking up an edge is by using the edge finder developed by the Moore Special Tool Co., Inc. Shown in Fig. 15-22 by the kind permission of this company is a detailed drawing of this edge finder. All of the critical dimensions on this edge finder are made to gagemaker's tolerances, including the parallelism and perpendicularity of the gaging surfaces. The upper part of this edge finder has a .400 in. (10 mm in metric) slot. The inner surfaces of this slot are gaging surfaces against which indicator readings are made. The inside edge of the lower leg is also a gaging surface. It is located in the exact center of the slot and, when used, it is held against the edge on the workpiece that is to be picked up. Figure 15-23 illustrates the procedure for using the edge finder to pick up the edge. The lower inside face of the leg is held against the edge on the workpiece while at the same time the two inside faces of the slot are indicated with a dial test indicator held in a jig borer spindle. When indicating each face, the spindle is rotated by hand to obtain the largest indicator reading. The table is moved to the position where both maximum indicator readings are equal; in this position the axis of the spindle will be aligned with the edge of the workpiece.

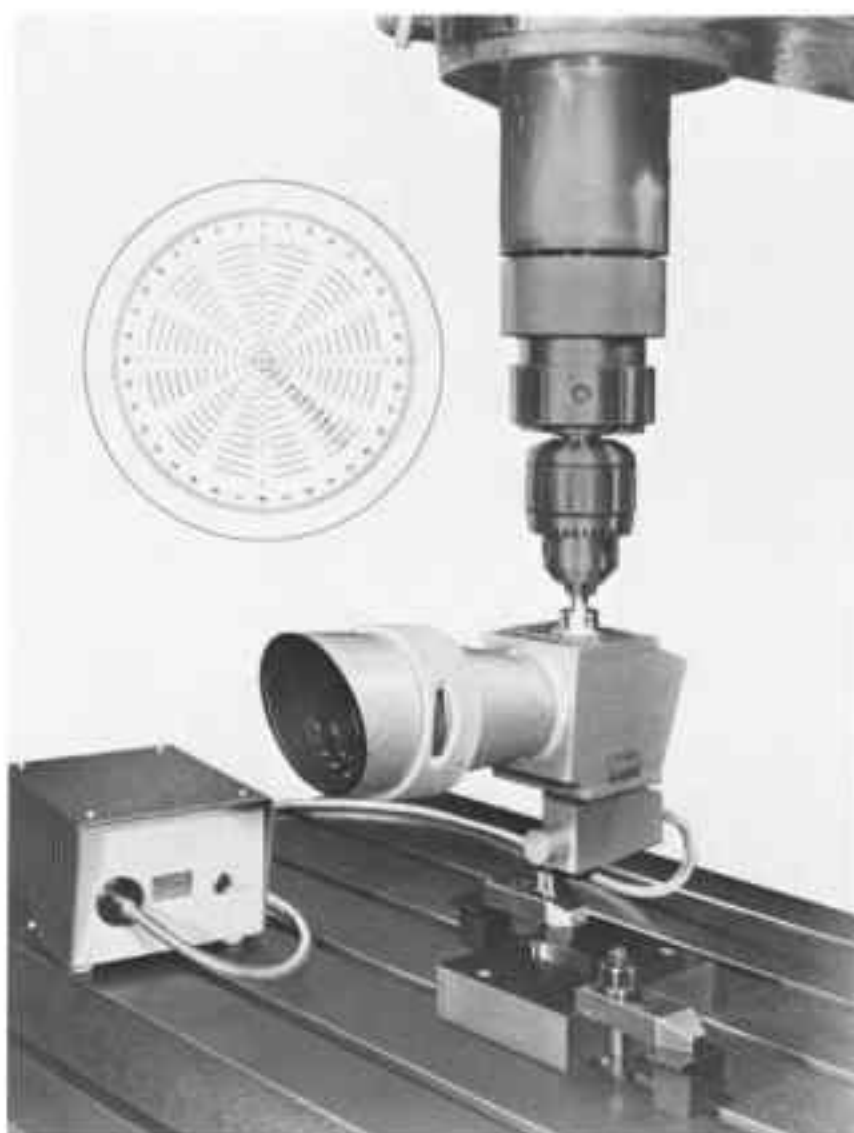
The centering projector, Fig. 15-24, is designed to be used as an edge finder. The eyepiece can be held stationary as the spindle is slowly turned. The magnification is such that "tenths" (.0001 inch) can be seen and the field of vision is large enough to show a fairly large area of the workpiece. Two crossed hairlines act as the center of the optical system. The center of the optical system should be checked, but when verified, it is only necessary to move the table until the edge of the workpiece coincides with one of the hairlines on the microscope in order to align the axis of the spindle on the edge of the workpiece.

A very popular edge finder is shown in Fig. 15-25. It can be obtained with a cone point at one end and a contact cylinder at the other end, or



Courtesy of The Moore Special Tool Company

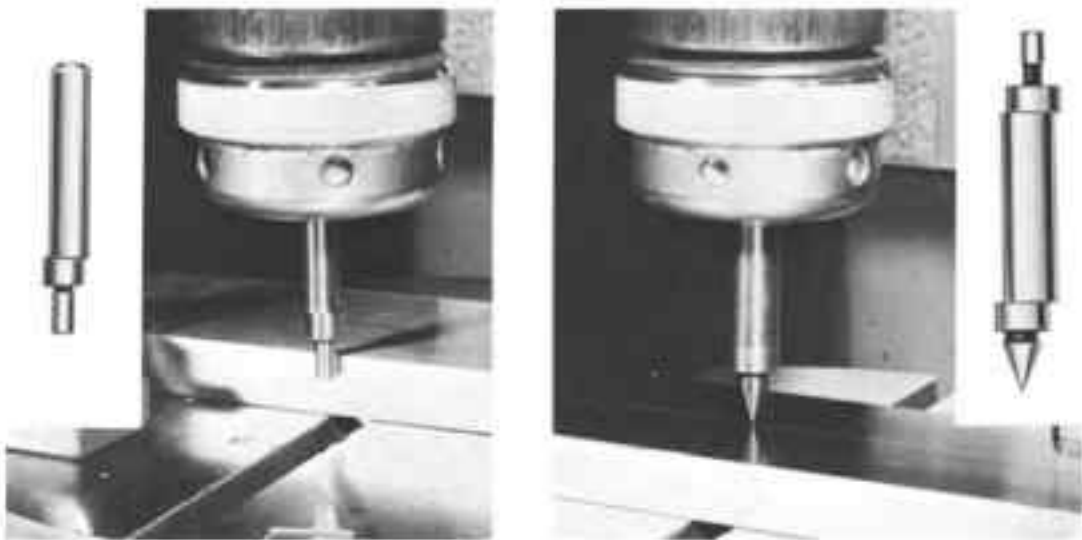
Fig. 15-23. Procedure for picking up the edge of a workpiece using a Moore Edge Finder.



Courtesy of Titan Tool Supply Co., Inc.

Fig. 15-24. Centering projector which can be used to pick up edges, small undercuts, circles, holes, radii, and outside diameters.

with only a contact cylinder. Only a very light pressure is required to slide the cone point and the contact cylinder across the end of the edge finder. The contact cylinder is always used to pick up an edge. The body of the edge finder is held in the machine spindle by a chuck. When the edge finder is used, the spindle must be rotating and the contact cylinder is intentionally offset so that there is a distinct wobble as it rotates. The edge of the workpiece is moved toward the edge finder by moving the table until they contact each other. As the workpiece continues to be moved toward the edge finder, the wobble of the contact cylinder decreases until it ceases altogether. When this occurs the table movement is stopped. At this position the distance between the edge and the axis of the spindle is



Courtesy of The L. S. Starrett Company

Fig. 15-25. (Left) Application of cylindrical contact of edge finder to pick up an edge. (Right) Application of pointed contact to pick up a scribed line.

equal to one-half of the diameter of the contact cylinder. Since the diameter of the contact cylinder on most edge finders is .200 inch, the distance that the table must be moved to align the axis of the spindle and the edge is .100 inch.

The cone-point shape contact is used to pick up lines scribed on the workpiece as shown in Fig. 15-25. In this case the cone point must be made to rotate without wobble. This can easily be accomplished by touching the rotating cone point lightly, with an object such as a pencil. The alignment is then determined visually by lowering the cone point over the line.

Locate the Hole. Precise coordinate table movements are used to position the workpiece so that the spindle axis is located exactly where the hole is to be machined. Essentially identical table position measuring systems are used on the saddle to measure the transverse table position and on the table to measure the longitudinal table position. On the Moore jig borer the coarse measuring system consists of a steel rule, called a scale, that is graduated in .1 inch increments with every inch marked. The fine measuring system is a micrometer dial attached to the very precise lead-screw which is graduated to read .001 inch and an easily read vernier scale graduated to read .0001 inch. On the Moore metric jig borer the coarse scale is graduated in 1.0 mm increments, the micrometer dial in 0.01 mm increments, and the vernier scale in 1.0 μ m, which is 0.001 mm. When the measuring system is set to read zero at the starting position, the coordinate readings at the hole location will be the same as the coordinate dimensions of the hole. The movement to each hole must be made in one direction only in order to eliminate the effect of backlash.

This should always be toward the left and toward the column of the machine. Although the holes can be machined in any sequence, it is easier and more efficient on the Moore jig borer to machine the hole that is closest to the starting point first, then the next closest, etc., until all of the holes have been machined. The same sequence is recommended for machines having an optical measurement system.

On jig borers having a table position measuring system utilizing end measuring rods, as shown in Fig. 15-16, this sequence is reversed. It is much more convenient and efficient on these machines to machine first the hole that is furthest from the starting position, then the next furthest hole, etc. The largest stack-up of end measuring rods in the troughs, in addition to a micrometer head, will occur when machining the hole furthest from the starting position. For machining the next furthest hole the length of this stack-up will decrease and will be shortest when machining the hole closest to the starting position. In this way end measuring rods are removed and replaced by shorter rods when positioning the table at each successive hole, and the table is moved progressively toward the starting position.

Drill and Bore the Hole. The sequence of operations used to produce a hole on the jig borer is based upon two basic principles: 1. A short, stiff drill, such as a center drill, that will not deflect when cutting should be used to start a hole whenever the work and the axis of the machine tool spindle are located at the exact position where the hole is wanted; 2. A single-point boring tool that is rotating about the axis of the machine tool spindle will generate or cut a surface that is concentric to the axis of rotation. Of course, a drill is required to open the hole so that the boring tool can enter and enlarge it.

The actual steps required to machine the hole are now listed in a numbered sequence.

1. Spot drill the hole with a center drill or a spotting drill. This operation is illustrated in Fig. 15-26. A conventional drill and countersink, or center drill, is recommended for this operation except when the holes to be machined are very small. In this case a special spotting drill, shown in Fig. 15-27, should be used. The spotting drills must be kept sharp because they are not intended to penetrate deeply into the workpiece. Sometimes all of the holes to be produced in a part are spot drilled before they are drilled and bored. Some jig borer operators will spot all of the holes lightly and then re-spot them to depth in order to insure against making errors.
2. Drill the hole. The initial hole should be drilled with a twist drill that will follow the countersink made by the center drill. Although larger drills are sometimes used, it is best not to start with a drill in excess of about one-half inch in diameter. If required, the hole can then be enlarged by drilling with successively larger drills.



Courtesy of The Moore Special Tool Company

Fig. 15-26. Spot-drilling holes in a die block on a jig borer.

The drill must leave enough metal in the hole to allow the boring tool to finish the hole to size. Holes that are beyond the range of ordinary twist drills may be enlarged with a sweep tool as shown in Fig. 15-28. The tool shown is called a "hole hog" and is particularly efficient for removing large amounts of metal. A hard-



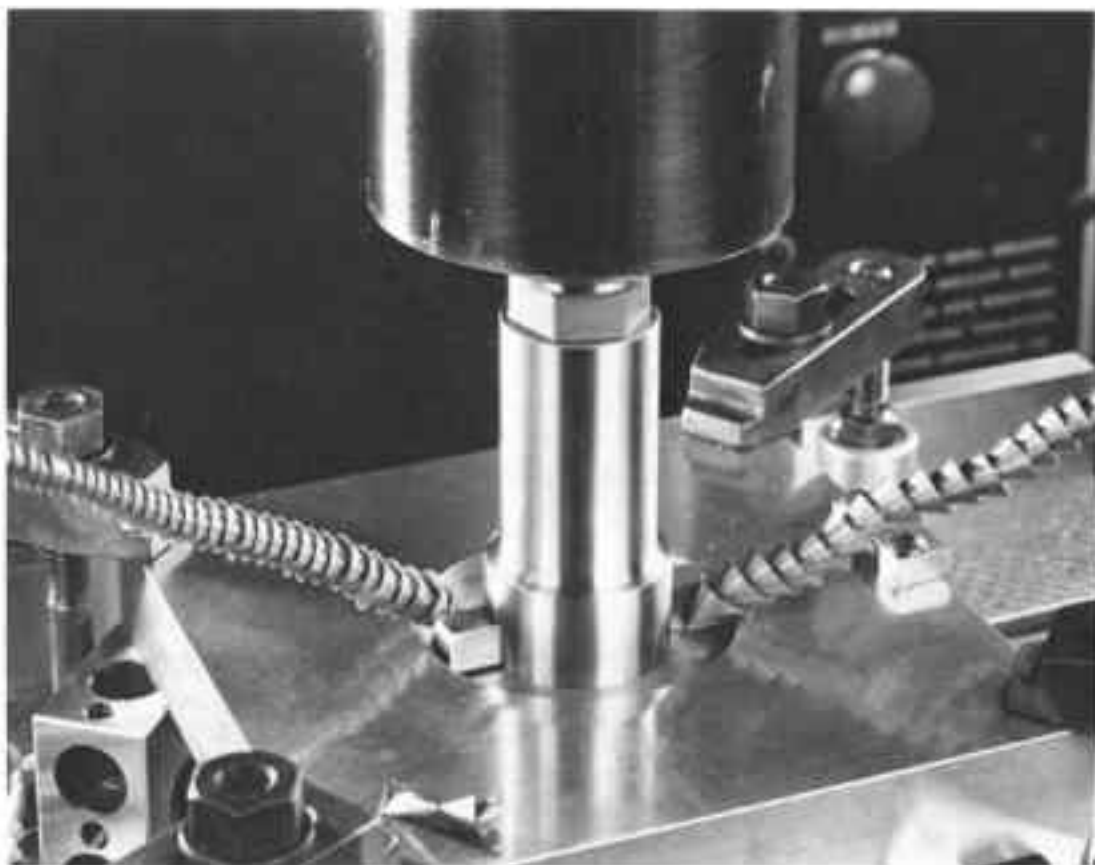
*Courtesy of The Moore Special Tool Company
Courtesy of The Dumont Drill Company*

Fig. 15-27. Spot drilling for producing very small holes using a spotting drill.
(Right) Special spotting drill.

ened pilot on the end of the bar guides the sweep tool in the hole that has been drilled previously.

Heat will be generated by the drilling operation, which will cause the work to expand, especially when much metal is removed by larger size drills. If the finish-boring operations are conducted while the work is very hot, the contraction occurring when the workpiece cools can cause inaccuracy. For this reason it is sometimes good practice to drill all of the holes first, allowing the workpiece to cool before the holes are bored to size. A procedure that is used at times is to spot drill all of the holes on the jig borer, rough drill all of the holes on a conventional drill press, and then finish the holes in a jig borer by boring.

Holes less than one-sixteenth inch in diameter are very difficult or even impossible to bore with a single-point tool. The first step in producing these small holes is to drill a spot with a spotting drill. The diameter of this spot should be somewhat less than the diameter of the hole. One method of finishing the hole is to drill it to size with a twist drill. A small drill will follow the spot made



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-28. Sweep tool used for rapid stock removal from a hole where hole size is beyond range of ordinary twist drill.

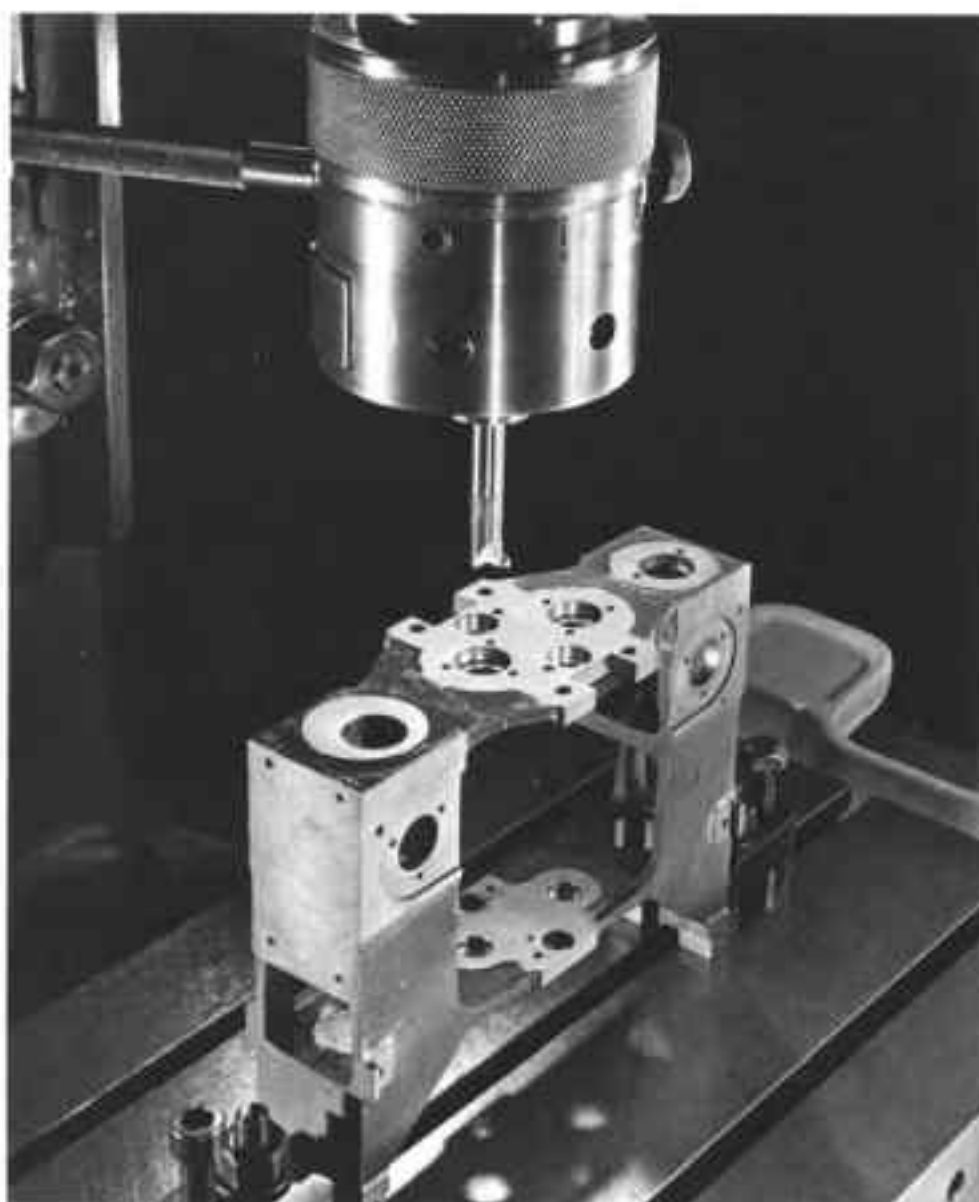
by the spotting tool quite accurately. Another method of finishing the hole after it has been spotted, is to drill it first with a twist drill that is one size smaller than the finish size of the hole. The hole is then finished with a twist drill of the size of the finished hole. This last drill is very carefully honed at the corners where the lip and the margin meet. Only a small amount of honing should be done on the two corners and care must be exercised to hone a relief angle so that the drill will cut properly. Small holes may be finished by reaming after they have been spotted and drilled. This is frequently done when no suitable drill is available corresponding to the diameter of the hole. In such cases it is not uncommon to make a special reaming tool or to grind a standard reamer to the desired diameter.

3. Bore the hole to size. The offset boring head in Fig. 15-29 provides an accurate and reliable method of adjusting the boring tool to cut the desired diameter on the workpiece. There are several designs of offset boring-heads but they all serve to radially offset the boring tool. The amount of offset can be accurately controlled by graduations which are usually in one-ten-thousandths (0.0001) inch. On some offset boring-heads the boring tool can be fed radially outward, continuously, which is done when taking external facing cuts around a hole, when facing internal shoulders, and when cutting grooves inside the hole.

