

# Basic Machining Reference Handbook

Second Edition

*Arthur R. Meyers*

*Thomas J. Slattery*

Industrial Press Inc.  
200 Madison Avenue  
New York, New York 10016-4078

**Library of Congress Cataloging-in-Publication Data**

Meyers, Arthur R.

Basic machining reference handbook.

Includes index.

1. Machine-shop practice—Handbooks, manuals, etc.

I. Slattery, Thomas. II. Title.

TJ1165.M45 1988 670.42'3 87-3645

ISBN 0-8311-1174-7

2<sup>nd</sup> Edition, 2001

**Industrial Press Inc.**

200 Madison Avenue  
New York, New York 10016

**Basic Machining Reference Handbook**

Second Edition

Copyright © 2001. Printed in the United States of America. All rights reserved.  
This book, or parts thereof, may not be reproduced, stored in a retrieval system,  
or transmitted in any form without the permission of the publishers.

Composition by Interactive Composition Corporation.

1 2 3 4 5 6 7 8 9 10

## PREFACE

*This book presents the principles of basic machining; it also summarizes the major considerations that are involved*

The original version of our book, *Basic Machining Reference Handbook*, makes the common sense point that for any production part, the first part must be made. That remains axiomatic, which, in turn, requires utilizing basic machine tools or machines providing the functions of the basic machines to make that part. Moreover, any replacement part must also be made using basic machine tools or machines combining their functions.

In other words, any production machine tool or machining center is an advanced version of one or more of the basic machine tools. It may feature computerized or numerical control, yet it must have the ability to perform all of the functions that the comparable basic machine or machines furnish. Thus while it may sport more accurate screws, ways, or other precision attachments or parts that facilitate set-up and production and enhance production accuracy, it must still provide all of the machining capabilities that the comparable basic machine tool or tools perform. Keep in mind that each of the precision attachments or parts added to the more advanced machines to facilitate set-up and machining and to enhance accuracy must be themselves produced using basic machine tool functions.

Since numerical control was introduced in the late 1950s and early 1960s, advances in the art and in the capabilities of computers have added dimensions that allow the parts designers or engineers, programmers, and machine or machining system operators to quickly, easily, and accurately select the speeds, feeds, cutting tools, coolants to use, and cutting tool paths. But whether these requirements are built into the machines or systems used or must be input by the operator, they must be provided. Consequently, it is helpful if not absolutely essential that those designing or producing a part understand how to calculate these

parameters, if only to help ensure that canned routines provide the proper values for the part to be produced.

Because of the significant advances in the application of computer control, monitoring, and information tracking functions of computerized controls, we have expanded the chapters dealing with N/C and computerized operations. Nevertheless, in this update of our book we still emphasize the key consideration that, however advanced, productive, and sophisticated a production machine or machining system may be, it still must provide the basic machine tool functions, even if it can furnish these at accuracy and quality levels difficult or impossible for basic machines to maintain. Those advances give more meaning to or original information and prompted us to provide addition speeds and feeds tables, general trouble-shooting concepts, and a basic review of relevant computer terms and applications.

We've also included tips on checking material hardness and ensuring that temperature variations won't put parts out of tolerance.

It is easy to forget that all of the high-tech medical, telecommunications equipment, television and cable systems, computers, embedded microchips, numerical controllers, digital cameras, printers, scanners, and hand held devices such as cellular phones and pocket-computers and items used in industry are only made possible by these very same basic machine tool functions—see the DEDICATION page.

Nevertheless, the book remains a basic reference, and the points made in the original version remain valid. Consequently, with the exception of the foregoing, including added machining guides, cost-per-cut calculations, charts and tables, and new photos, basic content and approach remain much the same as in the original book. It still presents the principles of basic machining and summarizes the major considerations involved, although it adds a bit more detail on many of these.

Basic machining in the past and on a job-by-job basis is a dynamic, discretionary, artful process based on ever-changing machines and methods with fundamental principles that, if not immutable, then change at a nearly imperceptible rate.

The dynamism is most apparent in the scenarios conjured up by the “Spirit of Machining Past and Present”:

*Historically, machine tools and the cutting tools and tooling, attachments and accessories, cutting fluids, and supporting equipment—measuring instruments, lead-faced hammers, workholding devices, etc.—that make up a basic machining process are being inexorably changed, improved, refined, altered, combined, **made** obsolete, bypassed, and otherwise rendered constantly different.*

But that's also true *in the shop this very day and every working day:*

*Every job can be done in more than one way: The machinist can perform some steps—drilling, milling, tapping, etc.—on more than one type of machine and vary the sequence of steps a number of ways while still efficiently and economically producing a part that satisfies the criteria posted on the “recipe” for the part, the blueprint.*

*Moreover, the part can be machined manually or automatically on N/C or CNC machines or machining centers, or any mix of combinations of these basic choices.*

Yet the basic principles do not change much.