A boring operation is illustrated in Fig. 15-30. The trial-cut procedure is used to perform the boring operation. A short trial cut is taken inside the bore and the resulting diameter is measured. Using the graduations on the offset boring-head, the boring tool is adjusted to the distance required to bore the hole to the desired size. If the hole is close to the finish size, another trial cut is taken and a measurement is made to verify the previous setting of the offset boring-head. The hole is then bored to the required depth.

When the diameter of the hole must be greatly enlarged by the boring tool, one or more rough boring cuts are taken to remove the bulk of the metal. If the workpiece becomes very hot in this process it should be allowed to cool before the finish boring cuts are taken. Before the hole is bored to the finish size, one or two shallow cuts should be taken through the hole with a single-point boring tool. These cuts are important because they remove any error in the location of the hole brought about by the operations performed beforehand. After a sufficient number of light boring cuts have been taken through the hole, it is finished to size with the single-point boring tool.

Sometimes, a reamer is used to finish the hole to size; however, before this tool is used, the location of the hole must be established by taking several light boring cuts with a single-point boring tool, as before. The hole should be bored .008 to .030 inch undersize

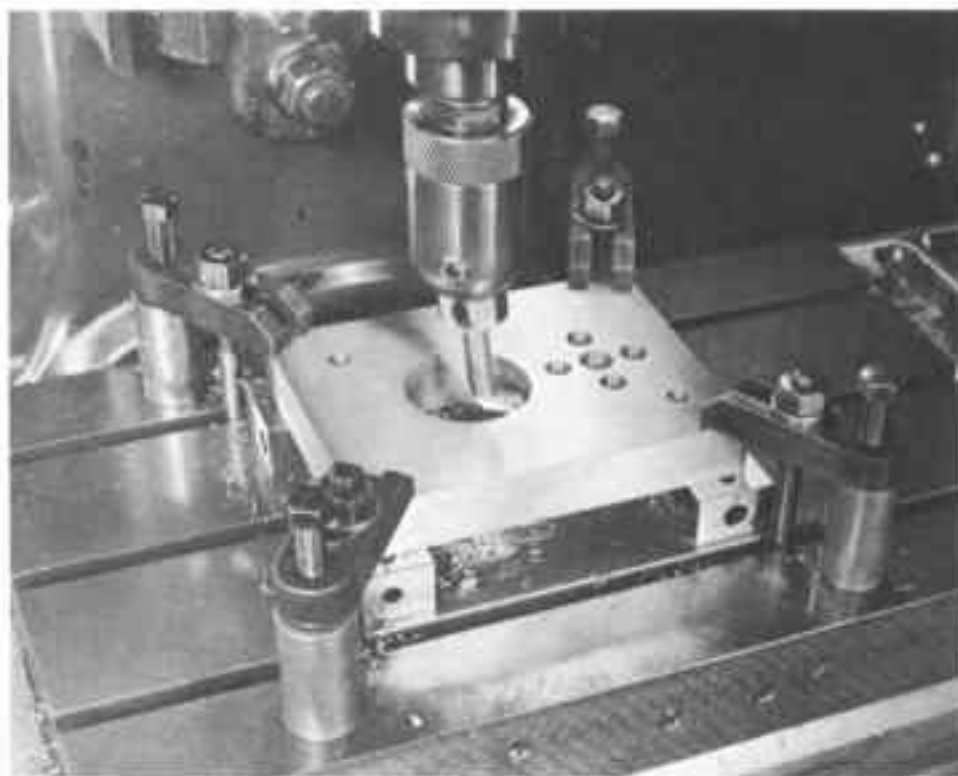


Courtesy of The Moore Special Tool Company

Fig. 15-29. Offset boring head.

and the reamer is used to remove the excess metal in the hole to bring it to the required size. While this procedure is faster, it is not quite as accurate as when the hole is finished by boring.

Using either sequence of operations, the best jig boring practice is to spot, drill, and rough bore all of the holes before they are finish bored, or finish bored and reamed. This reduces the possibility of human error because the table must be positioned at each hole location several times to serve as a check. Errors caused by temperature changes and by stresses are also reduced.



Courtesy of The Moore Special Tool Company

Fig. 15-30. Finish boring the hole to size on the jig borer.

Jig Borer Accessories

Many accessories are available to enlarge the scope of the work that can be done on a jig borer. Two accessories for which there is frequent use are shown in Fig. 15-31. In this illustration the workpiece is mounted on a rotary table held in an angular position by a sine table. The sine table works on the principle of a sine bar or sineplate (see Fig. 2-28). When in a horizontal position the table rests on two rolls attached to its underside precisely 10 inches apart. It can be raised to a precise angular position by placing a stack of precision gage blocks below one of the rolls while the other remains clamped to the base. The height of the gage block stack is calculated by multiplying the trigonometric sine function of the required angle by 10.

There are many uses for a rotary table on a jig borer. The combination of a rotary table clamped to the sine table as shown in Fig. 15-31 is often used to jig bore holes that are at a compound angle with respect to other holes and surfaces. In this illustration the holes are spaced around a common center by indexing the rotary table. Some rotary tables



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-31. Jig-boring holes at an angle and equally spaced in a circular pattern around a common center. The work is mounted on a rotary table which is clamped to a sine table.

have an index plate and an indexing crank, like a dividing head*, with which the table can be made to rotate an exact angular amount or through equally spaced divisions, such as gear teeth. Other rotary tables are graduated to read directly in degrees, minutes, and seconds. Precise rotary tables built by the Moore Special Tool Company, Inc. read to 5 seconds and have an accuracy of ± 6 seconds. Ultra-precise rotary tables built by this company read to 1 second and are accurate to ± 2 seconds. These tables can also be provided with index plates as an attachment.

When a rotary table that reads in degrees, minutes, and seconds must be rotated a number of equally spaced divisions, the angle between each division must be calculated by dividing the number of divisions into 360 degrees. Since the rotary table is graduated continuously from 1 to 360 degrees, the cumulative value of the angle between each division must also

* See Chapter 7, *Machine Shop Practice*, Volume 2.

be calculated. If the true angle between each division is not a whole number of degrees, minutes, and seconds, the remainder must be carried as a fraction of a second in order to avoid errors when calculating the cumulative angle.

Example 15-1:

Calculate the cumulative angular reading between each hole when 7 holes are to be bored in a circular pattern about a common center on an ultra-precision rotary table.

$$360 \div 7 = 51^{\circ} 25' 42\frac{6}{7}''$$

Thus, the true angle between each division is $51^{\circ} 25' 42\frac{6}{7}''$; this amount must be added to each successive angle to find the cumulative angular position of each hole.

	<u>Degrees</u>	<u>Minutes</u>	<u>Seconds</u>
Hole 1	51	25	$42\frac{6}{7}$
Add	51	25	$42\frac{6}{7}$
Sum	102	50	$84\frac{12}{7}$
Hole 2	102	51	$25\frac{5}{7}$
Add	51	25	$42\frac{6}{7}$
Sum	153	76	$67\frac{11}{7}$
Hole 3	154	17	$8\frac{4}{7}$
Add	51	25	$42\frac{6}{7}$
Sum	205	42	$50\frac{19}{7}$
Hole 4	205	42	$51\frac{3}{7}$
Add	51	25	$42\frac{6}{7}$
Sum	256	67	$93\frac{9}{7}$
Hole 5	257	8	$34\frac{2}{7}$
Add	51	25	$42\frac{6}{7}$
Sum	308	33	$76\frac{8}{7}$
Hole 6	308	34	$17\frac{1}{7}$
Add	51	25	$42\frac{6}{7}$
Sum	359	59	$59\frac{7}{7}$
Hole 7	360	0	0

The actual rotary table positions for indexing are the values opposite the Hole numbers in the above table. Since there are 60 minutes in one degree and 60 seconds in one minute, the values at the table positions of the minutes and seconds must not exceed 60. It is, therefore, necessary to resolve the values into numbers that can be read on the rotary table scale. For example, $84\frac{12}{7}$ seconds (see the row above Hole 2) is larger than

one minute, or 60 seconds. To resolve this number, remember that the numbers $59 \frac{7}{7}$ and 60 are equal. To find the seconds for Hole 2, subtract $59 \frac{7}{7}$ from $84 \frac{12}{7}$, which results in $25 \frac{5}{7}$ seconds. This leaves one minute ($59 \frac{7}{7}$ seconds) which must be added to the 50 minutes, making 51 minutes as the value for Hole 2. The other values are treated in the same manner, when necessary.

Precision V-blocks, Fig. 15-32, are used to hold cylindrical parts. In the operation shown in this illustration, a light facing cut is being taken inside of a hole with an offset boring head. Some offset boring heads, such as the one shown in Fig. 15-33, are capable of feeding outwardly by holding the upper ring fast with a hand lever while the body is rotated by the spindle. In this manner facing cuts, recessing cuts, undercutting, and outside diameters can be cut, in addition to the usual precision boring operations.

Also shown in Fig. 15-33 are some typical T-slot bolts, strap clamps, heel blocks, and precision parallel setup blocks used to make setups on jig borers. A plentiful supply of these accessories is required to accommodate the wide variety of setups that are made on a jig borer in an



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-32. Cylindrical workpiece set up on precision V-blocks for jig-boring.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-33. Universal offset boring and facing head shown taking a facing cut.

average shop, a typical example of which is illustrated in Fig. 15-34. Matched box parallels are used in this setup to support the workpiece. Other accessories that are also frequently used but not illustrated are precision angle plates and precision jig borer vises.

In addition to the usual precision hole machining operations, the jig borer can also be used to take light end milling cuts. In order to preserve the accuracy of the machine, it is not advisable to take extremely heavy cuts. Larger surfaces can be milled with a fly cutter, such as shown in Fig. 15-35. This cutter will sweep a relatively large area and yet not overload the spindle because only a single cutting edge is doing the cutting.

Jig Grinding on the Engine Lathe

Hardened metals cannot be cut with an ordinary single-point cutting tool. The most effective way of machining these metals is by grinding. Tools and dies must frequently be finish-ground after they have been hardened. Certain precision machine parts must also be ground after they are hardened. Holes in hardened machine parts are frequently ground on



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-34. Precision box parallels used to elevate and clamp work above table-top.

internal grinding machines, or on cylindrical grinding machines, using an internal grinding attachment. Although holes in hardened tools and dies are, on occasion, also ground on these machines, the size and nature of these components does not usually lend them to this method of grinding. Holes in hardened tools and dies can also be ground on an engine lathe (Fig. 15-39) but a preferable method is to grind them on a jig-grinding machine (Fig. 15-40) if one is available.

A hole must exist before it can be enlarged by grinding; thus, the holes that are to be ground have been drilled and bored into the workpiece before it is hardened. The heat treatment used to harden the workpiece usually causes a small change in the size of the hole and possibly, in its roundness. The grinding operation is used to correct these errors and to reestablish the precise location of the hole.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-35. Using a fly cutter to mill a large surface area on a jig-borer.

Enough stock must be left in the hole by the drilling and boring operation to allow the hole to be finished to size by grinding. The amount of grinding stock to be left in the hole will depend upon the size of the hole, the depth of the layer of hardened metal surrounding the hole, and the shape of the workpiece. Generally, but not invariably, the amount of stock left for grinding should be .008 to .015 inch for small holes and .016 to .030 inch for larger holes. Hardened tools and dies, such as the blanking die in Fig. 15-37, should be ground all over on a surface grinder before the holes are ground. These surfaces should be ground square and parallel to each other.

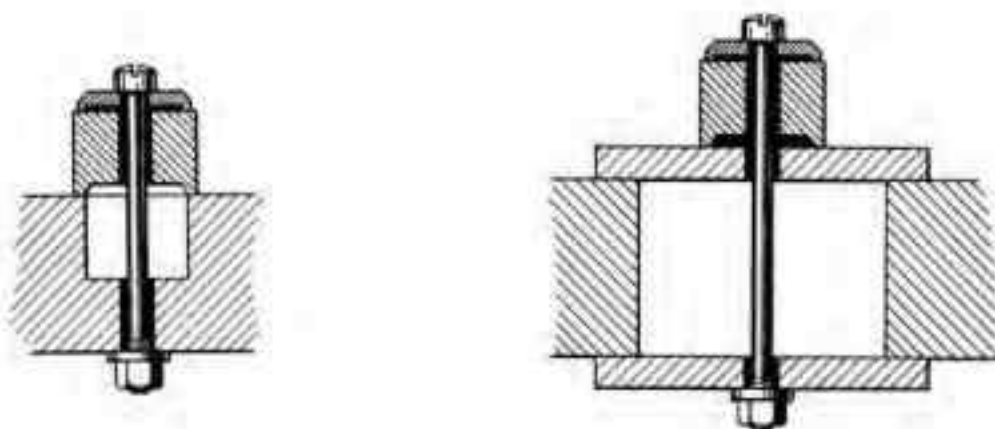
Usually, the existing holes in tools and dies form the basis from which the final hole location is determined. Thus, the first step is to measure the location of the existing holes. Errors caused by the heat-treating operation or by previous machining operations will thus be detected and small errors can be corrected by the hole-grinding operation. Sometimes this is

done by slightly shifting the location of each hole provided that there is sufficient grinding stock in each hole.

One method of locating the holes for grinding is by buttoning. The toolmakers buttons are fastened over the existing holes by one of the methods shown in Fig. 15-36. The buttons for all of the holes can be located from the two reference edges on the side of the workpiece by using the procedure described earlier in this chapter. In other instances, one of the holes is first ground to size and the remaining holes are then located from this hole.

A general outline of the procedure used is given using a blanking die as an example. The first step is to mount this die on a faceplate which has been removed from the lathe. (See Fig. 15-37.) It is easier to mount the die when the surface of the faceplate is in a horizontal position. After the faceplate is attached to the lathe spindle, the blanking die is accurately located by indicating the hole as shown in Fig. 15-38. The hole is then ground to size using a tool-post grinding attachment. This operation, illustrated in Fig. 15-39, also serves to locate this hole in the die. A slight taper, or "draft" as it is called, of about one-quarter degree is then ground in the bottom of the hole to provide a clearance so that, when the die is used, the stock can fall through the die. It is ground by traversing the grinding wheel through the hole with the compound rest which is positioned at the draft angle.

The blanking die is then removed from the faceplate and toolmakers buttons are attached roughly in place over the remaining holes. It is placed on a precision surface-plate and each button is located by measuring from the hole that has been ground. The blanking die must be mounted on the faceplate again and located by indicating one of the toolmakers buttons as shown in Fig. 15-13. After the button is located it is removed. The hole is ground to finish size and the draft is ground in the back of the hole. This procedure is repeated until each hole in the die has been ground.



Courtesy of The Moore Special Tool Company

Fig. 15-36. Methods of fastening toolmakers buttons over existing holes.



Courtesy of The Moore Special Tool Company

Fig. 15-37. Clamping the workpiece to a lathe faceplate. The faceplate is removed from the lathe spindle to make this job easier.

The Jig Grinder

An increase in accuracy and a savings in time are possible if the job described above is performed on a jig grinder. This machine, illustrated in Fig. 15-40, was developed by The Moore Special Tool Company as an answer to the jig-grinding problem. It is very similar in construction to the jig borer. The longitudinal and transverse table positions are obtained with the aid of a precise coordinate measuring system. On the Moore jig grinder, in Fig. 15-40, this system is based on two precision lead screws. For each direction of table movement the coarse table position is obtained by reading a steel rule, and the precise setting is obtained by reading a micrometer dial at the end of the screw. A vernier scale adjacent to the micrometer dial makes it possible to read one-ten-thousandth (.0001) inch as in the case of the jig borer.

In addition to this measuring system, the jig grinder in Fig. 15-40 has a switchable inch-metric digital readout that may be obtained to read either in increments of .00005 in./0.0005 mm or .00001 in./0.001 mm. The same digital readout can also be installed on jig borers.

A fixed bridge type jig grinder is shown in Fig. 15-41. On this jig borer



Courtesy of The Moore Special Tool Company

Fig. 15-38. Locating a blanking die on a faceplate by indicating the hole.



Courtesy of The Moore Special Tool Company

Fig. 15-39. Locating a hole in a hardened die by mounting it on a faceplate and grinding it in a lathe using a toolpost grinding attachment.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-40. Precision jig-grinder with switchable inch-metric digital readout. Positioning accuracy is .000030 inch in one inch and .000000 inch in 18 inches.

the table can only move in the longitudinal direction; the traverse movement is obtained by moving the spindle housing carrier across the top of the bridge. Vertical adjustments are made by moving the spindle



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-41. Fixed bridge type jig-grinder equipped with inch-metric preselect and readout systems.

housing up and down. When grinding, the grinding spindle moves up and down with a reciprocating motion. The coordinate hole locations are obtained by a switchable inch-metric preselect and readout system. The required coordinate dimensions are dialed into the system and displayed on the digital readout; then, the movements to the coordinate location are made by simply depressing a push button. These movements and the necessary position readings can also be made by means of manual controls.

On all jig grinders the grinding wheel spindle is driven by an air turbine. Four different turbines having different spindle speeds can be attached to the machine; the speeds range from 7,000 rpm to 175,000 rpm. The grinding wheel is moved radially outward from the axis of rotation of the main spindle by a coarse and fine control, thereby providing an

adjustment for grinding the holes to size. Provisions are also made to enable tapered holes to be ground, having an angle up to $1\frac{1}{2}$ degrees, or a 3 degree included angle. This feature is used to grind the draft or stock clearance in the lower portion of die openings. Holes as small as $1/64$ in. (0.4 mm) can be ground using small grinding mandrels that are charged with a diamond abrasive.

The type of abrasive used on the grinding wheel depends on the type of material to be ground, the amount of stock to be removed, the surface finish required, and the size of the grinding wheel and the size of the hole. Many steels, including tool steels, can be ground with an aluminum oxide abrasive. A diamond abrasive is used to grind very small holes and cemented carbides. Cubic boron nitride abrasive, sold under the trade name of **Borazon®**, is very effective for grinding tool steels, especially the high carbon, high chrome, and high-speed steels.

Jig Grinding Practice

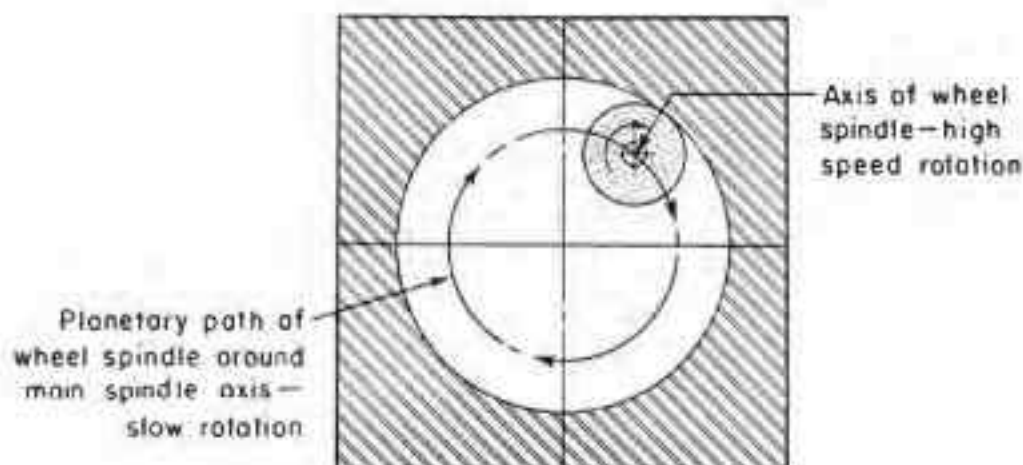
The jig grinder is used to grind holes and complex profile contours in punches, dies, jigs, molds, and on other precision parts. The operating procedure of a jig grinder is very similar to operating a jig borer, the principal difference being that the hole is ground to size instead of being bored to size. This procedure involves the same four basic steps which follow:

1. Align and clamp the workpiece on the jig grinder table.
2. Locate the reference axes, or surfaces, of the workpiece with respect to the main spindle.
3. Locate the spindle axis over the hole to be ground in the workpiece.
4. Grind the hole to size.

Since the first three steps are done exactly alike on the jig grinder and jig borer, they will not be repeated.

The general procedure for grinding a number of holes in a workpiece on a jig grinder will vary somewhat, depending on the nature of the workpiece. However, very frequently, the best procedure is first of all to rough grind all of the holes .001 to .004 inch (0.02 to 0.10 mm) undersize, after which the workpiece is allowed to cool to room temperature; then all of the holes are finish ground to size. Often it is helpful to unclamp the workpiece and then to clamp it again before finish grinding in order to relieve the stresses set up in the workpiece by the rough-grinding operation.

Holes are ground on the jig grinder by the planetary internal grinding method, which is illustrated in Fig. 15-42. In planetary grinding the grinding wheel rotates at a high speed about its own axis while at the



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-42. The principle of grinding a hole on a jig-grinder by planetary internal grinding method.

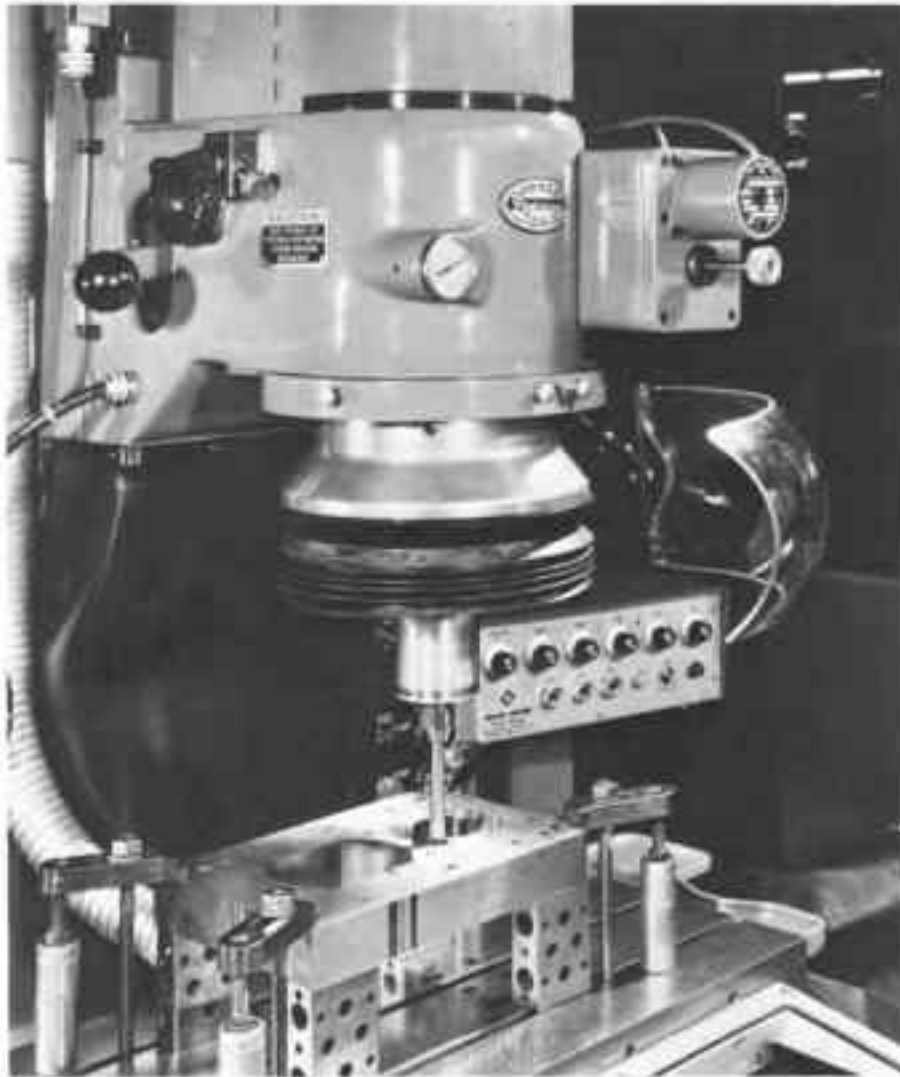
same time it is made to travel continuously in a circular path—called the planetary path—at a slow speed, thereby grinding a cylindrical surface inside of the hole. Simultaneously with these motions the grinding wheel is usually, but not always, made to reciprocate back and forth parallel to the axis of the hole in order to grind the entire wall of the hole. The axis of the ground hole will coincide with the axis of the main spindle. The size of the hole is determined by the outward radial position of the grinding wheel.

Except when grinding very small holes, the plunge grinding method is recommended for rough grinding when a large amount of stock must be removed, especially when grinding tough and abrasive resistant steels. From .010 to .040 inch (0.25 to 1.02 mm) stock can be removed in a single pass. In plunge grinding the reciprocating motion is not used; instead, a deep grinding cut is taken, as described above, while feeding the grinding wheel very slowly downward through the hole using the manual fine feed control. The grinding is done by the outer corner of the face of the wheel. When the abrasive in this region of the wheel gets dull it will break off, exposing sharp abrasive particles without altering the diameter of the wheel.

Finish grinding a hole to size is usually done by the reciprocating method. This method is sometimes also used for rough grinding, especially when the amount of stock to be removed is small. The length of the reciprocating stroke is adjusted according to the length of the hole, taking care that the grinding wheel is at all times in contact with the wall of the hole when grinding in order to prevent grinding a bell-mouth shape at the end of the hole. The hole is ground to size using the trial cut procedure. After a grinding cut has been taken in the hole, the grinding wheel

is withdrawn and the diameter of the hole is measured. The grinding wheel is then inserted back into the hole without making an outward adjustment. The required amount of outward radial adjustment of the grinding wheel is made slowly and carefully while the hole is being ground. Several trial cuts are usually necessary to finish a hole to size. Whenever the grinding wheel is withdrawn from the hole it should be allowed to reciprocate up and down a number of times without "outfeeding" the wheel to allow it to "spark out."

The jig grinder is also used to grind profile contours on punches, dies, and in molds. A typical contour grinding job is illustrated in Fig. 15-43, which shows a lamination die being ground. The profile of this die consists of a series of interconnected arcs. Each segment of the arc is ground by positioning the jig grinder table so that the center of the arc on the



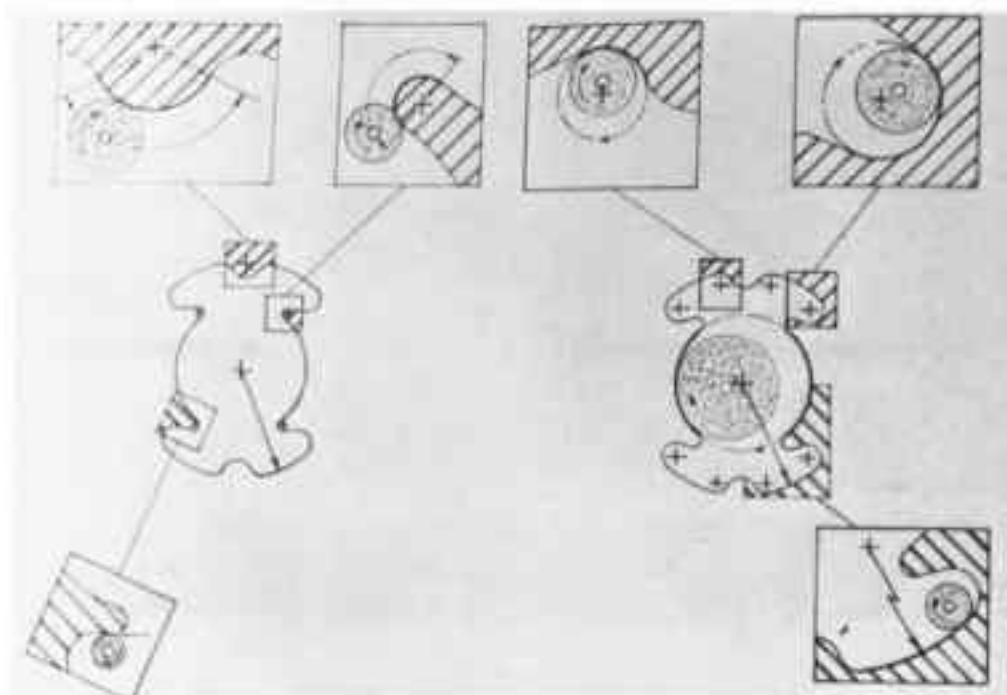
Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-43. Grinding a lamination die profile contour with a jig-grinder.

workpiece coincides with the axis of the main spindle. When in this position, the grinding wheel is offset radially the required distance and made to rotate along a planetary path (Fig. 15-42) by rotating the main spindle through the part of a revolution, or arc length, as required to grind the arc on the workpiece. The jig grinder in Fig. 15-43 has an attachment that rotates the main spindle by power through the exact arc length, or part of a revolution, required. On jig grinders that do not have this feature the main spindle is rotated manually. The complete contour grinding sequence necessary to finish this die is shown in Fig. 15-44. Often sharp internal corners are encountered in profile contour grinding, such as the intersection of the sides and ends of the blanking station of the lamination die shown in Fig. 15-45. Sharp corners such as these are ground with the slot grinding attachment shown in this illustration. In this case, the entire profile may be ground with this attachment. Many other attachments are available to increase the versatility of the jig grinder and to allow it to perform additional operations on punches, dies, and other precision parts.

Transfer of Tolerances

All of the dimensions that must be accurately held on precision machine and engine parts are usually given a tolerance. When such dimensions



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-44. Step-by-step procedure for grinding the contour profile of the lamination die.



Courtesy of the Moore Special Tool Co., Inc.

Fig. 15-45. Grinding internal profile of the blanking station of the lamination die using a slot-grinding attachment on a jig-grinder.

must be changed from the conventional system of dimensioning to coordinate dimensions, the tolerances must also be included. The tolerances of the dimensions on the coordinate system of dimensioning will not be the same as on the conventional system of dimensioning from which they were derived. The new tolerances on the coordinate system of dimensioning must be carefully calculated, otherwise serious errors will result. In most instances, changing from the conventional to the coordinate system of dimensioning will reduce the tolerances.

The principle of the transfer of tolerances can be stated as follows:

When transferring tolerances from the conventional dimensioning system to the coordinate dimensioning system, the sum of the tolerances of any pair of dimensions on the coordinate system must not exceed the tolerance of the dimension that they replace on the conventional system.

This principle can best be learned by following an example:

The $2.000 \pm .005$ dimension between two holes shown at A, in Fig. 15-46 is replaced by the $1.000 \pm .0005$ and the $3.000 \pm .0045$ -inch dimensions in the coordinate systems shown at C, in Fig. 15-46. The sum of the tolerances in view C; i.e. ($\pm .0005$ and $\pm .0045$) must not exceed $\pm .005$ -inch, otherwise the original tolerance of the $2.000 \pm .005$ dimension can be exceeded and the workpiece will not meet the original dimensional requirements. It is important to realize that the sum of the new tolerances on the coordinate dimensions may be less than the original tolerance which they replace; however, they must never exceed the original tolerance.

It is best to follow a set procedure in order to transfer dimensions and tolerances from a conventional to a coordinate dimensioning system.

This procedure is outlined below, and using the $2.125^{+.003}_{-.001}$ dimension in A, Fig. 15-46, as an example:

1. Where necessary, change the tolerances so that each dimension has an equal bilateral tolerance.

In order to maintain the original dimensional requirement, the corresponding dimension must also be changed. The $^{+.003}_{-.001}$ tolerance of the 2.125 dimension is changed to $\pm .002$, which is an equal bilateral tolerance. (Note that the total tolerance of .004 inch has not been changed.) The original dimensional requirements are maintained if the dimension is changed from 2.125 inches to 2.126 inches. The new dimension, $2.126 \pm .002$ is shown in view B. (Note that the dimension has not really changed, since in both cases, the limiting dimensions are 2.128 and 2.124 inches.)

2. Transfer the revised basic dimensions to the coordinate dimension system.

The original basic dimension was 2.125 inches, which was revised in Step 1 to 2.126 inches. In the coordinate system this dimension is replaced by the .8750- and the 3.0010-inches dimensions. (The 3.0010 dimension is obtained by adding $.8750 + 2.126$ inches.)

3. Transfer the tolerances.

The sum of the tolerances of the .8750 and the 3.0010 dimensions must not exceed the tolerance of the 2.126 dimension, or $\pm .002$ inch, which is the available tolerance. If the available tolerance is to be distributed equally, then each new dimension will have $\pm .0010$ tolerance. These dimensions will then read: $.8750 \pm .0010$ and $3.0010 \pm .0010$.

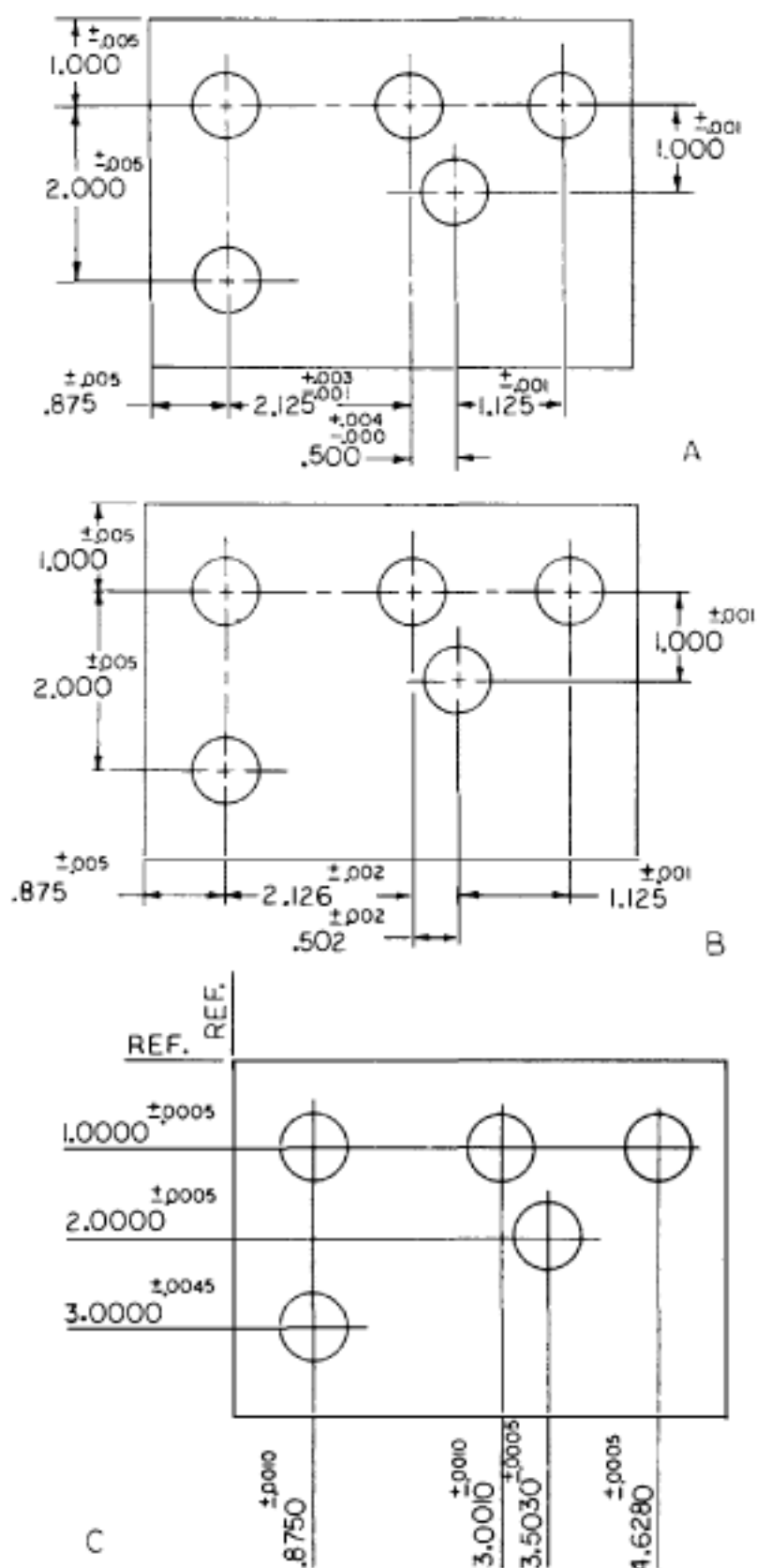


Fig. 15-46. A. Conventional dimensions—mixed tolerances; B. Conventional dimensions—all equal, bilateral tolerances; and C. Coordinate dimensions.

Sometimes the tolerance of a dimension on the coordinate-system drawing is affected by more than one dimensional requirement. In this case, the final tolerance used must be that which fulfills all of the requirements. For example, consider the $.502 \pm .002$ dimension in view B. It is replaced by the 3.0010 and the 3.5030 dimensions in view C. If the 3.0010 and 3.5030 dimensions are each given a $\pm .001$ tolerance, the sum of their tolerance would not exceed the original $\pm .002$ tolerance and, presumably, the requirements for the transfer of tolerances would be met. However, the 3.5030 dimension together with the 4.6280 dimension replaces the 1.125 dimension in view B, which has a tolerance of only $\pm .001$ inch. Thus, the sum of the tolerances on the 3.5030 and the 4.6280 dimensions cannot exceed $\pm .001$ inch which, when divided equally, amounts to $\pm .0005$ inch. The 3.5030-inch dimension must therefore be given the lesser tolerance of $\pm .0005$ inch.

For this reason, when transferring the tolerance it is usually best to start with the smallest tolerance. Also note that the sum of the tolerances replacing the $.502 \pm .002$ dimension is less than the $\pm .002$ inch. This is satisfactory since the sum of the new tolerances does not *exceed* the original tolerances.

When a tolerance is bound by a small tolerance, as in the case of the .502 dimension, it is sometimes possible to increase an adjacent tolerance on the coordinate dimension drawing. For example, the 2.000-inch dimension in view B, which has a tolerance of $\pm .005$ inch is replaced by a 1.0000- and a 3.0000-inch dimension on the coordinate drawing. Each of these dimensions could be given a tolerance of $\pm .0025$ inch; however, the tolerance of the 1.0000-inch dimension on the coordinate drawing is bound by the requirement of the $\pm .001$ tolerance of the 1.000 dimension shown at the right side, in view B. Thus, the 1.0000 dimension in view C, together with the 2.0000 dimension, must have a total dimension tolerance of $\pm .0010$ or $\pm .0005$ inch on the 1.0000-inch dimension. Since the sum of the tolerances of the 1.0000 and the 3.0000 dimensions (view C) can be $\pm .005$ inch, the tolerance of the 3.000 dimension can be increased to $\pm .0045$ inch.

The basis for the determination of the final tolerances for the coordinate dimensions (C, Fig. 15-46) is summarized below:

- $.8750 \pm .0010$: The $\pm .0010$ tolerance together with the $\pm .0010$ tolerance of the 3.0010 dimension is required to maintain the $\pm .002$ tolerance for the 2.126 dimension.
- $3.0010 \pm .0010$: See requirements for the $.8750 \pm .0010$ dimension given above.
- $3.5030 \pm .0005$: The $\pm .0005$ tolerance together with the $\pm .0005$ tolerance of the 4.6280 dimension is required to maintain the $\pm .001$ tolerance for the $1.125 \pm .001$ dimension.
- $4.6280 \pm .0005$: See requirements for the $3.5030 \pm .0005$ dimensions given above.
- $1.0000 \pm .0005$: The $\pm .0005$ tolerance together with the $\pm .0005$ toler-

ance of the 2.000 dimension is required to maintain the $\pm.001$ for the $1.000 \pm .001$ dimension given at the right, in view B.

$2.0000 \pm .0005$: See requirements for the $1.0000 \pm .0005$ dimension given above.

$3.0000 \pm .0045$: The .0045 tolerance, together with the $\pm.0005$ tolerance of the 1.0000 dimension in view C, is required to maintain the $\pm.005$ tolerance for the $2.000 \pm .005$ dimension in view B.

Metal Cutting Saws

Originally, metal-cutting saws were primarily used to cut off bar stock and structural steel, to length. This is still a very important area of application for metal cutting saws; however, the range of application has been extended to include shaping of metal into finished and semifinished machine components, tools, dies, jigs, and fixtures. This work is called band machining. It has been made possible by the development of the metal cutting band-saw blade along with the development of the vertical-contour band-sawing machine.