The latter consideration provides the rationale for this book: It is essentially a “machine-shop generic cookbook,” a series of “machining recipes” designed to serve as a sort of parts-and-tooling “string around the finger,” “picking list,” or a parts-and-tooling BOM or bill of materials, complete with a list of steps to be performed and considerations involved.

It is not a tables-and-formulas reference book—*Machinery’s Handbook* fills that function admirably, and we refer to it often—nor is it a text designed for teaching machining, although based upon the use of similarly structured resources in corporate basic machining instructional coursework, we know it can serve as a useful supplementary reference and teaching resource.

The intended purpose of the book is to serve as a comprehensive, yet easy-to-use, memory jog (“What do I need to turn that radius?”) for the experienced and a reference resource for programmers and for others who will not do the machining but do need to know exactly what’s involved in performing a given machining step, series of steps, or complete job (“How do I convert SFMs to RPMs and write the machining program?”).

So if we’ve accomplished what we started out to do, the book is useful to:

- \* *Experienced Machinists.* The various sections consist of checklists that the experienced machinist uses to be sure all bases are covered and that nothing is forgotten or overlooked—particularly when doing something new or something not done for a while on a machine only infrequently operated. The book lists the considerations involved, the steps and functions required, and, where vital, the most appropriate sequences. The machinist does what would be done in any event; that is, develop the sequence and execute it. The only difference is it should take a lot less time than it would if one had to sit down and draw up one’s own checklist or to try to find old job standards or earlier records for machining that or similar parts—the usual alternatives.
- \* *Programmers* (including machinists who program parts). This book includes a checklist of stock and the steps involved along with a review of both basic machining steps and the axes and other factors pertinent to N/C control and operation needed to complete the program. To tie this all together, this book has a series of projects that begin with the steps and procedures for machining the parts manually; it proceeds to illustrate how those same parts are made using progressively more sophisticated computer-aided systems.
- \* *Tooling, Design, and Production Engineers.* The enemy of excellence is not the poor or sloppy—those get noticed quickly—but, instead, the “satisfactory” or the “good enough.” That may at times be necessary (costs more to improve than improvement is worth), but more often it results from IAI effects (“ignorance at the interface”) that result in designs that make machining unnecessarily difficult or expensive. Since the tooling, design, and production engineers share common interests with the machinist—product

quality and consistency, minimal costs, minimal rejects, returns, and field failures, etc.—improved knowledge of what’s involved in turning a blueprint into an acceptable part helps make sure the design doesn’t sabotage that common interest by unknowingly requiring unnecessary or unnecessarily complicated, difficult, and time-consuming steps, setups, taking chips, and secondary operations.

- \* *Information Seekers.* The book can serve as (1) a handy resource for anyone who needs to grasp the essentials of any of the basic machining steps and (2) a library resource for a student or industry member needing to clear up various ambiguous or perhaps forgotten points concerning machining.

The book starts with those machining steps that most often begin the machining process—blueprint reading, laying out the workpiece, determining the machining sequence, assembling the stock, and tooling—and moves through the basic machining operations in a more or less logical sequence: cut-off plus contour band sawing and band machining; turning, milling, drilling, reaming, boring; surface and other grinding.

As the basic “stuff of machining,” metals and heat-treating are presented and reviewed with emphasis upon aspects most important to machining.

The shaper and planer are not presented, in part because of their dwindling use and importance and in part because their functions and applications are limited and straightforward; we could add little or nothing not already in the operating manuals.

We’ve tried to order the material and present it in ways that make subjects logical, easy to find, and easy to use.

We emphasize the “hands-on” concept of developing and refining skills, along with knowledge of metals and machining processes, by means of application.

These techniques were forged in the shop classroom and were refined (by Meyers) during the course of a career commencing with machining, progressing to tool and die making, and branching out to job spotting, sales of machine tools and cutting tools (the whole machining ball of wax). This led to positions as sales engineer, metals and machining consultant, and productivity expert (Director of Productivity) conversant with shops and plants in America and around the world; his career also includes experience with shops and plants in America and around the world and in the classroom (both corporate and university) teaching all aspects of basic machining.

The principles are timeless; the methods and procedures are up-to-date. We sincerely hope we’ve done them justice.

*Arthur R. Meyers, Denver, Colorado*

*Thomas J. Slattery, Chicago, Illinois*

# CONTENTS

	Preface	vii
Chapter 1.	Measurement Standards	1
Chapter 2.	Cut-off	16
Chapter 3.	Turning and the Lathe: Definition and History	43
Chapter 4.	The Milling Machine	100
Chapter 5.	Sensitive, Gear-Head, and Radial Drill Presses	139
Chapter 6.	Grinding	174
Chapter 7.	Steels, Alloys, and Other Materials	238
Chapter 8.	Numerical Control and CNC	267
Chapter 9.	Cost Per Cut in the Computer Age	289
	Index	301

## Chapter 1

# MEASUREMENT STANDARDS

### PREFACE

Measurement is the comparison of what is known to what is not known.