The Power Hacksaw

The power hacksaw is used to cut off bar stock, to length. The blade is held in a frame that is given a back and forth, or reciprocating, motion. The saw blade is forced into the work during the cutting stroke, while during the return stroke it is backed off from the work. This is done hydraulically on the saw shown in Fig. 16-1. The hydraulic pressure can be varied to suit the type of workpiece being cut. It has four cutting speeds (35, 70, 100, and 140 spm) which enable it to cut a variety of materials from aluminum to tough alloy steels. A coolant reservoir and a coolant pump located in the cabinet base provide an ample supply of cutting fluid at the saw blade. A fully automatic, power hacksaw is shown in Fig. 16-2. The operator merely loads the stock on the machine and sets it into operation by pushing the start button. After each cut has been completed, the traveling carriage, located on the stock rack, feeds the bar stock to the required position for the next cut. The hydraulically actuated vise then clamps the work and the saw begins to cut. An overhead clamp enables the work to be bundled, thereby making it possible to cut off two or more bars of stock during one cut. This machine can also be operated manually for short runs or single cuts.

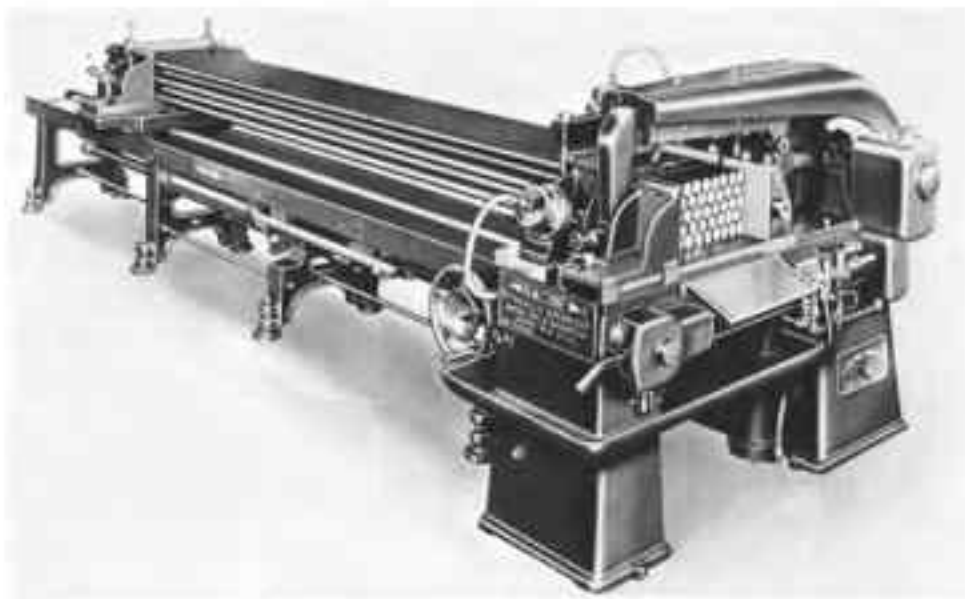
Most power hacksaw blades are made from high-speed steel. Some blades have hardened high-speed steel teeth which are welded to a tough steel back. The size of the hacksaw blade teeth is dependent on the number of teeth per inch; fewer teeth per inch will result in a larger tooth. The number of teeth per inch to be used depends upon the hardness of the material being cut and upon the size and shape of the workpiece. The power hacksaw blade to be used in cutting mild materials in large sections should have 4 to 6 teeth per inch, while harder materials in large



Courtesy of Racine Hydraulics & Machinery Inc.

Fig. 16-1. Power hacksawing machine.

sections should have 6 to 10 teeth per inch. When sawing pipes or tubing, the saw blade should have at least 14 teeth per inch. Structural steel shapes such as angles, T's, and channels require saw blades having 6 to



Courtesy of Racine Hydraulics & Machinery Inc.

Fig. 16-2. Automatic power hacksaw provided with overhead clamp to cut off bar stock in bundles.

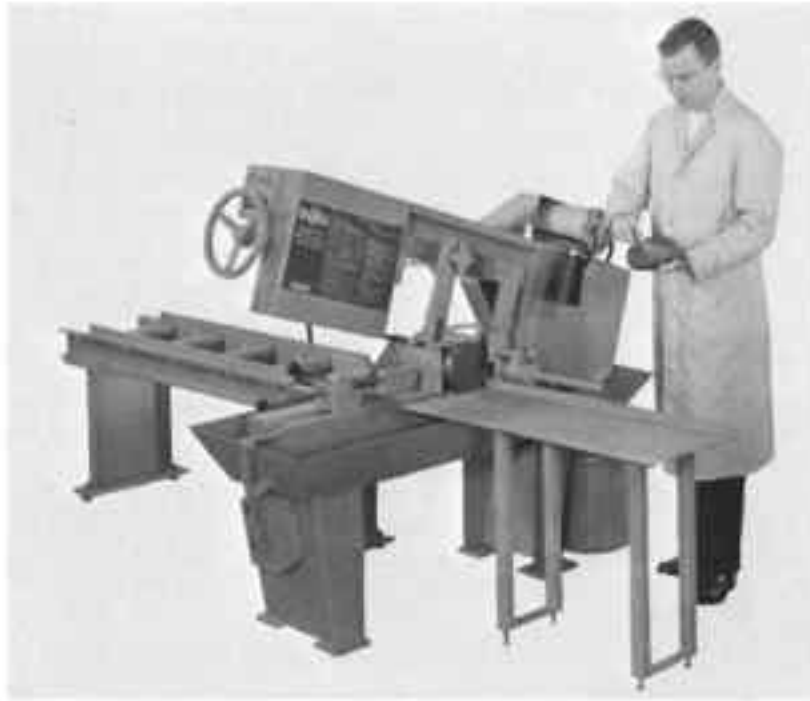
14 teeth per inch depending on the thickness of the thinnest section and the way that the work is clamped in the hacksaw vise. Recommendations for cutting speed, teeth per inch, and feeding forces are given in Table 16-1.

Table 16-1. Recommended Saw-blade Pitch, Cutting Speed, and Feed for Power Hacksaw Operation

Material	Saw-blade Teeth, per inch	Cutting Speed, fpm	Feed Weight or Force, lbs.
Steel (125-180 Bhn)	4 to 6	120	150
Steel (181-250 Bhn)	6 to 10	100	125
Ferritic Stainless Steel	4 to 6	110	125
Austenitic Stainless Steel	4 to 6	80	125
Martensitic Stainless Steel	4 to 6	60	125
Carbon Tool Steel	6 to 10	75	125
High-Speed Steel	6 to 10	60	125
Structural Steel	6 to 10	120	125
Steel Tubing	14	120	60
Copper	6 to 10	130	125
Brass Tubing	14	130	60
Soft Brass	6 to 10	150	60
Hard Brass	6 to 10	110	60
Bronze	6 to 10	110	125
Nickel Silver	6 to 10	60	150
Manganese Bronze	6 to 10	80	60
Cast Iron	6 to 10	120	125
Iron Pipe	10 to 14	120	125
Aluminum Alloy	4 to 6	150	60
Aluminum-Heat Treated	4 to 10	120	60
Aluminum-Soft	4 to 6	180	60

Band Type Cutoff Saw

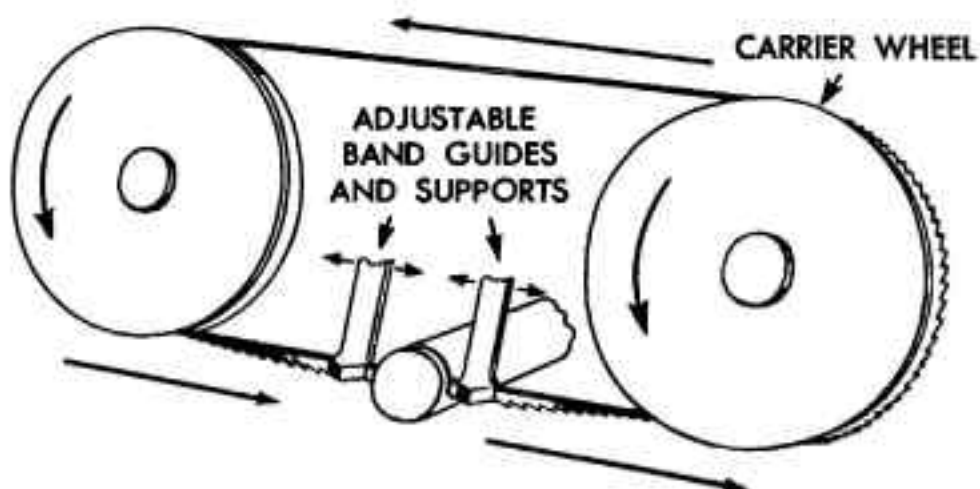
The band type cutoff saw, Fig. 16-3, utilizes a metal cutting band-saw blade to cut off bar stock and structural-steel sections, to length. The band-saw blades for these saws can be obtained already welded together into a loop of suitable size for each machine. The principle of operation of the band type cutoff saw is shown in Fig. 16-4. The saw blade revolves around the drive wheel and the idler wheel. The blade is twisted by the saw guides to bring the teeth into a vertical position in order to cut through the work. The guides also support the blade near the work, thereby reducing the tendency of the blade to bend away from the work. The blade can be made to cut at different speeds in order to cut different materials. One advantage of the band type cutoff saw is that the saw blade cuts continuously. Thus, the non-cutting time during the return stroke of the power hacksaw is saved. Furthermore, since it is not necessary to reverse the direction of the saw blade at the end of each stroke,



Courtesy of the DoALL Company

Fig. 16-3. Band type cutoff sawing machine.

higher cutting speeds can be used on band type cutoff saws where the characteristics of the material being cut will allow their use. A continuously uniform feeding pressure is kept on the saw blade which is adjusted to suit the material being cut. Angular cuts can be taken on some band cutoff saws. Production band cutoff saws can be obtained that will handle bundles of stock and that will operate completely automatically.



Courtesy of the DoALL Company

Fig. 16-4. Operating principle of the cutoff band saw.

Vertical-Contour Band-Sawing Machine

A vertical-contour band-sawing machine is illustrated in Fig. 16-5 with cabinet cover removed in one view, and all parts of the machine named. Although there are different makes and models of these machines their basic features are similar. The base supports the column and houses the hydraulic system for actuating the table feed, the electrical controls, the electric-drive motor, the transmission, the drive wheel, and the air compressor. The column encloses the return route of the saw blade. It supports the head, and mounted on its face are the band-saw speed-tachometer, the blade shear, the table feed-rate and force control, the butt welder for welding blades, the grinder, and the on-off push button. The head of the machine houses the idler wheel and its adjusting mechanism, and it supports the saw guidepost.

The band-saw blade is driven by the lower drive wheel. The blade is supported by and runs the idler wheel located in the head. It must be held between these wheels with a predetermined amount of tension which is dependent upon the size of the blade and the material from which it is made. The tension is adjusted by turning the wheel-elevating screw. Many band saws have a tension indicator which is calibrated to read the recommended tension. The band must then be checked for tracking and alignment with the saw guides. The idler wheel in the head can be tilted to make this adjustment by releasing the lock knob and turning the adjusting knob (located in the center of this wheel), while turning the wheels slowly by hand. Some wheels are crowned, while others have a flange against which the blade runs. Crowned wheels are adjusted until the center of the blade runs on the crown of the wheel. On flanged wheels the blade is adjusted to ride against the flange. The blade must also be adjusted to the back-up thrust bearings which are located behind the saw guides, Fig. 16-6. The wheel-adjusting knob should be turned until the back edge of the saw blade just touches the back-up thrust bearing with no noticeable gap between them.

There are two sets of saw guides; one set is located below the table and the other set is attached to the guidepost above the table. In Fig. 16-6, the table has been removed to show both sets of saw guides. Saw guides are made in different widths to match the width of the saw blade. The guides should be adjusted until there is about .002-inch clearance between the guide and the saw blade. The upper saw guide can be adjusted vertically. It should always be positioned as close to the work as possible in order to provide the maximum support to the saw blade. When sawing at high speeds (2,000 fpm, or more), roller type saw guides must be used.

Most vertical-contour band-sawing machines are equipped with an infinitely variable speed drive. The speeds are adjusted by simply turning a handwheel. The recommended cutting speeds for different materials are given in Table 16-2.

The saw table on most vertical band-sawing machines can be tilted 10- to 15-degrees to the left, and 45 degrees to the right. Some machines

are equipped with a stationary table where the work must be fed into the saw blade by hand or by a weight-actuated feeding arrangement. The weight is attached to a cable system which pulls the work into the saw blade. Many contour band-sawing machines are equipped with a hydraulic table feed. The workpiece is clamped to the table and the table feed, actuated by a hydraulic cylinder, feeds the work into the saw blade. The hydraulic table feed has two characteristics, the *feed rate* and the *feeding*

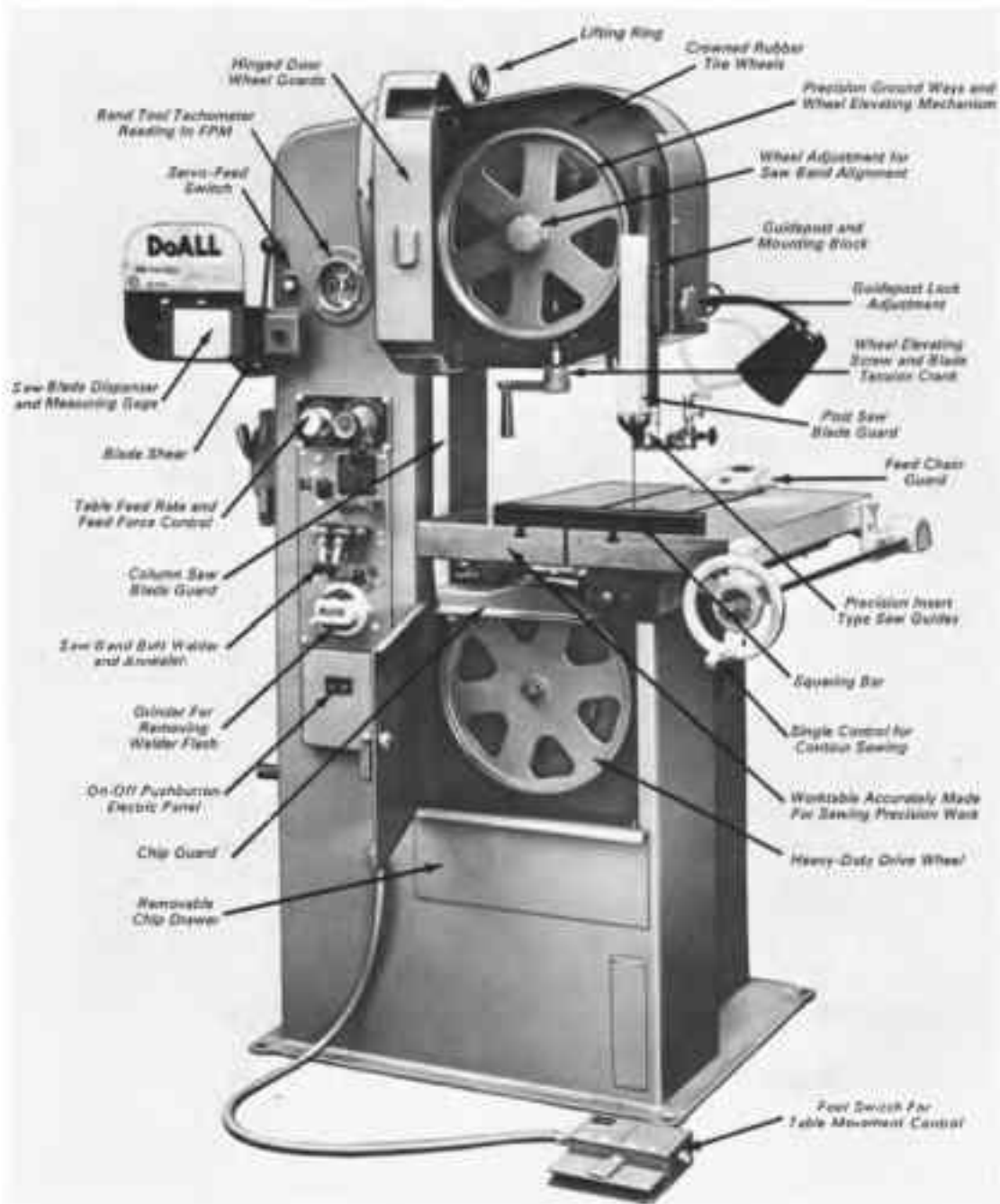
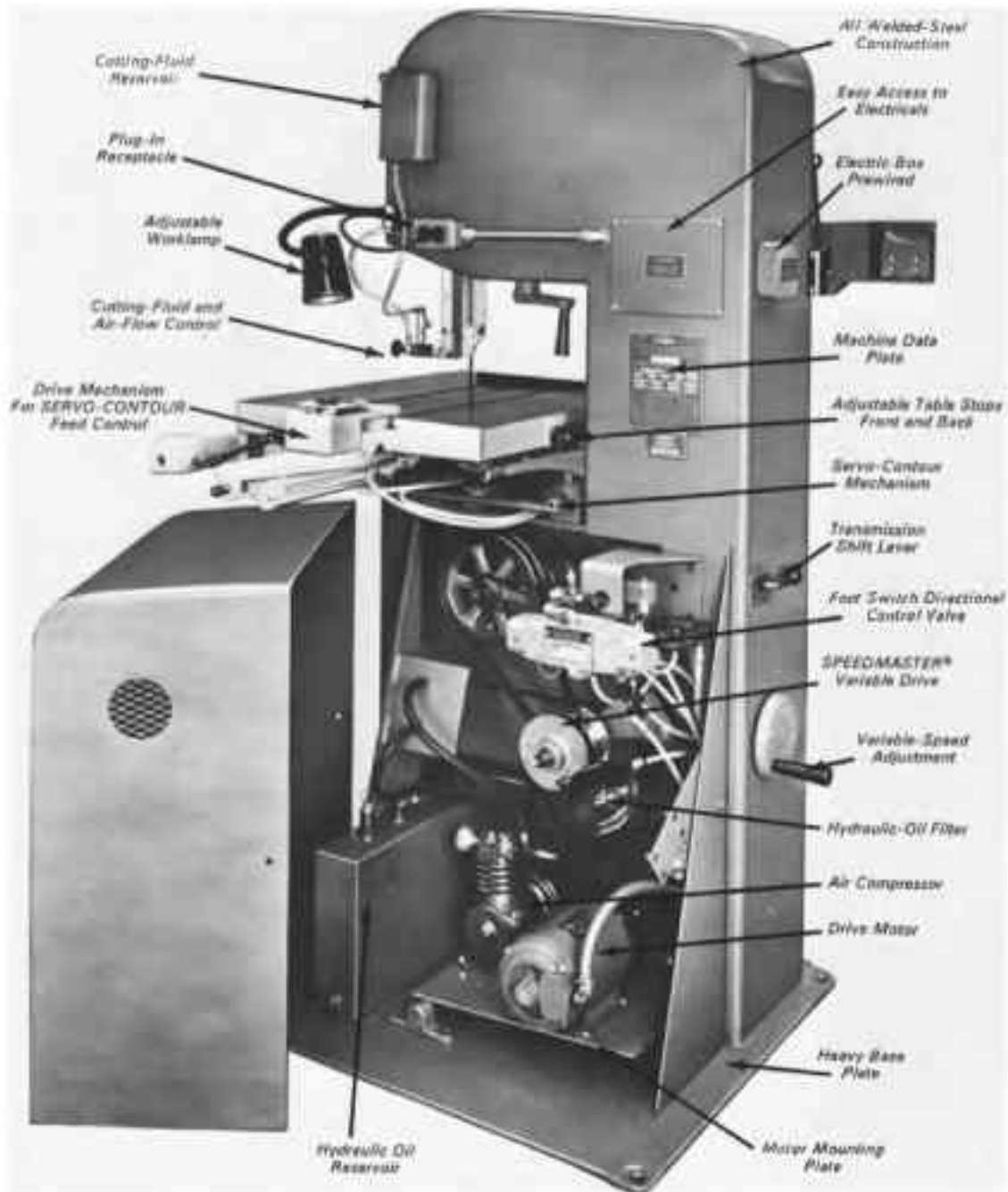


Fig. 16-5. Vertical-contour band-sawing machine.

pressure, which are independent of each other and are adjusted separately. Their purpose should be understood.

The feed rate determines how fast the table will feed. It is controlled by a hydraulic flow-control valve which determines the rate of flow of the oil into the table feed-cylinder.