Parts can be fitted to one another without standards—a plug worked until it fits exactly into a hole, for example. But there's very little likelihood that this plug and hole would be interchangeable with other plugs and holes made the same way; for that reason, measurement is necessary, and to be able to make measurements, a standard must be set.

The need for measurement standards has been recognized for more than 5000 years: When the Pyramids were built, the Egyptians worked to a standard of measurement known as *the royal cubit*. Specified as the length of the Pharaoh's arm, the royal cubit was subdivided into hands (the width of his hand), digits (his finger width), and then smaller units to create a progressive scale. This scale was then carefully inscribed on a polished strip of black granite to create a *master standard*.

Black granite is extremely stable and holds its dimensions exceptionally well. It was, however, difficult to shape with the copper tools then available, so the Egyptians used the master standard to make working cubits of wood to check the stone blocks used in the pyramids.

Wood readily abrades, expands, contracts, twists, swells, and cracks with changes in moisture and temperature. Yet those stone blocks for the Pyramids were made and fitted to such precise dimensions that a knife blade cannot be inserted between them. The "secret" is simple: by royal order, every working cubit had to be checked against the master standard—a step known as *calibration*—at specified intervals.



There are several lessons to be learned from this little piece of history:

1. Standards may be arbitrary (the Pharaoh's arm), but that doesn't matter as long as there's a "master" unit to calibrate against (we now use light waves).
2. When things must fit together very precisely, measurement, measurement standards, and a reliable master unit are indispensable.
3. Once a standard is set, the tools used to check the work for conformance to the standard must themselves be periodically calibrated to ensure that *they* conform to the standard.

Measurement has always been important, but precision measurement is only important when things must be very accurately made.

From prehistoric times until early in this century, most of the tools, materials, vehicles, utensils, and other useful things were made by hand, and parts made for one device would not fit or work with another. This is no longer the case: Whereas the Pyramids made precise measurement important to the ancient Egyptians, the concept of "interchangeable parts" does that for modern man. This idea has been around since the late 1700s: A Frenchman named La Blanc attempted it around 1775; however, Eli Whitney has been credited with introducing it in the early 1800s when he produced 10,000 muskets with interchangeable parts—but it didn't become truly practical and successful until well into the twentieth century.

As one historian put it, the concept of interchangeable parts is like love—the basic idea is simple, the practice is not:

1. *American physicist Albert Michelson* invented the interferometer (uses light wave interference patterns to measure to millionths of an inch) in 1893.
2. *C. E. Johansson* made the first set of modern gage block sets (81 blocks that could be combined to accurately measure 125,000 separate lengths) in 1896.
3. *Major William Hoke* found a way to make gage blocks inexpensively in 1918, but they didn't really reach the shop and factory scene until the 1930s. After that, it was "Katy bar the door"—gage blocks made calibration practical, calibration made precision measurement practical, and precision measurement made interchangeable parts practical: Abracadabra, an industrial revolution, leading to the high-technology space age of today.

Measurement, standards, and calibration may all be ancient ideas but their practice is strictly modern, their power is quietly awesome, and their importance is immeasurable.

## THE GAGE BLOCK

This measuring tool, accurate to within a few millionths of an inch, makes possible the ultraprecise standards in use today (Figure 1.1). Before interferometry and optical flats teamed up with the gage block to bring high precision to the shop, the



Figure 1.1. Rectangular and square gage blocks and their accessories. A micrometer gage block check set is at the lower right corner. (Courtesy of DoAll.)

“master” standard of length was the international meter in the form of a meter-long stick kept in France under very secure conditions. The trouble was that it was next to impossible to accurately duplicate it so it could serve the world of manufacturing.

C. E. Johansson was the man responsible for developing gage blocks. Not only did he help set the standards of temperature for manufacture of gage blocks but he also developed the arithmetical progression that led to the sets now used. His original contribution was a set of 81 various sizes of blocks that could be used in combinations to make up to 125,000 distinct dimensions. This made “Jo blocks” commercially practical. His method produced gage blocks, accurate to within millionths of an inch, with excellent surface finish—they had to be flat and uniform enough so that two or more blocks could be pressed together so tightly they would adhere to each other, a procedure known as *wringing the blocks*.

While Johansson’s innovations made gage blocks practical and effective, the extensive handwork involved made them quite expensive. Large companies could afford them but smaller ones were limited in what they could buy. Major William E. Hoke overcame that problem when, in 1918 while he was with the Bureau of Standards in Washington, he developed the first automatic lapping machine for manufacture of gage blocks.

Major Hoke took an old sewing machine, added special fixtures, and perfected a method of lapping the blocks that sharply reduced their cost and put gage blocks within the reach of every machine shop.

When Major Hoke lapped the original set of gage blocks, he noticed the center of each was slightly higher than the rest of the block. To eliminate the high spot, he drilled a hole in the center, and now gage blocks made that way are called *Hoke blocks*. Lapping machines are now so accurate that the problem does not exist; utilized with accessories to tie the blocks together, the holes—and the name—remain as reminders of his contribution.