The feeding pressure determines the maximum force that can be exerted by the workpiece against the saw blade before the table will stall. In-

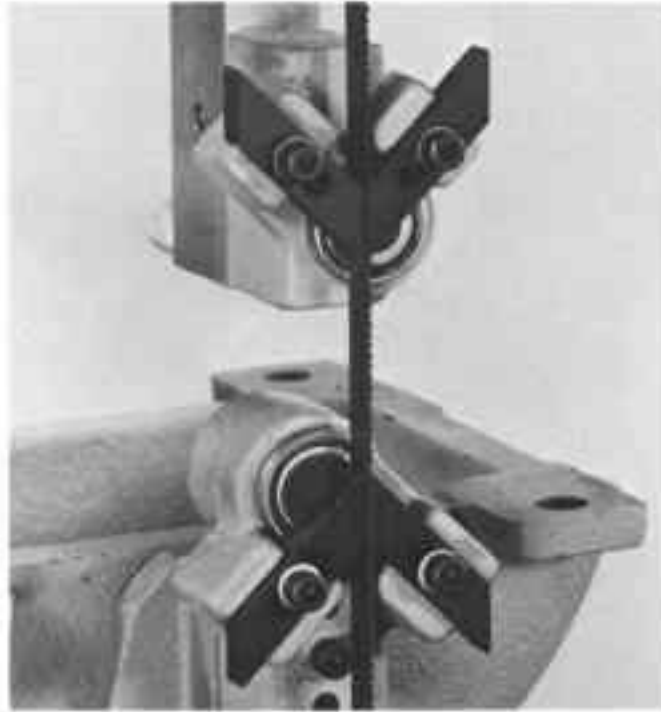


Courtesy of the DoALL Company

(Left) Front view; (Right) Rear view.

Table 16-2. Recommended Cutting Speeds for Band Sawing

Material	Carbon Tool-Steel, High-Speed Steel Work Thickness, inches						
	$\frac{1}{2}$	1	3	$\frac{1}{2}$	1	3	6+
	Cutting Speed, fpm						
Steel (AISI Numbers Given): 1108, 1132, 1212, 1213, 1137	200 to 225	175 to 200	140 to 175	275 to 350	225 to 275	200 to 225	150 to 200
Steel (AISI Numbers Given): 1151, 1008, 1013, 1020	150 to 175	125 to 150	100 to 125	250 to 300	200 to 250	150 to 200	125 to 150
Steel (AISI Numbers Given): 1035, 1320, 3115, 3130, 3135, 4017, 4130, 4317, 4608, 4621	125 to 150	100 to 125	75 to 100	200 to 250	175 to 200	125 to 175	90 to 125
Steel (AISI Numbers Given): 1040, 1060, 1345, 2330, 2345, 2512, 2517, 3150, 3310, 3315, 4068, 4042, 4047, 4140, 4150, 4340, 4350, 4640, 4812, 4820, 5045, 5120, 5135, 5140, 5168, 52100, 50100, 6117, 6120, 6145, 6152, 8615, 8715, 9255, 9260	100 to 125	80 to 100	50 to 80	175 to 225	140 to 175	115 to 140	75 to 115
Steel (AISI Numbers Given): 1090, 1095, 8645, 8715, 8647, 8660, 9310, 9317, 9347, 9445, 9447, 9763, 9840, 9850	80 to 100	50 to 80	50 to 70	200 to 225	150 to 200	125 to 150	90 to 125
Tool Steel: A-2, O-1, O-2, O-6	125 to 150	100 to 125	80 to 100	175 to 200	150 to 175	100 to 125	80 to 100
Tool Steel: M-1	100 to 125	75 to 100	50 to 75	130 to 160	100 to 130	75 to 100	50 to 75
Tool Steel: S-1, S-2, S-5, D-2, D-3, T-1, T-2, T-4, T-5, M-2, M-6	70 to 90	50 to 70	40 to 50	100 to 125	80 to 100	60 to 80	50 to 60
Tool Steel: H-12, H-13, H-21, H-22, H-24, H-25	60 to 75	50 to 60	40 to 50	160 to 200	125 to 150	100 to 120	80 to 90
Tool Steel: M-4, M-10, M-15	60 to 70	50 to 60	40 to 50	80 to 90	60 to 80	50 to 60	40 to 50



Courtesy of the DoALL Company

Fig. 16-6. Precision, insert type saw guides and back-up thrust bearings.
Table removed to show upper and lower guides.

creasing the pressure setting will not increase the feed rate. The pressure setting does not determine the pressure that will be used to saw the workpiece. It will determine the pressure inside the table feed-cylinder at which the table will stall, as stated above.

The feed rate and the pressure that should be used depend upon many factors that, in combination, are unique for each job. Among the factors that must be considered in selecting the feed rate and pressure settings are: type of work material, thickness of the workpiece, type of saw blade, pitch of the saw blade, width and thickness of the saw blade, sharpness of the saw-blade teeth, surface finish required on the work, and the distance between the upper and lower saw guides.

Saw Blades

Most power hacksaw and band-saw blades are made from carbon alloy tool steel or from high-speed steel. Less commonly used are band-saw blades with small, cemented-carbide tips brazed on the end of the teeth. Band-saw blades are also made with industrial diamonds coating the edge of the blade or with either aluminum oxide or silicon-carbide abrasives coating the edges. The cemented-carbide tipped blades are used to saw difficult-to-machine materials such as: pure tungsten, beryllium, high-temperature alloys, foamed glass, etc. The abrasive-coated steel bands are used for deburring, corner breaking, and for rough and finish polishing of metals. They are also used to cut hardened steel, exotic

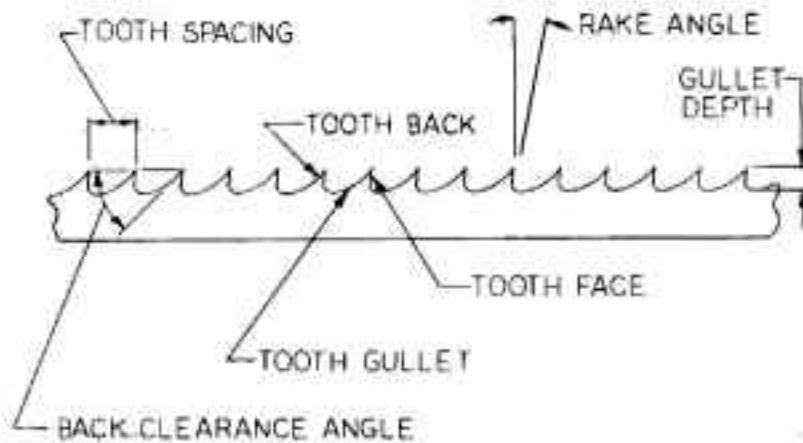


Fig. 16-7. Names of the parts of a saw blade.

alloys, and abrasive nonmetallic materials. Special steel blades are available for slicing such materials as cork and foam rubber.

The majority of the work on band saws, however, is done with carbon-alloy steel or high-speed steel saw blades. The recommended cutting speeds given in Table 16-2 are for sawing different grades of steel with carbon-tool steel or high-speed steel saw blades. These recommendations should only be used as a starting point. Conditions at the job can make deviations from these recommendations necessary.

The names of the parts of a saw blade are given in Fig. 16-7. There are four, basic tooth forms which are shown in Fig. 16-8. The precision, or regular-tooth, form is the most common. It has a zero-degree rake angle and a 30-degree back-clearance angle. The gullet is deep and has a

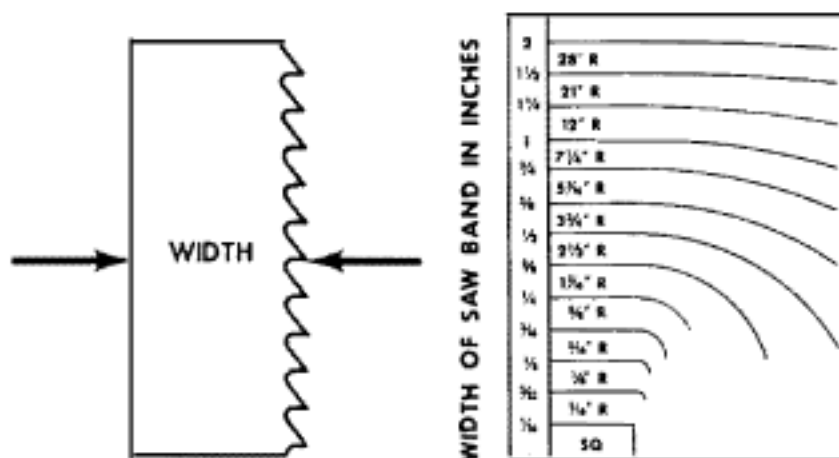


Courtesy of the DoALL Company

Fig. 16-8. The basic band-saw blade-tooth forms. The names of the tooth forms from top to bottom are: butress, claw tooth, precision, and tungsten carbide.

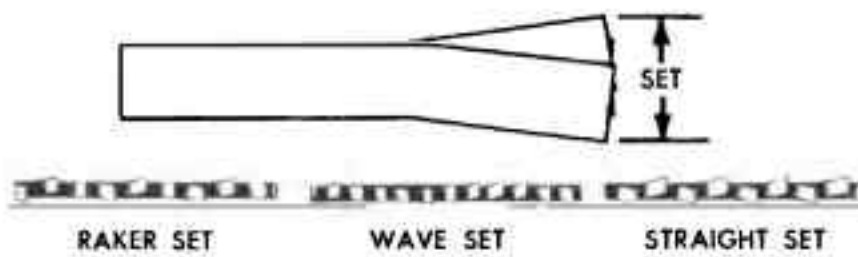
smooth radius at the bottom providing an ample chip-capacity for most sawing operations. It cuts accurately and can produce a good surface finish. The precision-tooth form is the most versatile and is recommended for contour work and for straight cuts. The buttress, or skip tooth, tooth form is the same as the precision-tooth form given except that the gullet has been lengthened to handle larger chip loads. It is primarily recommended for taking deep cuts in nonferrous metals, wood, and plastics. The claw-tooth form has a positive rake angle and a lengthened gullet similar to the buttress-tooth form. It is recommended for cutting all metals, wood, and plastics and is particularly recommended for thick work-sections. The positive rake angle requires a minimum feed pressure to penetrate into the work, thereby extending the tool life of the blade. The carbide blade has small carbide tips brazed on the end which are used for cutting difficult-to-cut materials.

There are five characteristics that must be specified in order to select a saw blade. These characteristics are: the tooth form, pitch, width, gage, and set. The pitch is the number of teeth per inch. The pitch is selected primarily on the basis of the thickness of the workpiece to be cut. At least two teeth should always be in contact with the work; however, it is preferable to have even more teeth in contact. Other factors, such as work material and the finish desired must also be considered in making the final selection of the pitch. For average work conditions the following pitches are recommended: $\frac{1}{4}$ - to $\frac{1}{2}$ -inch thick, 8-10 pitch; $\frac{1}{2}$ - to 1-inch thick, 8 pitch; 1- to 3-inches thick, 6 pitch; 3- to 6-inches thick, 4 pitch; 6 inches and over, 3 pitch claw-tooth form. The above pitches recommended should have the precision-tooth form. The width of the blade is the distance from the tip of the tooth to the back of the blade. The width of the blade is selected on the basis of the minimum radius that is to be sawed. A radius chart for selecting the blade width is shown in Fig. 16-9. A wide blade is most accurate for cutting a straight line. The gage of the blade is the thickness of the band expressed in inches. A thicker blade has



Courtesy of the DoALL Company

Fig. 16-9. Radius chart for selecting band-saw width.



Courtesy of the DoALL Company

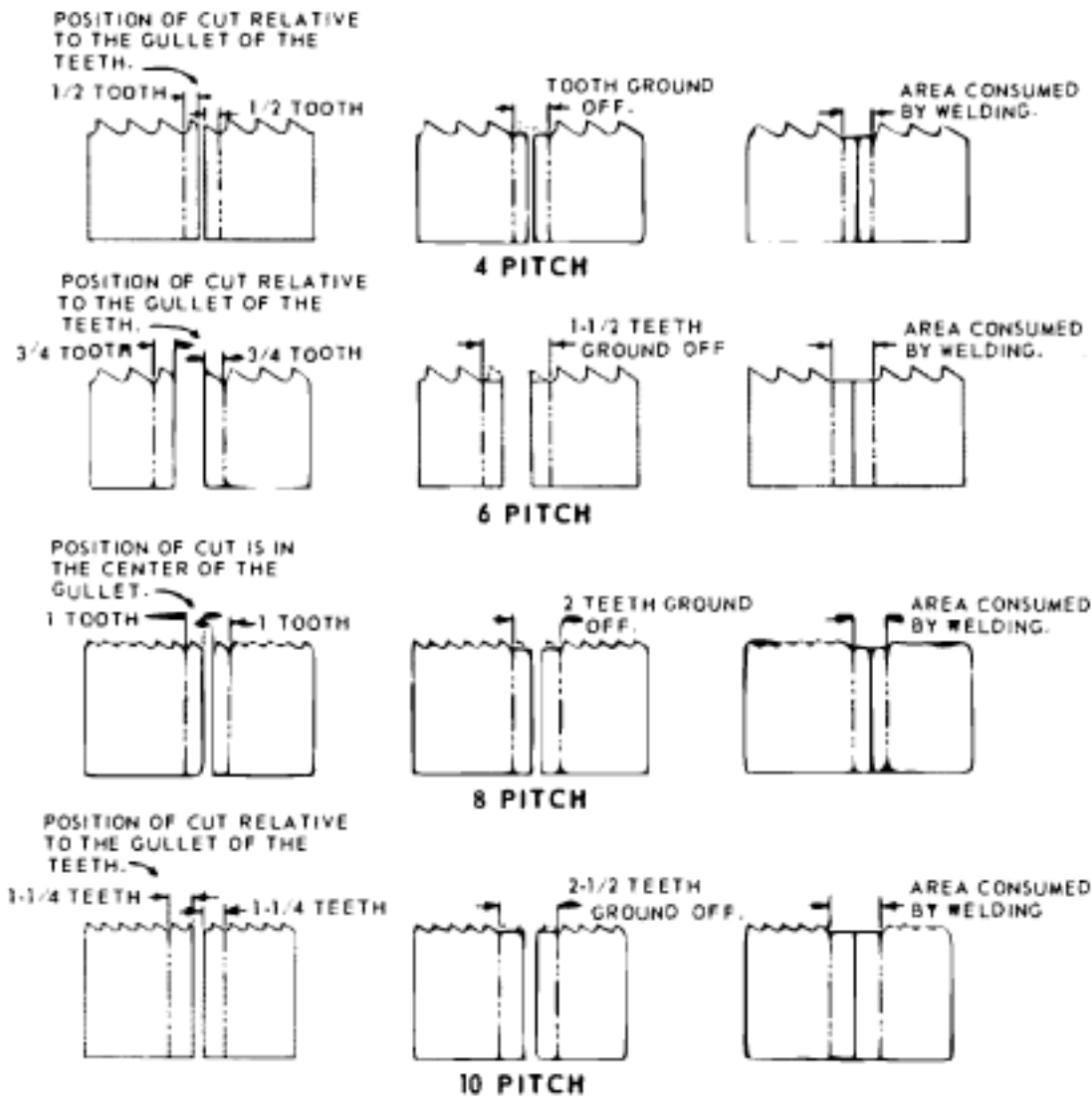
Fig. 16-10. Basic band-saw tooth patterns. (Left) Wavy set. (Center) Straight set. (Right) Raker set.

a greater beam strength, or resistance to bending. When the width of the blade is limited by the machine capacity or by the radius to be cut, the beam strength can be increased by using the same width blade but in a heavier gage. The heavier gages also have more resistance to side displacement and are recommended for sawing large workpieces where extreme accuracy is required. Set is the total amount that the teeth are offset (Fig. 16-10), to provide side clearance for the back of the band. The set of the saw blade enables it to cut contours. The width of the kerf, or slot, cut by



Courtesy of the DoALL Company

Fig. 16-11. Grinding end of band-saw blade in preparation for welding. Note saw teeth on opposite sides.



Courtesy of the DoALL Company

Fig. 16-12. Preparation of teeth prior to welding, to insure correct tooth-spacing on welded blade.

the blade is greater than the gage which allows the back of the saw blade to turn. There are three types of set patterns; raker, straight, and wave. The raker set is the most common. It has one tooth offset to the right, one to the left, and the next one remains straight. It is recommended for work of constant cross section, from thin to extra heavy, such as bar stock, forgings, and die blocks. The wave set has a group of teeth offset to the left and another group offset to the right to form a wave pattern. It is recommended for work having a varying cross section, such as pipe, tubing, angles, and structural sections. The straight set has every other tooth alternating from left to right. It is used for free-cutting materials such as aluminum or wood.

The saw blade used on vertical-contour band-sawing machines is obtained in coils and must be welded together to form a loop. The band

may be cut to length on the blade shear (which is attached to the machine) or with tin snips. When snips are used, the ends of the blade must be ground to make certain that they will fit together perfectly. The ends are brought together with the teeth on opposite sides, as shown in Fig. 16-11, and the ends are ground. It is not necessary to grind the ends perfectly square. When the blade is ground with the ends held as in Fig. 16-11 they will match perfectly when brought together to form the loop regardless of the angle to which they are ground. Some of the teeth must be ground off from all blades having a pitch from 4 to 10 so that the tooth spacing will be uniform after the weld has been made. About $\frac{1}{4}$ inch of metal will be consumed during the welding process and this loss of material must be taken into account. Instructions for grinding are given in Fig. 16-12. This grinding operation must be done with great care in order to grind the right number of teeth, or portion of a tooth, and not to grind below the gullet which would weaken the blade.

After the blade has been properly prepared for welding it is inserted in the butt welder, Fig. 16-13, with the teeth facing out and with the back of the blade firmly seated against the back alignment surface of the welder. One jaw of the welder is adjusted to bring a pressure to bear on the two ends of the blade that are in contact. The amount of this pressure and the voltage setting of the welder are obtained on charts that are furnished with each machine. If the saw blade is to be used to make an inside cut, as in Fig. 16-13, make certain that the blade passes through the hole in the workpiece before the weld is made. The weld is made auto-



Courtesy of the DeALL Company

Fig. 16-13. Welding a band-saw blade together in preparation for making an inside cut on the part.



Courtesy of the DoALL Company

Fig. 16-14. Example of a perfect hand-saw-blade weld.

matically when the welding switch is pressed. The weld is made by passing a low-voltage high-amperage current through the welder jaws and into the blade. The current flows toward the ends of the blade where the resistance to the current flow causes the metal in this area to heat up. When this happens, the metal in the joint becomes soft and flows together. The movable jaw of the welder causes the metal in the weld to compress together and form a flash. When the weld is completed it will be very hard and very brittle. Before it can be used it must be annealed and the flash must be removed. The welding flash can be removed on the grinder. Care should be exercised to avoid grinding the blade too thin and thereby reducing its strength. The weld is annealed by clamping the blade in the jaws of the butt welder. The welder is adjusted to anneal and heat is applied by pushing in or jogging the annealing switch until the weld reaches a dull cherry-red color. Avoid overheating the weld as this could well re-harden it. Before the blade is placed in the machine it should be inspected visually. One example of a perfect weld is shown in Fig. 16-14.

Band Machining

The basic approach to band machining is similar to working on other machine tools. The following steps should be taken: 1. analyze the job; 2. make a layout; 3. set up the machine; 4. cut the work to size on the machine; and 5. inspect the work. The job is begun by analyzing how it can be done most effectively. The next step is to make a layout. In the case of band machining this must be done carefully and accurately because the final results can be no better than the layout. The machine setup involves installing the correct blade, and setting the machine to cut at the correct cutting speed. If a power feed is to be used the workpiece will have to be held in place or clamped to the table, and the correct feed rate and pressure settings made. The part is cut by feeding the workpiece past the blade and keeping as close as possible to the layout line.

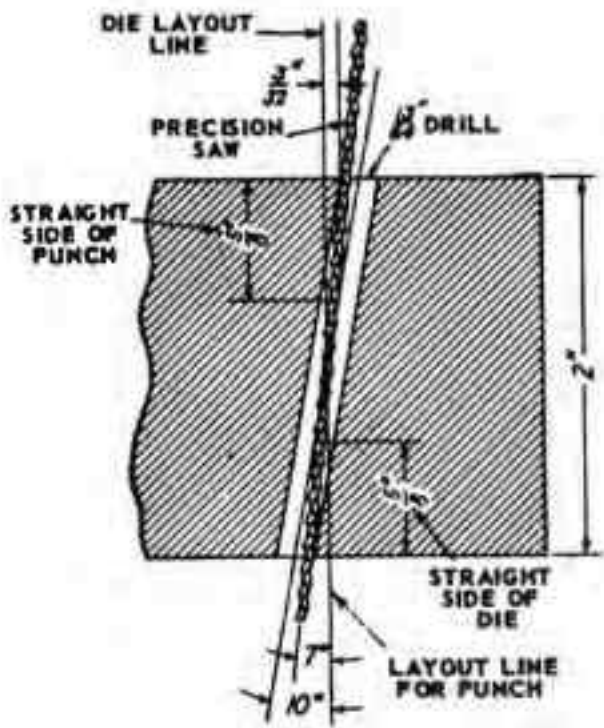
Finally, the work is inspected in order to make certain that the results are as specified by the part drawing.

The workpiece is frequently fed into the cut by hand. It can be held directly in the hand as shown in Fig. 16-15. This illustration shows a blanking punch and die, being made in one operation. The job is started by first laying out the die. A hole is drilled inside the layout of the die at an angle (Fig. 16-16), so that it intersects the location of the layout line at the center of the two-inch thickness and comes through the die block on the other side, at a location that is outside the layout line of the punch. A narrow saw blade is passed through the hole and welded together. The saw table is set at the required angle for the saw cut (7 degrees, in this case) and the punch is cut out by sawing just inside the die layout-line and just outside the punch layout-line (Fig. 16-16). Thus, there is enough material on the punch and the die so that they can be filed to fit together with the required amount of clearance. Several punches and dies finished by this method are shown in Fig. 16-17. The filing of the punch and the die can also be done on the vertical-contour band saw, using a file band. This is a quick and economical method of making a short-run trimming, blanking or shaving die. It saves material because the slug that is removed from the center is made into the punch. If this method of making a punch and a die from one part is to be used, the part cannot have any sharp corners or radii. In other words, the saw must be able to cut continuously around the die without interruption.



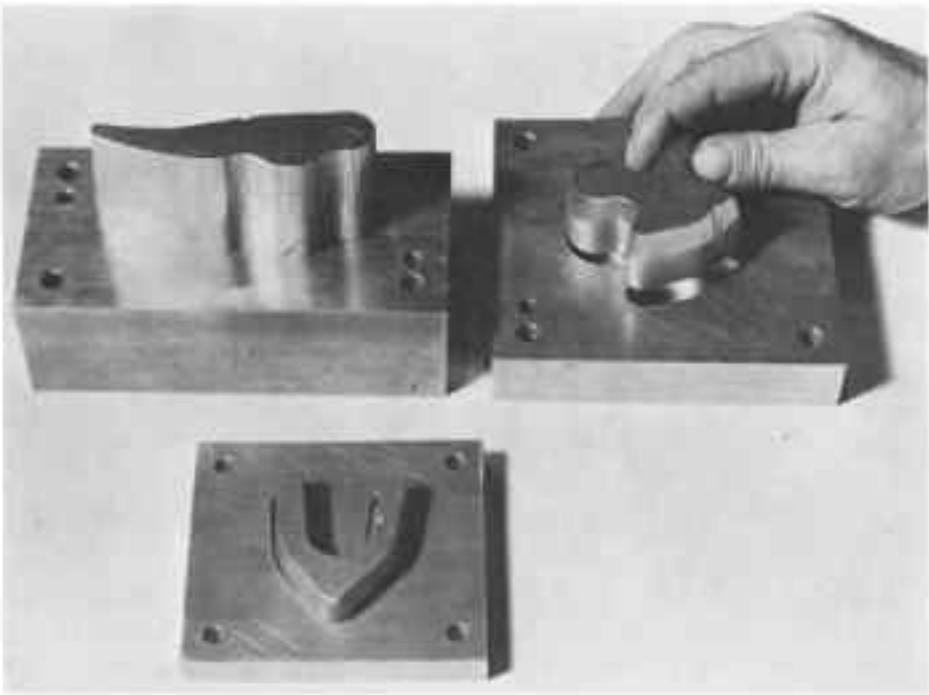
Courtesy of the DeALL Company

Fig. 16-15. Sawing a blanking punch and die in one operation, with the work table tilted.



Courtesy of the DoALL Company

Fig. 16-16. Diagram showing the angles of the starting hole and the saw band in 2-inch material when making a blanking punch and die in one operation.



Courtesy of the DoALL Company

Fig. 16-17. Several finished punches and dies made on the vertical-contour band saw. Note remainder of starting hole on punch in upper-right view.

The dimensions necessary to design and make a die by this method are given in Table 16-3.

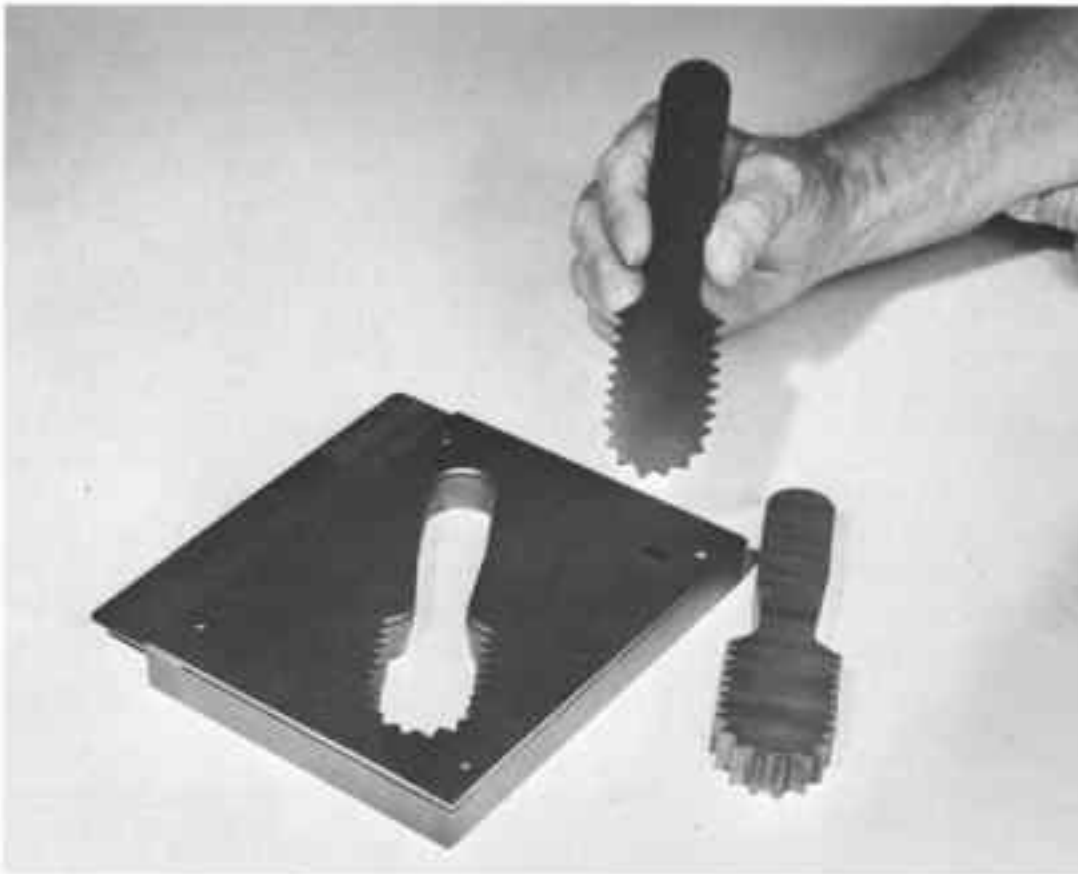
Table 16-3. Basic Dimensions for Making Punch and Die, by Method Shown in Fig. 16-16

Die Thickness, inches	Angle of Saw Starting Hole, degrees	Angle for Saw Cut, degrees	Distance from Die Layout Line to Center of Saw Kerf, inches	Distance from Die Layout Line to Center of Starting Hole, inches	Diameter of Drill, inches	Width of Starting Saw, inches	Amount of Straight Sides on Punch and Die
$\frac{1}{2}$	21	18	$\frac{5}{64}$	$\frac{3}{32}$	$\frac{1}{8}$	$\frac{3}{32}$	$\frac{3}{16}$
$\frac{3}{4}$	18	15	$\frac{3}{32}$	$\frac{1}{8}$	$\frac{1}{8}$	$\frac{3}{32}$	$\frac{9}{32}$
1	14	11	$\frac{3}{32}$	$\frac{1}{8}$	$\frac{9}{64}$	$\frac{1}{8}$	$\frac{3}{8}$
$1\frac{1}{4}$	12	9	$\frac{3}{32}$	$\frac{1}{8}$	$\frac{9}{64}$	$\frac{1}{8}$	$1\frac{5}{32}$
$1\frac{1}{2}$	11	8	$\frac{7}{64}$	$\frac{9}{64}$	$\frac{9}{64}$	$\frac{1}{8}$	$\frac{9}{16}$
2	10	7	$\frac{1}{8}$	$\frac{3}{16}$	$\frac{13}{64}$	$\frac{3}{16}$	$1\frac{3}{16}$
3	9	6	$\frac{5}{32}$	$\frac{1}{4}$	$\frac{17}{64}$	$\frac{1}{4}$	$1\frac{1}{8}$
4	8	6	$\frac{7}{32}$	$\frac{9}{32}$	$\frac{17}{64}$	$\frac{1}{4}$	$1\frac{5}{8}$
5	7	6	$\frac{1}{4}$	$\frac{5}{16}$	$\frac{17}{64}$	$\frac{1}{4}$	$2\frac{1}{4}$
6	6	5	$\frac{1}{4}$	$\frac{5}{16}$	$\frac{17}{64}$	$\frac{1}{4}$	$2\frac{1}{2}$
Die Thickness, inches	Minimum Outside Radius, inches		Minimum Inside Radius, inches		Saw Width, inches		
$\frac{1}{2}$	$\frac{9}{32}$		$\frac{1}{8}$		$\frac{3}{32}$		
$\frac{3}{4}$	$\frac{5}{16}$		$\frac{1}{8}$		$\frac{3}{32}$		
1	$\frac{7}{16}$		$\frac{1}{4}$		$\frac{1}{8}$		
$1\frac{1}{4}$	$\frac{7}{16}$		$\frac{1}{4}$		$\frac{1}{8}$		
$1\frac{1}{2}$	$1\frac{5}{32}$		$\frac{1}{4}$		$\frac{1}{8}$		
2	$1\frac{1}{16}$		$\frac{7}{16}$		$\frac{3}{16}$		
3	$\frac{7}{8}$		$\frac{9}{16}$		$\frac{1}{4}$		
4	$1\frac{1}{32}$		$\frac{9}{16}$		$\frac{1}{4}$		
5	$1\frac{1}{16}$		$\frac{9}{16}$		$\frac{1}{4}$		
6	$1\frac{1}{8}$		$\frac{9}{16}$		$\frac{1}{4}$		

Courtesy of the DoAll Company

Another fast method of making low-cost dies is called the Continental Process. This process is illustrated in Figs. 16-18 through 16-21. The die, punch, and a guide plate are made from thin sections of steel, on a vertical-contour band saw. To make the stamping, Fig. 16-18, the work material, which is a piece of sheet metal, is placed between the die and the guide plate. The punch is then placed in the guide plate, over the work. The ram of a punch press (or arbor press) forces the punch into the die, thereby cutting out the blank. Dies for drawing, bending, and a variety of other stamping operations can be made by this process.

In this case, the punch and the die are made separately. The first step is to carefully lay out the die. Four holes are drilled in the corners of the die block to receive dowel pins which are used to hold the guide plate. Corresponding holes are drilled in the guide plate using the holes in the die block to guide the drill. After the punch plate is assembled tightly against the die block, a hole is drilled in this assembly, inside of the layout lines. The band-saw blade is inserted through this hole (see Fig.



Courtesy of the DoALL Company

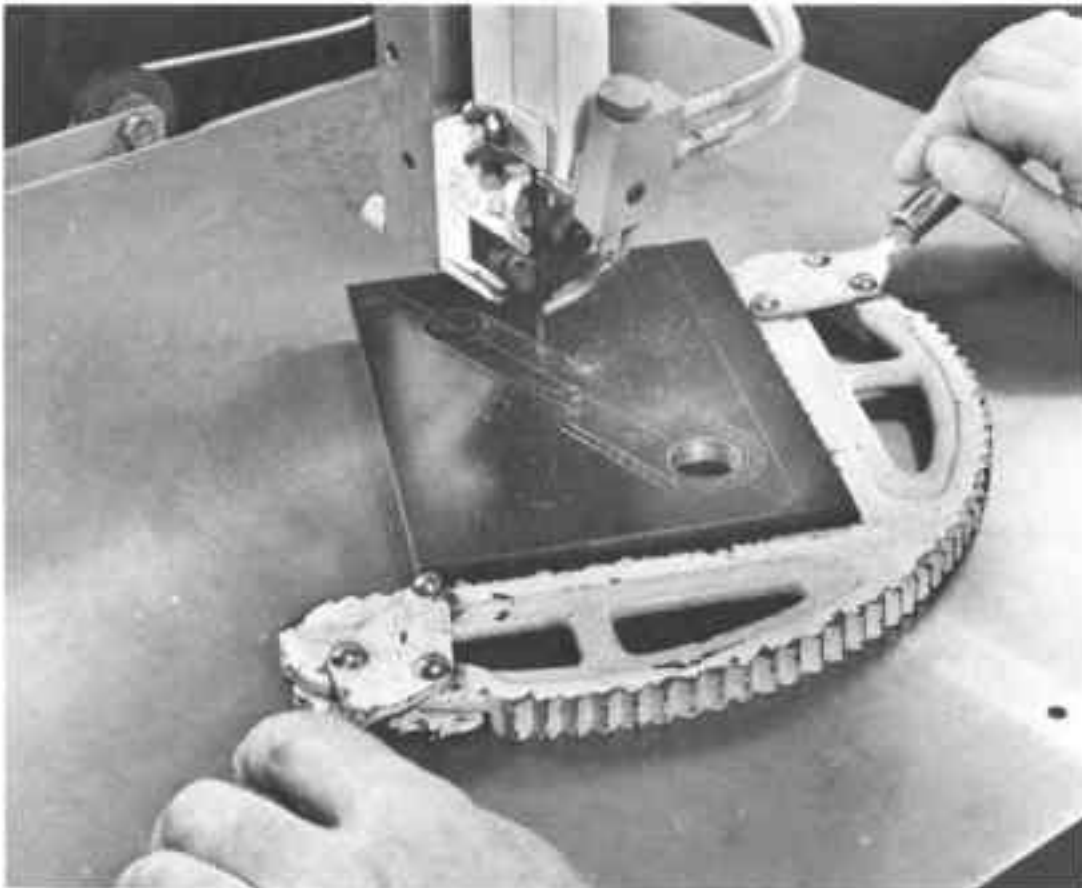
Fig. 16-18. Blanking punch and die made by the Continental Process on a vertical-contour band-sawing machine.

16-13) and welded together. The opening is then sawed out by manually guiding the work so that the saw cuts just inside the layout line allowing a small amount of metal to remain for filing, Fig. 16-19. The workpiece is held in a workjaw which makes it easier to guide the workpiece and follow the layout. After cutting, a band file (Fig. 16-20) is placed in the saw and the die block and guide plate are filed to size. The punch plate is disassembled from the die block and is used to lay out the punch. It is then reassembled to the die block with suitable shims between them. The shims should be just large enough to allow the work material to be inserted. The punch is then cut and filed to size on the band saw (see Fig. 16-21).

The cutting of contours can be facilitated by the attachment shown in Fig. 16-22. This attachment is called a servo-contour directional feed control. The operator follows the contour on the part by regulating the hydraulic table-feed rate during the cut and by turning the handwheel which rotates the parts through a movement of the roller chain. The roller chain can be placed directly around the work, as shown, or the work can be placed in a workjaw (Fig. 16-19) which would, in this case,

have the roller chain placed around it. Three-dimensional parts such as those shown in Fig. 16-23 can be made on the vertical-contour band-sawing machine. This machine can be provided with fixtures enabling it to do production work. An example is seen in Fig. 16-24 where the cap of a large Diesel engine connecting-rod forging is being separated. The fixture firmly holds the forging and accurately locates it with respect to the saw blade. The fixture is clamped to the table so that the cut is made using the power table-feed.

A great variety of additional work can be done on the vertical-contour band saw. Friction sawing, which utilizes the friction between the saw blade and the work to generate high temperatures a short distance ahead of the blade in the workpiece, is capable of cutting at extremely high feed rates. Metals that are impossible to cut in any other way can be cut by this method. Also, thin aircraft materials, distorted when sawed by conventional methods, can be sawed safely by friction sawing. The cutting speed of the saw blade for friction sawing is between 6,000 and 15,000 fpm. Friction sawing is possible because each saw tooth is in contact with the work for only a fraction of a second and there is too little time



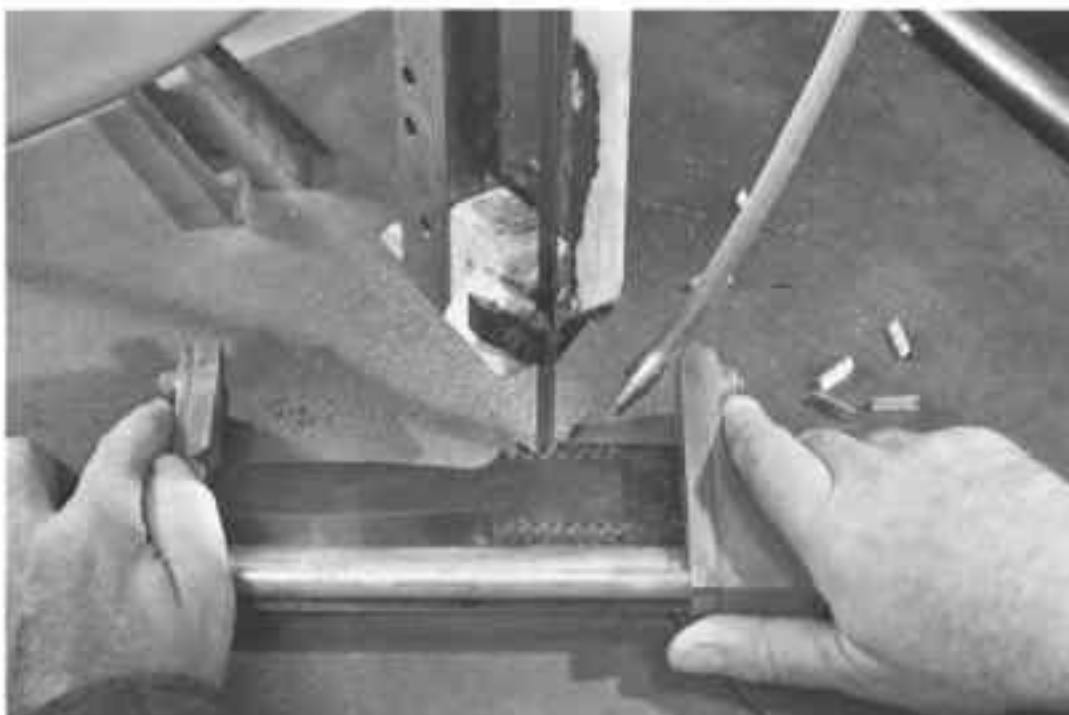
Courtesy of the DoALL Company

Fig. 16-19. Sawing contour of die to layout line using work jaw to feed the workpiece.



Courtesy of the DeALL Company

Fig. 16-20. Filing the block of the die using a file band.



Courtesy of the DeALL Company

Fig. 16-21. Sawing the contour of a punch using a flat vise to hold and feed the workpiece.