Why have blocks accurate to within millionths of an inch when tolerances in the typical shop usually run a few thousandths? The reason is quite simple: gage blocks are used mainly for inspection and to calibrate calipers, micrometers, and other working measurement tools. The rule of thumb is that gages must be at least 10 times more accurate in dimensional tolerances than the tool or part they are measuring, and the setting gage (the gage block) must be 10 times more accurate than the measuring tool: For example, if the part to be made has a tolerance of 0.001 in., then the gage used to check that part should be accurate to at least 0.0001 in. In turn, to check the accuracy of that gage, a gage block should then be 10 times closer, i.e., 0.000010 in. or 10 millionths of an inch.

## Wringing Rectangular Blocks

1. Clean gaging surfaces.
2. Start by overlapping the gaging surfaces about 1/8 in.
3. Press the blocks lightly together, then slide one block over the other.
4. You will now notice the blocks adhering to one another.
5. Continue sliding the blocks smoothly until the surfaces are evenly matched and mated.
6. When the blocks are together in this manner, all air between them is squeezed out and the size of the combined blocks is exactly the sum of those wrung together.
7. Square blocks are wrung together in the same manner as rectangular blocks.

Accessories are provided for both rectangular and square blocks to build required heights. With special torque wrenches or screws, air can be squeezed from the blocks to give accurate sizes.

## PLANES OF REFERENCE (FLATNESS)

While standards of measurement are critical for accuracies of length measurement, another standard that is needed to aid in determining length or height is a flat surface or “plane.” Without an extremely accurate surface, precision measurement is impossible. To achieve accuracy, a surface plate must be used.

The first surface plates were made from cast iron and were given their accurate flat surfaces via the three-plate method. The procedure is:

1. Three plates are cast with ribbed construction for rigidity.
2. They are then rough-machined along the top surface and edges.
3. Cast iron has to be normalized so the material is stable. It can be compared to green wood. If a cabinet is made from green wood, movement will be heard because as the moisture it still contains escapes and the wood dries out, size changes are taking place.
4. To normalize cast iron, the old practice was to let it normalize while exposed to the air for a year or two, which stabilized it. Today, special furnaces speed up the stabilization process considerably.

5. The plates are numbered 1, 2, and 3 and plates 1 and 2 are scraped and matched together by blueing the surfaces and placing one on the other to determine the high spots. When they are matched, then the next step takes place.
6. Plates 1 and 3 are matched and plate 3 is then scraped to match plate 1 by blueing plate 1 and removing the high spots from plate 3. For this step, plate 1 becomes a gage plate.
7. Now, plates plate 2 and 3 are scraped together. This now makes plates 2 and 3 flatter than plate 1. Plate 2 is used in this step as the gage plate and is used to get plate 1 flat with plate 2.
8. Plate 1 is scraped by blueing plate 2 and removing the high spots from plate 1 to make it match with plate 2.
9. Next, plates 1 and 3 are scraped together to bring them into flatness with each other.
10. Finally, with plates 1 and 3 matched plate 3 is used to control the flatness of plate 2 so plate 2 is scraped to match plates

It can be seen that the above procedure, complicated to describe and very good for its time, is very time-consuming and expensive.

In addition, cast iron or steel will rust if protection is not provided, and if a part falls on its surface, it will rise and change size. When that happens, the plate must be rescraped to ensure accuracy.

Granite has mostly much replaced cast iron as the basic material for surface plates and their accessories because of its stability—it has, in effect, been stabilizing in the ground for millions of years—and cost savings. Because of the nature of the material, a part dropping on granite will not cause its size to change even if chipping occurs.

In addition, granite can be machined with diamond tools to accuracies of within millionths of an inch with virtually no problems. This eliminates costly handling and helps make the surface plates comparatively inexpensive.

The hardness range of granite is equal to about Rockwell 80 on the C scale, much harder than any cast iron plate, so the life of the plate is extremely long.

The accessories that are available for surface plates include squares, V blocks, parallels, and other devices that help keep parts perpendicular to the surface of the plate. Perpendicularity must be as accurate as the plate itself to maintain gaging standards.

## OTHER METHODS OF MEASUREMENT

### Air Gaging

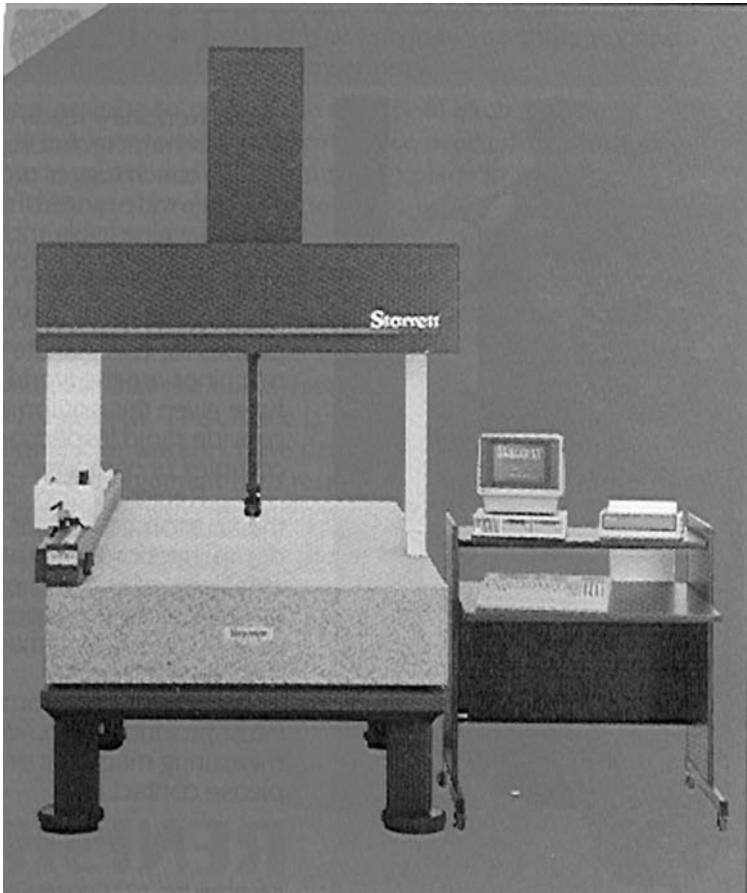
Air gaging, known as *pneumatic measurement*, is a very popular type of gaging for high-production work. This gaging is accomplished by changes in a calibrated flow as it responds to changes in a part's feature.

In addition to low cost, its advantages are that it eliminates metal-to-metal contact and provides power amplification and linear amplification. There are several types of air gaging, but since the process is mainly used for production gaging, it is only touched upon lightly here.

## Coordinate Measuring Machine System

This is a method that has become very popular for checking parts in a three-dimensional mode, the x, y, and z axes. It can measure accurately to within 50 millionths of an inch and can be computerized to facilitate recordkeeping, calculations based on the measurement readings, and similar uses.

The measurement is determined by a series of dark and light bands called the *Moiré fringe pattern*. The principle of this system is a series of etched lines on a reading head that is placed at an angle to another series of lines located on a spar or other main support. A lamp passes light through a collimating lens that converts the light into a parallel beam. When this light strikes the fringe pattern, it is reflected back to a photoelectric cell, which converts the fringe pattern into an electrical signal. The reading head moving longitudinally will move the band



*Basic CMM [Coordinate Measuring Machine] complete inspection in minutes and automatically provides statistical process control data. (Courtesy of The L.S. Starrett Co., Athol, MA).*

vertically on the face of the spar and it will be picked up by the next photoelectric cell, creating yet another signal. These signals are transmitted to a digital readout display box where they indicate the head travel at any point. Depending on the direction of movement, each signal will increase or decrease the reading on the digital readout box by 0.0001 in. When used for measuring parts, orientation of the part is not necessary because the fringe lines compensate for that. Therefore a part can be read in three axes when located on the table of the coordinate measuring machine.

The same principle is used on digital readout boxes mounted on machines. In that use, however, the scales must be located and clamped securely to the machine tables and slides for accurate repeatable measurements.

When the earlier Coordinate Measuring Machines were introduced, an operator manipulated the part under the probe and then used a print to determine what to measure. While many of today's machines are still manually operated, direct computer control is the method now most often used. The operator develops a measuring program directly from a Computer Manufacturing Program and then lets the computer conduct the measuring sequence.

The CMM must be rigid with a heavy granite so that the environment has little effect on its stability. Steel or cast iron can be affected by temperature changes and while a controlled environment can compensate for temperature deviations, any need for that can be eliminated by using a stable material such as granite, which can be used in the plant with little effect on part measurement due to temperature changes. Even with a stable base material, when used on the shop or plant floor, the CMM still should be housed in an isolated enclosure.

The probe is one of the most important elements of a CMM. The tip contacts the part surface and when it does it registers the coordinates and these are displayed on the computer screen. The strain on the probe tip provides the coordinate position and this information is transferred to the computer by an electric circuit when the probe contacts a part. Some CMMs use an X, Y, and Z temperature compensation featuring a probe that feeds back temperature signals for both the workpiece and the surrounding air, allowing the machine to compensate for higher or lower temperatures.

## Optical Flats and Monoliths

Optical flats are devices made of lapped quartz that are used to read light-wave interference patterns to measure the degree of flatness of a surface. The surface must have a finish of  $2\ \mu\text{in.}$  or better to successfully use an optical flat to check it. In addition, the wavelength of the light reflecting through the optical flat must be known.

The reason the optical flat must have a fine finish is straightforward: When an optical flat is laid on the surface of the part to be finished, it has a film of air under it; that air is important to permit the reflection of the fringe lines used to measure the surface.