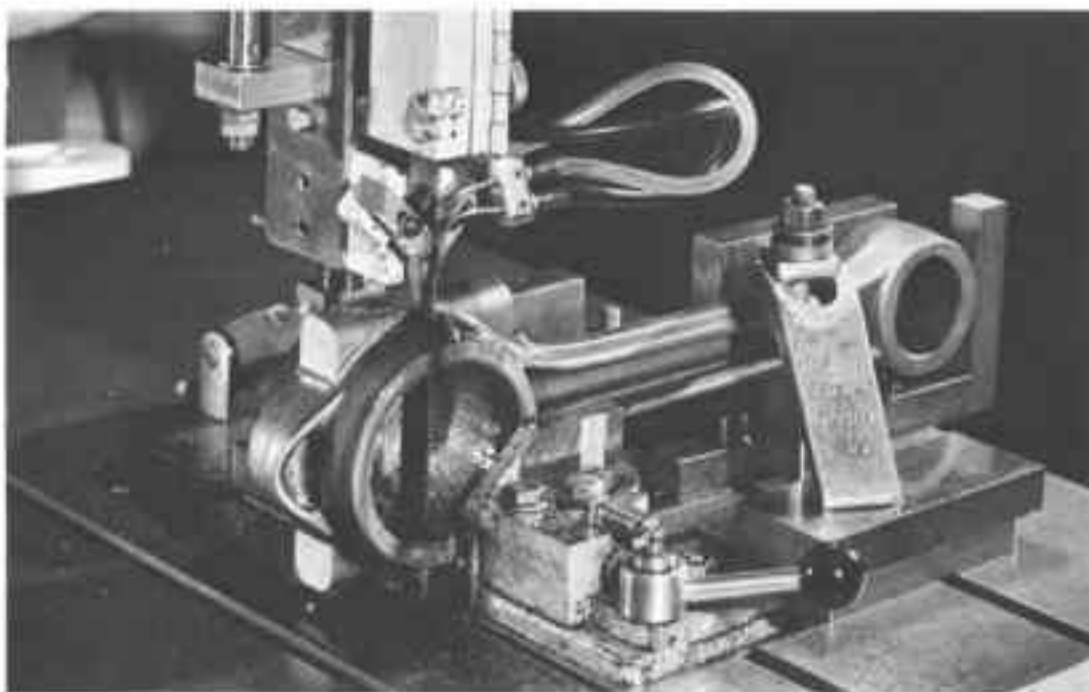
Courtesy of the DoALL Company

Fig. 16-22. Combining the hydraulic table-feed and servo-contour directional feed-control for cutting straight lines and contours.



Courtesy of the DoALL Company

Fig. 16-23. Combining various shaping cuts in several planes enables the vertical-contour band saw to produce very complicated parts.



Courtesy of the DoALL Company

Fig. 16-24. Separating the cap of a large, connecting-rod forging which is held in a fixture.



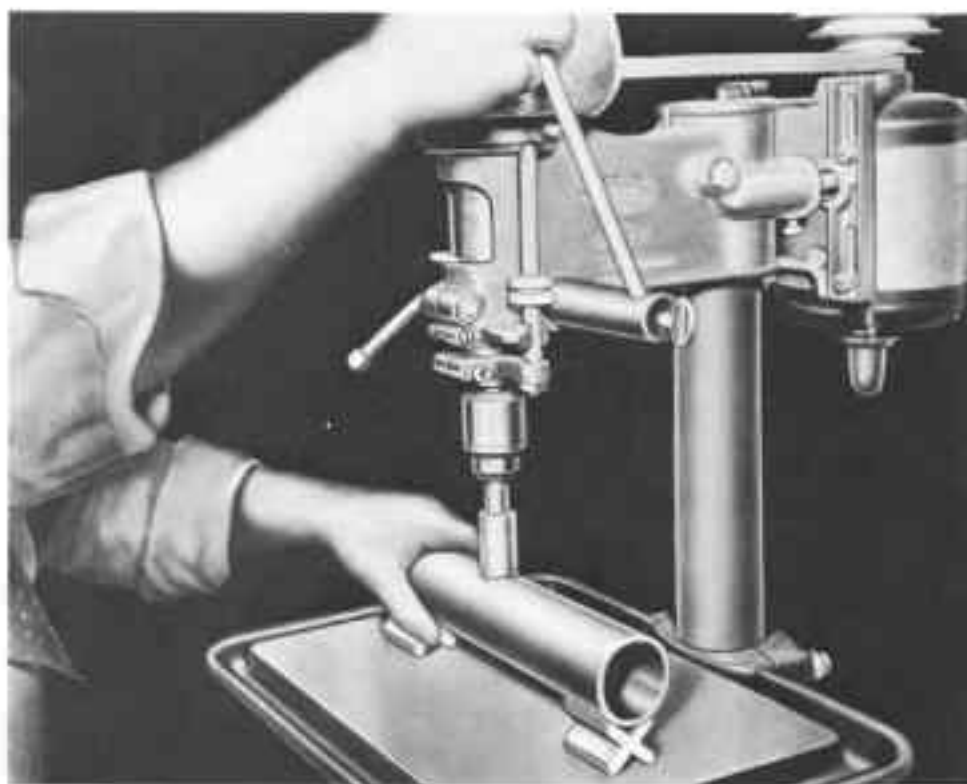
Courtesy of The L. S. Starrett Company

Fig. 16-25. Hole saw.

for heat to transfer into the tooth. Certain materials such as sponge rubber, cloth, cork, leather, synthetic sponges, and corrugated aluminum are cut by slicing with a continuous knife blade that is revolved on the wheels of the saw. Intricate scroll work can be done with spiral saw bands. Grinding and polishing blades are used to cut certain materials, to deburr, and to break sharp corners. Electro-band machining is done by feeding a low-voltage high-amperage current into the knife band at a rate that causes it to discharge a sustained arc from the knife edge onto the material being cut. A flood of coolant quenches the arc to prevent it from burning the material. The material is disintegrated by the arc without any actual contact between the work and the band. This band-machining process is used to cut heat-exchanger cores, honeycomb structures, flexible tubing, etc. The method requires a special machine and cannot be adapted to the ordinary vertical-contour band saw.

Hole Saws

Hole saws are used to cut holes in sheet metal, tubing, and in other thin metal parts. The thrust force of the larger twist drills would deflect and bend, rather than penetrate, these parts. The hole saw, Fig. 16-25, has a small twist drill in the center which drills a small hole before the saw comes in contact with the work. The small twist drill then acts as a



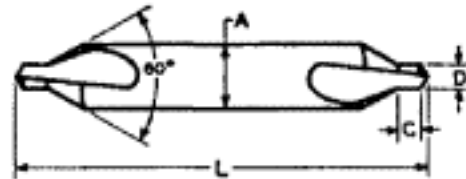
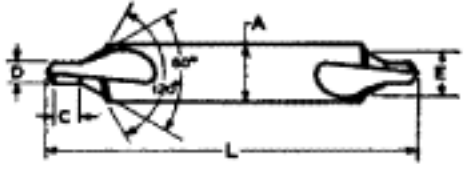
Courtesy of The L. S. Starrett Company

Fig. 16-26. Drilling a hole in tubing with a hole saw.

pilot to locate and to steady the saw while it cuts through the metal to produce a larger hole. Figure 16-26 illustrates a hole being drilled in tubing with a hole saw.

Appendix

Table A-1. ANSI Standard Combined Drills and Countersinks, Plain and Bell Types (ANSI B94.11-1967, R1972)

PLAIN TYPE					BELL TYPE					
										
Size Designation	Body Diam.	Drill Diam.	Drill Length	Overall Length	Size Designation	Body Diam.	Drill Diam.	Drill Length	Overall Length	Bell Diam.
	A	D	C	L		A	D	C	L	E
00	3/8	.025	.040	1 1/2	11	3/8	3/64	3/64	1 1/4	0.100
0	3/8	3/32	3/64	1 1/2	12	3/8	1/16	1/16	1 1/4	0.150
1	3/8	3/64	3/64	1 1/4	13	3/4	3/32	3/32	2	0.200
2	3/8	3/64	3/64	1 1/4	14	3/8	3/64	3/64	2 1/4	0.250
3	3/4	7/64	7/64	2	15	3/8	3/32	3/32	2 1/4	0.350
4	3/8	3/8	3/8	2 1/8	16	3/4	3/16	3/16	3	0.400
5	3/8	3/16	3/16	2 3/4	17	3/8	7/32	7/32	3 1/4	0.500
6	3/4	7/32	7/32	3	18	3/4	1/4	1/4	3 1/4	0.600
7	3/8	1/4	1/4	3 1/4	..	..	..	..	..	..
8	3/4	3/8	3/8	3 1/2	..	..	..	..	..	..

All dimensions except size designation are in inches.

Tolerance: Plus 0.0000 minus 0.0020 on A; plus 0.0030 minus 0.0000 on D; plus 3/64 minus 1/16 on L; plus 0.0080 minus 0.0080 on C for sizes 00 through 2, 11 and 12; plus 3/64 minus 3/64 on C for sizes 3 through 8 and sizes 13 through 18.

Table A-2. ANSI Drill Sizes, Their Inch and Millimeter Equivalents, and Millimeter Size Drills for Which They Can Be Used

ANSI Size (inch)	Decimal Equivalent, inch	Millimeter Equivalent	Use For mm Drill Size	ANSI Size (inch)	Decimal Equivalent, inch	Millimeter Equivalent	Use For mm Drill Size
80	.0135	0.34	0.35	36	.1065	2.71	2.70
79	.0145	0.37	0.38	$\frac{7}{64}$	.1094	2.78	2.80
$\frac{1}{64}$	.0156	0.40	0.40	35	.1100	2.79	2.80
78	.0160	0.41	0.42	34	.1110	2.82	2.85
77	.0180	0.46	0.45	33	.1130	2.87	2.90
76	.0200	0.51	0.50	32	.1160	2.95	2.95
75	.0210	0.53	0.55	31	.1200	3.05	3.05
74	.0225	0.57	0.60	$\frac{1}{8}$	.1250	3.18	3.20
73	.0240	0.61	0.60	30	.1285	3.26	3.25
72	.0250	0.64	0.65	29	.1360	3.45	3.45
71	.0260	0.66	0.65	28	.1405	3.57	3.60
70	.0280	0.71	0.70	$\frac{9}{64}$	.1406	3.57	3.60
69	.0292	0.74	0.75	27	.1440	3.66	3.65
68	.0310	0.79	0.80	26	.1470	3.73	3.75
$\frac{1}{32}$	.0312	0.79	0.80	25	.1495	3.80	3.80
67	.0320	0.81	0.80	24	.1520	3.86	3.85
66	.0330	0.84	0.85	23	.1540	3.91	3.90
65	.0350	0.89	0.90	$\frac{5}{32}$	.1562	3.97	4.00
64	.0360	0.91	0.90	22	.1570	3.99	4.00
63	.0370	0.94	0.95	21	.1590	4.04	4.05
62	.0380	0.97	1.00	20	.1610	4.09	4.10
61	.0390	0.99	1.00	19	.1660	4.22	4.25
60	.0400	1.02	1.05	18	.1695	4.31	4.30
59	.0410	1.04	1.05	$\frac{11}{64}$	.1719	4.37	4.40
58	.0420	1.07	1.10	17	.1730	4.39	4.40
57	.0430	1.09	1.10	16	.1770	4.50	4.50
56	.0465	1.18	1.20	15	.1800	4.57	4.60
$\frac{3}{64}$	.0469	1.19	1.20	14	.1820	4.62	4.65
55	.0520	1.32	1.35	13	.1850	4.70	4.70
54	.0550	1.40	1.40	$\frac{3}{16}$	.1875	4.76	4.75
53	.0595	1.51	1.50	12	.1890	4.80	4.80
$\frac{1}{16}$	.0625	1.59	1.60	11	.1910	4.85	4.85
52	.0635	1.61	1.60	10	.1935	4.91	4.90
51	.0670	1.70	1.70	9	.1960	4.98	5.00
50	.0700	1.78	1.80	8	.1990	5.05	5.05
49	.0730	1.85	1.85	7	.2010	5.11	5.10
48	.0760	1.93	1.95	$\frac{13}{64}$	.2031	5.16	5.15
$\frac{5}{64}$	.0781	1.98	2.00	6	.2040	5.18	5.20
47	.0785	1.99	2.00	5	.2055	5.22	5.25
46	.0810	2.06	2.05	4	.2090	5.31	5.30
45	.0820	2.08	2.10	3	.2130	5.41	5.40
44	.0860	2.18	2.20	$\frac{7}{32}$	.2188	5.56	5.55
43	.0890	2.26	2.25	2	.2210	5.61	5.60
42	.0935	2.37	2.40	1	.2280	5.79	5.80
$\frac{3}{32}$	.0938	2.38	2.40	A	.2340	5.94	5.95
41	.0960	2.44	2.45	$\frac{15}{64}$	.2344	5.95	5.95
40	.0980	2.50	2.50	B	.2380	6.05	6.05
39	.0995	2.53	2.55	C	.2420	6.15	6.15
38	.1015	2.58	2.60	D	.2460	6.25	6.25
37	.1040	2.64	2.65	E	.2500	6.35	6.35

Courtesy of the National Twist Drill & Tool Div., Lear Siegler Inc.

Table A-2 (Cont.). ANSI Drill Sizes, Their Inch and Millimeter Equivalents, and Millimeter Size Drills for Which They Can Be Used

ANSI Size (inch)	Decimal Equivalent, inch	Millimeter Equivalent	Use For mm Drill Size	ANSI Size (inch)	Decimal Equivalent, inch	Millimeter Equivalent	Use For mm Drill Size
1/4	.2500	6.35	6.35	15/32	.4688	11.91	11.90
F	.2570	6.53	6.55	31/64	.4844	12.30	12.30
G	.2610	6.63	6.65	1/4	.5000	12.70	12.70
17/64	.2656	6.75	6.75	33/64	.5156	13.10	13.10
H	.2660	6.76	6.75	17/32	.5312	13.49	13.50
I	.2720	6.91	6.90	35/64	.5469	13.89	13.90
J	.2770	7.04	7.05	9/16	.5625	14.29	14.30
K	.2810	7.14	7.15	37/64	.5781	14.68	14.70
9/32	.2812	7.14	7.15	19/32	.5938	15.08	15.10
L	.2900	7.37	7.40	39/64	.6094	15.48	15.50
M	.2950	7.49	7.50	5/8	.6250	15.88	15.90
19/64	.2969	7.54	7.55	41/64	.6406	16.27	16.30
N	.3020	7.67	7.70	21/32	.6562	16.67	16.70
5/16	.3125	7.94	7.95	43/64	.6719	17.07	17.10
O	.3160	8.03	8.05	11/16	.6875	17.46	17.45
P	.3230	8.20	8.20	45/64	.7031	17.86	17.85
21/64	.3281	8.33	8.35	23/32	.7188	18.26	18.25
Q	.3320	8.43	8.45	47/64	.7344	18.65	18.65
R	.3390	8.61	8.60	3/4	.7500	19.05	19.05
11/32	.3438	8.73	8.75	49/64	.7656	19.45	19.45
S	.3480	8.84	8.85	25/32	.7812	19.84	19.85
T	.3580	9.09	9.10	51/64	.7969	20.24	20.25
23/64	.3594	9.13	9.15	13/16	.8125	20.64	20.65
U	.3680	9.35	9.35	53/64	.8281	21.03	21.05
3/8	.3750	9.52	9.55	27/32	.8438	21.43	21.45
V	.3770	9.56	9.55	55/64	.8594	21.83	21.85
W	.3860	9.80	9.80	7/8	.8750	22.23	22.25
25/64	.3906	9.92	9.95	57/64	.8906	22.62	22.65
X	.3970	10.08	10.10	29/32	.9062	23.02	23.05
Y	.4040	10.26	10.25	59/64	.9219	23.42	23.45
13/32	.4062	10.32	10.35	15/16	.9375	23.81	23.80
Z	.4130	10.49	10.50	61/64	.9531	24.21	24.20
27/64	.4219	10.72	10.75	31/32	.9688	24.61	24.60
7/16	.4375	11.11	11.10	63/64	.9844	25.01	25.00
29/64	.4531	11.51	11.50	1	1.0000	25.40	25.40

Courtesy of the National Twist Drill & Tool Div., Lear Siegler Inc.

Table A-3. Some Frequently Used Conversion Factors

Quantity	Multiply	By	To Obtain
Length	inch (in.)	0.0254*	meter (m)
	inch (in.)	2.54*	centimeter (cm)
	inch (in.)	25.4*	millimeter (mm)
	foot (ft)	0.3048*	meter (m)
	foot (ft)	304.8*	millimeter (mm)
	yard (yd)	0.9144*	meter (m)
	yard (yd)	914.4*	millimeter (mm)
	mile (mi)	1.609344*	kilometer (km)
	millimeter (mm)	0.03937008	inch (in.)
	millimeter (mm)	0.00328084	foot (ft)
	millimeter (mm)	0.00109361	yard (yd)
	centimeter (cm)	0.3937008	inch (in.)
	meter (m)	39.37008	inch (in.)
	meter (m)	3.280840	foot (ft)
	meter (m)	1.093613	yard (yd)
	kilometer (km)	0.6213712	mile (mi)
	inch ² (in. ²)	645.16*	millimeters ² (mm ²)
	inch ² (in. ²)	0.00064516*	feet ² (ft ²)
	foot ² (ft ²)	92.90304*	millimeters ² (mm ²)
	foot ² (ft ²)	0.09290304*	feet ² (ft ²)
Area	millimeters ² (mm ²)	0.001550003	inch ² (in. ²)
	millimeters ² (mm ²)	0.0001076387	foot ² (ft ²)
	meters ² (m ²)	1550.003	inch ² (in. ²)
	meters ² (m ²)	10.763910	foot ² (ft ²)
	inch/minute (in./min)	25.4*	millimeter/minute (mm/min)
	foot/minute (fpm)	0.3048*	meter/minute (m/min)
	mile/hour (mph)	1.609344*	kilometer/hr (km/hr)
	millimeter/minute (mm/min)	0.03937008	inch/min (in./min)
Velocity	meter/minute (m/min)	3.280840	foot/minute (fpm)
	kilometer/hour (km/hr)	0.6213712	mile/hour (mi/hr)
	pound (lb)	0.4535924	kilogram (kg)
	ton (long 2240 lb)	1016.047	kilogram (kg)
Mass	ton (short 2000 lb)	907.1847	kilogram (kg)
	ton (metric)	1000.*	kilogram (kg)
	kilogram (kg)	2.204622	pound (lb)
	kilogram (kg)	0.0009842064	ton (long 2240 lb)
Force	kilogram (kg)	0.001102311	ton (short 2000 lb)
	kilogram (kg)	0.001*	ton (metric)
	pound-force (lbf)	4.448222	newton (N)
	ounce-force (oz)	0.2780139	newton (N)
Torque	newton (N)	0.2248089	pound-force (lbf)
	newton (N)	3.596942	ounce-force (oz)
	pound-inch (lb-in.)	0.11298448	newton-meter (N·m)
	pound-foot (lb-ft)	1.355818	newton-meter (N·m)
Pressure	newton-meter (N·m)	8.850745	pound-inch (lb-in.)
	newton-meter (N·m)	0.7375621	pound-foot (lb-ft)
	pound/inch ² (lb/in. ²)	6894.757	pascal (Pa)
	pound/inch ² (lb/in. ²)	6894.757	newton/meter ² (N/m ²)
	pound/inch ² (lb/in. ²)	0.0048894757	newton/millimeter ² (N/mm ²)
	pound/inch ² (lb/in. ²)	0.07500667	kilogram/centimeter ² (kg/cm ²)
	pound/inch ² (lb/in. ²)	0.06894757	bar
	pascal (Pa)	0.00001*	bar
	pascal (Pa)	0.0001450377	pound/inch ² (lb/in. ²)
	newton/meter ² (N/m ²)	0.0001450377	pound/inch ² (lb/in. ²)
	newton/millimeter ² (N/mm ²)	14.50377	pound/inch ² (lb/in. ²)
	kilogram/centimeter ² (kg/cm ²)	14.22334	pound/inch ² (lb/in. ²)
Power	bar	14.50377	pound/inch ² (lb/in. ²)
	bar	100,000.*	pascal (Pa)
	horsepower (hp)	745.6999	watt (W)
	horsepower (hp)	0.7456999	kilowatt (kW)
	watt (W)	0.001341022	horsepower (hp)
	kilowatt (kW)	1.341022	horsepower (hp)

* Where an asterisk is shown, the conversion factor is exact.