When the light from the source is projected through the flat, the amount of air under the flat can be measured by counting the number of fringe lines shown. If

there is too much air under the flat, there will be so many fringe lines that it will be impossible to distinguish them.

If all the air is wrung out from between the flat and the part to be measured, no fringe lines will appear. Ideally, four or five lines will show through the flat when measuring.

If the surface finish of the part is rougher than  $2\ \mu\text{in.}$ , the amount of air under the optical flat will be excessive because the roughness of the part holds the optical flat too high and no readings will be possible.

A light source called a *monolite*, using helium as its light source, is widely used; the wavelength of helium is  $23.2\ \mu\text{in.}$

Using an optical flat made from quartz and with a surface finished to  $1\ \mu\text{in.}$ , the light source can be shone through the optical flat. When the flat is laid on a finished part such as a gage block or valve seat and the light is shone through, a sine wave occurs and two bands are reflected, one a dark band and the other a light band.

Each band occurs every  $\frac{1}{2}$  wavelength, with the dark band plainly visible. The dark band is called a fringe line. Totalling each  $\frac{1}{2}$  wavelength of the sine wave gives  $23.2\ \mu\text{in.}$  Only the dark bands are used, however, so the distance between each dark band is  $\frac{1}{2}$  wavelength or  $11.6\ \mu\text{in.}$  This length is always  $11.6\ \mu\text{in.}$  between each band regardless of how many can be seen, whether 3 bands or 10 bands are displayed.

If the bands are perfectly parallel to each other, the surface is flat. If they bend, much as lines bend on a topographical map when heights of mountains are shown, then the part is not flat and has high spots. These high spots can be measured by estimating the amount of bend the fringe lines have between each other (Figure 1.2).



*Figure 1.2. That absolute and immutable standard—the wave length of light—becomes a practical measuring tool for industry through the use of optical flats and monochromatic light generators.*

This method of checking flatness is extremely accurate, inexpensive, and easy to use. It is used when calibrating gage blocks for certification and for inspecting valve seals, micrometer anvil and spindle surfaces, and any other part that needs a restricted surface for efficient operation.

## Comparators

There are two types of comparators: electronic and dial. Both types are methods for comparing an unknown with a known. When using either comparator, it must be set with a gage of known dimension, such as a gage block or a standard already calibrated. Then, when the part is measured, it can be determined whether or not it falls into an acceptable range.

The major difference between various types of dial indicators and of electronic indicators is their magnification. The use of one or the other is determined by how accurate the part to be made must be.

The dial indicator (Figure 1.3) is amplified by one of three methods, the lever, gear train, or pointer length.



*Figure 1.3. Array of gages, including gage blocks, combination square, surface gage, and other tools. (Courtesy of DoAll.)*

When using the lever principle, the amplification is provided by the ratio of the distance the contact point moves to the length of the pointer. If this ratio is 10:1, then for each 0.1 in. movement of the contact point the pointer will move 1 in.; if the ratio is 16:1, each 0.01 in. movement of the contact point will produce a 0.16 in. movement in the pointer; and so on for other ratios.

Levers have largely given way to gears. When developing ratios of gears, the same principle applies. A 10:1 ratio of the driven gear to the driving gear will reflect the same as the lever ratios.

If the pointer length is extended, the amplification increases, providing greater discrimination in the reading. Sensitivity increases as discrimination increases. Dial



indicators are inexpensive and quite reliable. When higher amplifications are necessary and greater accuracy is required, then electronic gages are recommended.

## STEEL RULE SCALES, VERNIERS, MICROMETERS

The basic machining workaday “tools of the trade” are, in ascending order of sophistication and reading accuracy, graduated steel rule scales, vernier calipers, and micrometers.

Other working measuring devices such as height gages, telescopic gages (Figure 1.4), combination rules and squares, and depth gages measure in much the same way, so a mastery of the three basic measuring tools readily transfers to these instruments.



*Figure 1.4. Telescopic gage measuring an internal surface. (Courtesy of DoAll.)*

### Steel Rule Scales

Steel rule scales come in styles that vary from narrow, flexible strips to thick, rigid, and wide with such features as hooks and adjustable clips that help position the scales for reading, scribed scales in decimal or fractional subdivisions—or both—and lengths ranging from a few inches to several feet.

The unofficial “standard” steel rule scales—the tools used every day in most, if not all, machine shops in the United States—are the six-inch-rule scales. This rule is has four sets of subdivisions or “scales” (hence the name), two inscribed on the opposite edges of one side and two on the other.

The scales on the edges of one side are divided into subdivisions of  $\frac{1}{8}$  in. and  $\frac{1}{16}$  in., whereas the subdivisions on the other side are  $\frac{1}{32}$ -in. and  $\frac{1}{64}$  in. The body of

the thin, flexible six-inch steel rule is about  $\frac{1}{2}$  in. wide, whereas the thicker and stiffer version is about  $\frac{3}{4}$  in. wide.

The flexible rule is useful for checking lengths on relatively gentle curving surfaces, whereas the stiff one is the choice for measurements and such steps as layout scribing on flat surfaces and other uses in which flexing might result in misreadings; both are used to make the quick rough-and-ready checks of dimensions necessary to prepare for and set up and run a job.

In general, the accuracy of readings using steel scales depends more on eyesight, steady nerves, and experience than the accuracy of the scales, so a dimensional reading taken with this type of scale is referred to as a “reference” rather than a “precision” measurement. Although some experienced machinists can obtain accurate readings with them, that’s the exception and not the purpose of scales. They’re intended for approximate readings, since that’s all that’s called for in the preparation and set-up stages; taking precise readings would be a waste of time—precision they’re not, handy and very useful they are. When the work reaches the stage where precision readings become important, more accurate instruments do that easier, faster, and better and should, or rather, *must* be used.

Humble as they are, steel rule scales are used alone or in conjunction with layout tools, height instruments, depth instruments, protractor instruments, and centering instruments. In fact, they sometimes become part of other tools, as for example:

1. One of the most popular layout tools is the combination square. This tool features scales in various sizes and lengths, with the most common being 6 in. and 12 in. The combination square consists of a blade and a square head. The head provides a right angle reference that can be used, for instance, to check parts on a surface plate for squareness and height. It usually comes with two attachments, (1) a protractor head for checking and laying out angles and (2) a centerhead, both of which are interchangeable with the square head. The protractor head consists of a rotatable, graduated turret in degrees and minutes from 0 to 90 and from 90 to 180 degrees.

The center head provides an accurate, fast, and very convenient way of finding and marking the centerpoints on the ends of round stock for machining. When placed on the end of the round stock while the V portion rests against the circular end of the stock, the scale will lie along the center line of that workpiece end. Just draw two lines 90 degrees apart on the workpiece; the two lines cross at the center.

2. A simple but handy depth scale consists of narrow ( $\frac{1}{4}$  in. wide) scale and head with graduations of 30 degrees, 45 degrees, and 90 degrees and a sliding positioning clip. The depth of blind holes, shoulders, and similar features and angles can be measured and laid out with this rule. It is, however, less accurate than a protractor and head.
3. Dividers, inside and outside calipers, and trammels are all used in conjunction with scales. Dividers are typically used with scales for laying out circles and distances. Calipers have been used with scales to check parts turned on lathes since the beginning of the machine tool age. Used for convenience in the roughing stages prior to more precise instruments

for final size, these calipers have no scales of their own—the reading is “transferred” from the calipers to the steel rule scales.

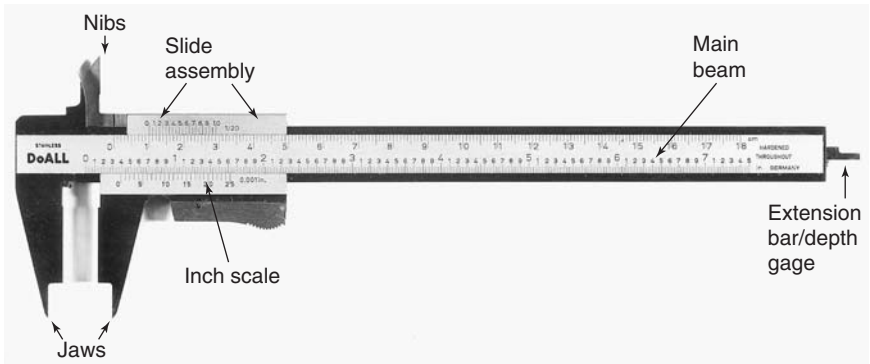
## Verniers

In 1631, a Frenchman named Pierre Vernier invented a system of scales that capitalizes on the ability of the human eye to accurately align two lines so they coincide. The basic idea is simple: Inscribe a set of scales (the “vernier” scale) with units smaller than a standard subunit of the inch (or metric unit) by a precise, known amount. This permits reading dimensions that cannot be detected by the unaided human eye but that can be measured with an accurate vernier instrument. Vernier instruments give a greater degree of accuracy than scales and come in various forms.

Today the vernier instruments have graduated into electronics, making them even more versatile and easier to read than before.

## The Vernier Calipers

The basic vernier calipers consist of a beam (the graduated center of the caliper), a fixed jaw (part of the beam), a movable jaw, and a vernier scale plate (Figure 1.5).



*Figure 1.5. Vernier slide rule with English-unit scale at bottom and metric-unit scale at top. (Courtesy of DoAll.)*

It is not difficult to read the vernier scales. Almost all hand-held vernier scales come with the vernier itself divided into 25 divisions. The readable length of the scale is 6 in., divided the following way:

1. *On the standard unit scale*, each inch is marked with a large line and numbered 1, 2, etc. Between each number, the scale is divided into 10 divisions with smaller lines numbered from 1 to 9. Between those numbers are three smaller divisions; each of which represents 0.025 in.
2. *On the vernier unit scale*, each inch is divided into 25 subunits. The vernier scale slides along the main beam and allows the smaller divisions to be broken into smaller units—0.001 in. to be exact.