Table A-4. Decimal Equivalents

	$\frac{1}{64}$	.015625		$\frac{33}{64}$	.515625
	$\frac{1}{32}$	.03125		$\frac{17}{32}$	.53125
	$\frac{3}{64}$	.046875		$\frac{35}{64}$	.546875
$\frac{1}{16}$	$\frac{5}{64}$	.0625	$\frac{9}{16}$	$\frac{37}{64}$	.5625
	$\frac{3}{32}$	.078125		$\frac{19}{32}$	.578125
	$\frac{7}{64}$	.09375		$\frac{39}{64}$	.59375
$\frac{1}{8}$	$\frac{7}{64}$	.109375	$\frac{5}{8}$	$\frac{39}{64}$	.609375
	$\frac{9}{64}$	.125		$\frac{41}{64}$	.625
	$\frac{5}{32}$	.140625		$\frac{21}{32}$	.640625
	$\frac{11}{64}$	.15625		$\frac{43}{64}$	.65625
$\frac{3}{16}$	$\frac{11}{64}$	.171875	$\frac{11}{16}$	$\frac{43}{64}$	.671875
	$\frac{13}{64}$	.1875		$\frac{45}{64}$	.6875
	$\frac{7}{32}$	.203125	$\frac{23}{32}$	$\frac{45}{64}$	.703125
	$\frac{15}{64}$	.21875		$\frac{47}{64}$	.71875
$\frac{1}{4}$	$\frac{15}{64}$	.234375	$\frac{3}{4}$	$\frac{47}{64}$	.734375
	$\frac{17}{64}$	.250		$\frac{49}{64}$	.750
	$\frac{9}{32}$	.265625		$\frac{25}{32}$	.765625
	$\frac{19}{64}$	.28125		$\frac{51}{64}$	.78125
$\frac{5}{16}$	$\frac{19}{64}$	.296875	$\frac{13}{16}$	$\frac{51}{64}$	.796875
	$\frac{21}{64}$	.3125		$\frac{53}{64}$	.8125
	$\frac{11}{32}$	.328125		$\frac{27}{32}$	.828125
	$\frac{23}{64}$	.34375		$\frac{55}{64}$	.84375
$\frac{3}{8}$	$\frac{23}{64}$	.359375	$\frac{7}{8}$	$\frac{55}{64}$	.859375
	$\frac{25}{64}$	.375		$\frac{57}{64}$	.875
	$\frac{13}{32}$	.390625		$\frac{29}{32}$	.890625
	$\frac{27}{64}$	.40625		$\frac{59}{64}$	.90625
$\frac{7}{16}$	$\frac{27}{64}$	.421875	$\frac{15}{16}$	$\frac{59}{64}$	.921875
	$\frac{29}{64}$	.4375		$\frac{61}{64}$	.9375
	$\frac{15}{32}$	.453125		$\frac{31}{32}$	.953125
	$\frac{31}{64}$	.46875		$\frac{63}{64}$	.96875
$\frac{1}{2}$	$\frac{31}{64}$	.484375	$\frac{1}{1}$	$\frac{63}{64}$	.984375
	$\frac{1}{2}$	.500		$\frac{64}{64}$	1.000

Table A-5. Cutting Speeds for Machine Tapping

Material	Cutting Speed, fpm	Material	Cutting Speed, fpm
<i>Low Carbon Steels</i> Up to .25% C	40 to 80	<i>Aluminum</i>	50 to 200
<i>Medium Carbon Steels</i> .30 to .60% C Annealed	30 to 60	<i>Brass</i>	50 to 200
Heat Treated (220 to 280 Bhn)	20 to 50	<i>Manganese Bronze</i>	30 to 60
<i>Tool Steels, High Carbon &</i> <i>High-Speed Steel</i>	20 to 40	<i>Phosphor Bronze</i>	30 to 60
<i>Stainless Steels</i>	5 to 35	<i>Naval Brass</i>	80 to 100
<i>Gray Cast-Iron</i>	40 to 100	<i>Monel Metal</i>	20 to 40
<i>Malleable Iron</i>		<i>Tobin Bronze</i>	80 to 100
Ferritic	80 to 120	<i>Plastics</i>	
Pearlitic	40 to 80	Thermoplastic	50 to 100
<i>Zinc Die Castings</i>	60 to 150	Thermosetting	50 to 100
		<i>Hard Rubber</i>	50 to 100
		<i>Bakelite</i>	50 to 100

Index

- Accuracy, 52
- Allowance, 51
- Angle plate, 54, 120, 294, 298-308
- Band machining, 473-482
 - feeding pressure, 464-467
 - feed rate, 464-467, 474
 - fling, 474, 477, 479
 - servo-contour feed control, 477, 478
 - three-dimensional sawing, 478
- Bandsaw, vertical contour, 463-467
- Band-type cutoff saw, 461, 462
- Basic size, 51
- Bilateral tolerance, 52
- Boring head, offset, 405, 436
- Boring, internal tapers and angles, 270-272
 - internal threads, 373-375
 - objectives of, 270
 - on drill presses, 133
 - on engine lathes, 270-273
 - on jig borers, 404-406, 433-437
 - on turret lathes, 388
 - principles of, 7, 270, 404-406
 - taper, 291-293
- Boring mill, vertical, 398-402
- Brinell hardness, 161
- Broaching, principles of, 8, 9
- Calipers, 15-18
 - gear tooth (vernier), 345
 - hermaphrodite, 15, 16, 55, 58
 - inside, 15-18
 - inside micrometer, 31
 - outside, 15-18
 - outside micrometer, 26-31, 45-48
 - vernier, 24
- Center drill, 223
- Center drilling
 - for starting a drilled hole, 125-129, 264, 410
 - to prepare work for turning, 223-227
- Center gage, 362
- Center head, 18, 19, 58
- Centering projector, 430
- Chatter, 95, 183, 184
- Chip breakers, 186-188, 192, 204, 205
- Chip control, 204, 205
- Chip formation, 199-204
 - in drilling, 90-92
- Chucking part
 - in collet chuck, 257-260
 - in four-jaw independent chuck, 253-257
 - in step chuck, 259
 - in three-jaw universal chuck, 252, 253
- Chucks (see *Lathe attachments*)
- Collet chuck work, 261-263
- Combination drill and countersink, 223
- Combination square, 18, 19
- Comparators, optical, 332-335, 362
- Continental Process, for making dies, 476-478
- Coordinate dimensions, 403, 404
 - transfer of tolerances to, 453-457
- Coordinate measuring system, 419-421, 422
- Counterbores, 96
- Counterboring, principles of, 8
- Countersink, 96
- Cut-off tool, 194, 197, 273, 274
- Cutting conditions, selection of, 171, 218
- Cutting fluids, 212, 213
 - for preventing built-up edge, 209
 - for reaming, 133
 - for tapping, 135
 - for turning, 170, 245
- Cutting-off operation, 273-275, 392
- Cutting speed, 162-171
 - factors affecting, 162-171
 - for boring, 271, 272
 - for cutting-off operations, 274
 - for drilling, 108-116
 - for reaming, 95, 111-116
 - for sawing, 461, 466

- for tapping, 134
 - for thread cutting, 356, 357
 - for turning, 162-172
- Cutting tools, indexable insert type, 185-189, 197
- Cutting tool materials, 172-179
 - carbon tool steel, 110
 - cast nonferrous alloys, 179
 - cemented carbides, 174-177
 - ceramic, 177, 178
 - cermets, 177, 178
 - cubic boron nitride, 178
 - diamond, 178
 - for twist drills, 110
 - high-speed steel, 173, 174
 - polycrystalline diamond, 178
- Cylindrical grinding, 11
- Dial test indicator, 34, 35
- Diamond boring, 278
- Die head, self-opening, 390
- Digital readout systems, 242, 243
- Dividers, 55
- Drill
 - center, 223
 - combination drill and countersink, 223
 - core, 387, 388
 - gun, 101, 102
 - point angles, 84-90
 - relief angles, 84, 85
 - spade, 100, 101, 389
 - split point, 86-88
 - spotting, 433
 - start, 389
 - twist, 81-83
 - web thinning, 90
- Drill chuck, 155
- Drill jigs, 135-138
- Drilling
 - basic principles of, 7, 8
 - chip formation, 91, 92
 - feed rate, 112, 113, 116
 - hole location transfer, 130-132
 - on a drill press, 124-129
 - on a lathe, 264-269
 - on a turret lathe, 387-389
 - oversize amount normally cut, 83, 84
 - power required, 102-107
 - setups, 114-124
 - speed, 108-116
 - thrust, 102-107
 - torque, 102-107
- Drilling machines, 75-81
 - gang, 80, 81
 - multiple spindle, 80, 81
 - radial, 78, 79
 - sensitive, 75
 - turret, 79, 80
 - upright, 75-78
- Drill presses (see *Drilling machines*)
- Edge finder, 429-431
- Edge finding, 424-432
- Electro-band machining, 482
- Engine lathe, 139-155
 - apron, 153
 - back gears, 143
 - bed, 141, 142
 - carriage, 149
 - change gears, 146, 346-356
 - compound rest, 152
 - cross slide, 149-152
 - driver plate, 236
 - headstock, 142, 143
 - headstock spindle, 143-146
 - headstock spindle bearings, 143
 - headstock spindle nose, 143-146
 - micrometer dials, 149-152
 - quick-change gearbox, 146-148, 351, 352
 - saddle, 149
 - tailstock, 153-155
 - ways, 142
- Faceplate, 155, 294
 - chuck, 295, 296, 305
 - jaws, 295, 296
 - setups, 296-310
 - work, precision hole location, 412-417, 442-445
- Facing
 - back facing, 192, 196
 - end of work, 227, 263, 264
 - end of work held between centers, 194, 237-240
 - shoulders, 244, 245
 - to true faceplate, 294, 295
- Feed rate
 - for boring, 272
 - for contour bandsawing, 464-467, 474
 - for cutting-off, 274, 275
 - for drilling, 110-113, 116
 - for power hacksawing, 461
 - for reaming, 95
 - for turning, 169, 170, 218, 228
- Filing on a lathe, 242
- Fixtures
 - contour band saw, 478
 - drill press, 135-138
 - engine lathe, 155, 275-278, 305-310
 - turret lathe, 392
- Fly-cutter, 442
- Formulas for
 - American Standard Acme Threads, 320

- American Standard Unified Threads, 314
- ANSI-ISO Metric Threads, 326, 327
- base pitch of metric thread dial, 360
- change gear ratios for cutting inch threads, 347, 356
- change gear ratios for cutting metric threads, 350, 353, 354
- change gear ratios for modifying gearbox output, 353-356
- cord of hole circle, 60
- equivalent threads per inch, 354, 379
- ISO-Metric Trapezoidal Threads, 328
- power for drilling, 105
- power for turning, 217
- screw thread pitch, 311
- sine bar, 283
- spindle speed for drilling, 109
- spindle speed for turning, 171
- tap drill size, 283
- taper calculations, 280, 281, 283, 284
- taper turning, 289
- three-wire thread measurement, 336-339
- thrust for drilling, 105
- torque for drilling, 105
- Friction sawing, 478-482

- Gage blocks, precision, 35-39
- Gages
 - center, 262
 - depth (vernier), 24
 - height (vernier) 24, 34
 - planer, 40, 41
 - small hole, 18
 - surface, 39, 40
 - taper, plug and ring, 281, 282
 - telescoping, 18
 - thread, plug and ring, 330-332
 - thread, roll type, 332
 - thread, snap, 332
 - universal precision, 40, 41
- Grinding attachment, lathe tool post, 160
- Grinding
 - cylindrical, 11
 - jig, 442-453
 - precision, 10-12
 - single-point tool, 198, 199
 - surface, 12

- Hacksaw, power operated, 459-461
- Hardness, 161, 162
- Hole saws, 482, 483

- Indicator, dial test, 34, 35

- Jig borer, 419
- Jig boring practice, 421-437
 - aligning and clamping the workpiece, 421-424
 - drilling and boring the hole, 433-437
 - locating the hole, 432, 433
 - locating the workpiece and spindle, 424-432
 - picking up an edge, 426-432
- Jig grinder, 446-450
- Jig grinding practice, 442-453

- Lathe attachments
 - collet chuck, 155, 257-260
 - driver plate, 236
 - faceplate, 155
 - follower rest, 248, 249
 - four-jaw independent chuck, 155, 253-257
 - micrometer carriage stop, 156, 157
 - square turret, 268, 269, 306, 307
 - steady rest, 157, 246, 247
 - taper attachment, 285-287
 - thread chasing dial, inch, 357-359
 - thread chasing dial, metric, 359-362
 - three-jaw universal chuck, 155, 252, 253
 - tool post grinder, 160
 - tracer attachment, 157-160
- Lathe centers, 223, 229
 - live, 231, 232
- Lathe operations
 - aligning centers, 231-235
 - angle boring, 291-293
 - angle turning, 289-291
 - back facing, 192, 196
 - basic operations, 220
 - boring, 270-273
 - chucking, 252-257
 - cutting-off, 194, 197, 273-275
 - drilling, 264-269
 - end facing 237-240
 - faceplate work, 294-310
 - facing, 194, 197, 245, 263, 264, 295
 - filing, 242
 - forming, 194, 245
 - grinding, 160, 230
 - grooving, 194, 197, 244
 - knurling, 249-251
 - necking, 194, 197, 244
 - polishing, 242
 - reaming, 269, 270
 - shoulder turning, 243-245
 - taper boring, 291-293
 - taper turning, 285-289
 - tapping, 376
 - thread cutting, 362-381

- truing centers, 230, 231
- turning on centers, 220-251
- Lathe tools
 - dogs, 236
 - drill chuck, 155
 - drill holder, 267, 268
 - floating reamer holder, 269, 390
 - mandrel, 245, 246, 305
- Lathes
 - automatic, 393-397
 - engine, 139-155
 - hollow spindle, 141
 - production, 397, 398
 - sliding gap-bed, 140, 141
 - turret, 382-392
 - vertical, 398-402
- Layout machines, 70-74
- Layout procedures, 53-70
 - adjusting the layout, 69, 70
 - for castings, 60-68
 - for cylindrical parts, 55-58
 - for equally spaced holes on a circle, 58-60
 - for hole locations, 54, 55, 407-409
 - painting, 51, 52
- Layout scale, 64
- Leveling lathes, method of, 229
- Limits, dimensional, 51
- Mandrels (*see Lathe tools*)
- Machine shop, the work of, 1
- Machine tools, general description, 2
- Measurement, basic nomenclature of, 51, 52
- Metrical, 42-44
- Micrometer measuring instruments, 26-33
 - checking, 32, 33
 - depth micrometer, 31
 - inside micrometer, 31
 - Mul-T-Anvil micrometer, 32
 - outside micrometer, 26, 27, 31, 45
 - screw thread micrometer, 339, 340
- Micrometer scale reading
 - inch, 27-31
 - metric, 45-48
- Milling, 5-7
- Nominal Size, 51
- Optical comparators, 332-335
- Parallels, 41, 42
 - box type, 442
- Planer gage, 40, 41
- Planing, 5
- Power, estimating
 - for drilling, 102-107
 - for maximum utilization, 218, 219
 - for turning, 213-219
- Precision, 52
- Precision hole location methods, 403-453
 - buttoning, 414, 418
 - jig boring, 421-437
 - layout and drilling, 407-410
 - layout, drilling, and boring, 411-414
- Protractor
 - bevel, 18, 19
 - vernier bevel, 24-26
- Pump center, 412
- Quick-change gearbox, 146-148, 351, 352
 - modifying output of, 352-356
- Reamers, 92-96
 - floating blade type, 390
 - hand, 92-95
 - machine, 92-95
 - rose, 95, 96
 - taper, 95, 96, 292
- Reaming, basic principles of, 92-95
 - on a jig borer, 436, 437
 - on a lathe, 269, 270
 - stock removal for, 95, 436
 - tapers, 292, 293
- Relief angle
 - of boring tools, 194
 - of single-point tools, 179-181
 - of thread cutting tools, 342, 344
 - of twist drills, 84-87
- Rockwell hardness, 161
- Rotary table, 438-441
- Rules, precision steel
 - inch, 13-15
 - metric, 44
- Saw blades
 - for power hacksaw, 459-461
 - for vertical contour saw, 467-473
- Saw guides, adjustment of, 463
- Saw, hole, 482, 483
- Sawing, friction, 478-482
- Sawing machines, 459-483
 - band-type cutoff, 461, 462
 - power hacksaw, 459-461
 - vertical contour band, 463-467
- Screw threads (*see Threads*)
- Shaping, 5
- Sine bar, 282, 283
- Sine plate, 36, 37
- Sine table, 438
- Single-point cutting tools, 161-219
 - grinding, 198, 199
- Single-point cutting tool
 - end cutting edge angle, 182

- lead angle, 182-184, 194, 271
- nomenclature, 179
- nose radius, 184, 185
- rake angles, 181, 182
- relief angle, 179-181
- side cutting edge angle, 182-184
- Single-point cutting tool design
 - boring, 189, 197, 270, 271
 - broad-nose finishing, 197, 198
 - contour turning, 197
 - cut-off, 189, 197, 273, 274
 - end facing, 194, 239
 - facing, 194, 197
 - grooving and necking, 189, 194, 197
 - rough and finish turning, 197
 - thread cutting, 189, 342-345
 - turn and facing, 192, 196, 244, 245
 - turning, 191, 192, 196
- Spot facer, 96
- Square
 - combination, 18
 - cylindrical, 20
 - method of checking squareness, 19, 20
 - precision steel, 19
- Surface gage, 39, 40
- Surface grinding, 12
- Tap, 97-99
- Tap drill sizes, 98
- Tapping
 - by hand, 97, 99, 100
 - on a drill press, 133-135
 - on an engine lathe, 376
 - on a turret lathe, 390
- Taper attachment, 285-287
- Taper boring, 291-293
- Taper calculations, 280, 281
- Taper reaming, 292, 293
- Taper thread turning, 377, 378
- Taper turning, 285-289
- Taper turning calculations, 289
- Tapers
 - American Standard Machine, 280
 - Brown and Sharpe, 279
 - Jarno, 279, 280
 - Milling Machine, 280
 - Morse, 279
- Tapers, measuring and gaging, 281-285
- Telescoping gage, 18
- Threads
 - American National Thread System, 318
 - American Standard Acme Threads, 319-321
 - American Standard Buttress Screw Threads, 322
 - American Standard Microscopic Objective Threads, 319
 - American Standard Taper Pipe Threads, 321, 322
 - American Standard Unified Miniature Screw Threads, 319
 - American Standard Unified Threads, 313-318
 - ANSI Metric Threads, 323-329
 - British Association Standard Threads, 323
 - British Standard Fine Threads, 322, 323
 - British Standard Whitworth Threads, 322
 - definitions of terms, 311-313
 - Interference Fit Threads, 321
 - ISO Metric Trapezoidal Threads, 328, 329
 - Square Threads, 321
- Thread cutting
 - on an engine lathe, 341-381
 - on a turret lathe, 390
- Thread measuring and gaging, 329-340
 - mating part as a gage, 332
 - optical comparator, 332-335
 - plug and ring gages, 330-332
 - roll-type thread gage, 332
 - screw thread micrometer, 339, 340
 - snap gage, 332
 - three-wire method, 335-339
 - toolmakers microscope, 335
- Tolerances, bilateral and unilateral, 51, 52
- Tool holders for single-point cutting
 - tools, 185-189
- Tool life, 169, 170
- Toolmakers buttons, 414, 415
- Toolmakers microscope, 335, 362
- Tool materials (see *Cutting tool materials*)
- Tool wear, 205-212
 - built-up edge, 202-204, 207-209
 - chipping, 211
 - crater wear, 209
 - deformation, 211
 - flank wear, 205-207
 - notching, 209, 210
 - oxidation, 211, 212
 - pullouts, 210
 - thermal cracking, 210
- Transfer punch, 130
- Transfer screw, 131, 132
- Transfer of tolerances, 453-458
- Trial cut procedure, 240, 241
- Turret lathe, 382-386
- Turret lathe operations
 - boring, 388
 - cutting-off, 392
 - drilling, 387, 388
 - multiple turning, 387, 388

- overhead turning, 387
- planning, 392
- reaming, 389, 390
- roller bar turning, 391
- spade drilling, 389
- start drilling, 389
- stock shop, 389
- tapping, 390
- thread cutting, 390
- Turret lathe tooling, 387-392
 - combination stock stop and start drill, 389
 - cutter holders, 387
 - floating reamer holder, 390
 - multiple turning head, 387
 - rear cutter block, 392
 - releasing type tap holder, 390
 - roller bar turner, 391
 - self-opening die head, 390
 - slide tool, 388
- Twist drill, 81-90
 - chip formation, 90-92
 - point geometry, 84-90
 - nomenclature, 81-83
 - sizes, 83
- V-blocks, 57, 123, 304, 441
- Vernier measuring instruments, 21-26
 - bevel protractor, 24-26
 - caliper, 24
 - depth gage, 24
 - gear tooth caliper, 345
 - height gage, 24, 35
- Vernier scale reading
 - inch, 21-24
 - metric, 48-50
 - vernier bevel protractor scale, 24-26
- Vertical boring mills, 398-402
- Vertical contour bandsaw, 463-467
- Vertical turret lathes, 398-402
- Welding saw blades, procedure for, 471-473
- Witness marks, 55, 125, 409