The vernier scale is read in the following way:

1. Read the large numbered division first, 1 in., 2 in., etc.
2. Next, read the small numbered division.
3. Now read one of the three smaller divisions located after step 2. Each of these divisions represents 0.025 in. If, for example, you pass two divisions, 0.050 in. would be the sum.
4. Finally, looking at the vernier, see what line of the vernier lines up with a line on the scale of the main beam. Then count the vernier lines from 0: that gives the thousandths to be added to the above measurement (Figure 1.6).

Vernier height gages are used to check and compare heights and for layout work. Some gages stand as high as 36 in. and have vernier scales that read in increments of 0.050 in. rather than the more unusual 0.025 in. divisions, making them easier to read.

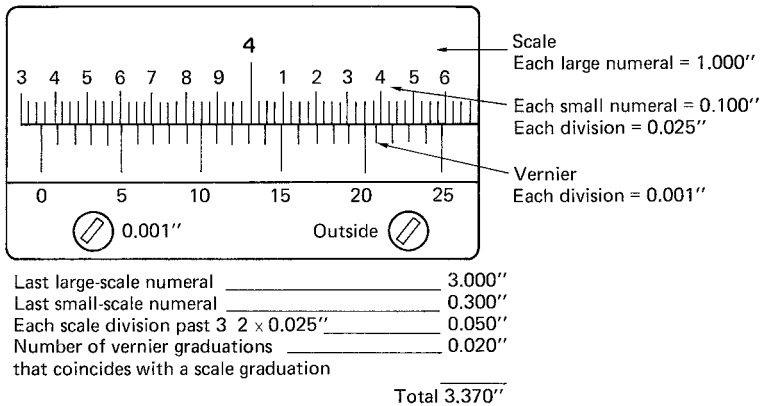


Figure 1.6. How to read a vernier gage with 25 divisions. (Courtesy of DoAll.)

A dial vernier caliper is very popular because it makes reading to thousandths much easier than reading off the beam. The large size is read directly off the beam with the thousandths read off the dial; some dial height gages also are available.

Vernier calipers and height gages with digital electronic readouts that make them very simple to use and much more accurate than manual units—parallax does not occur and the measuring jaws always set at the same pressure—are available today. Their costs are greater but the speed of reading and accuracy more than make up for the higher initial cost.

## Micrometers

Micrometers are the machinist’s stand-by for precision work. They are easily transported, very easy to read, hold accuracies to within 0.001 in., and, with a small vernier reference on the side of the barrel, can provide slightly greater accuracy

than one without a vernier. Micrometers range in size from 1 in. to 90 in., although the most popular ones are from 1 to 6 in.

A micrometer consists of a frame with the base holding the anvil that is pressed or screwed into the frame with a barrel fastened to the frame (Figure 1.7). The barrel has the graduations inscribed on it and houses the nut in which the screw turns to position the movable face. The graduations on the barrel are from 1 to 9, each one representing 0.100 in. Between each division are three divisions, each representing 0.025 in. The full travel of a micrometer is 1 in.

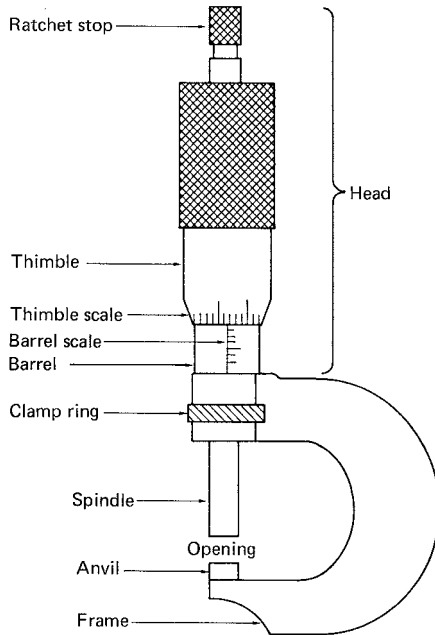


Figure 1.7. Functional features of a micrometer. (Courtesy of DoAll.)

The thimble—graduated into 25 parts, each part representing 0.001 in.—is attached to the screw.

There are 40 threads to the inch in the thimble and barrel; dividing 1 in. by 40 the space (pitch) between each thread is 0.025 in. Thus one revolution of the spindle advances or retracts the movable face exactly 0.025 in.

To read the micrometer, note the exposed number on the barrel; next read the number of small divisions past that number; and finally, read the division on the spindle. Add them up to get the correct size (Figure 1.8).

Digital micrometers that read the correct size directly, providing a much easier way of measuring, are now in use, but they are much heavier than standard micrometers and awkward to carry around.

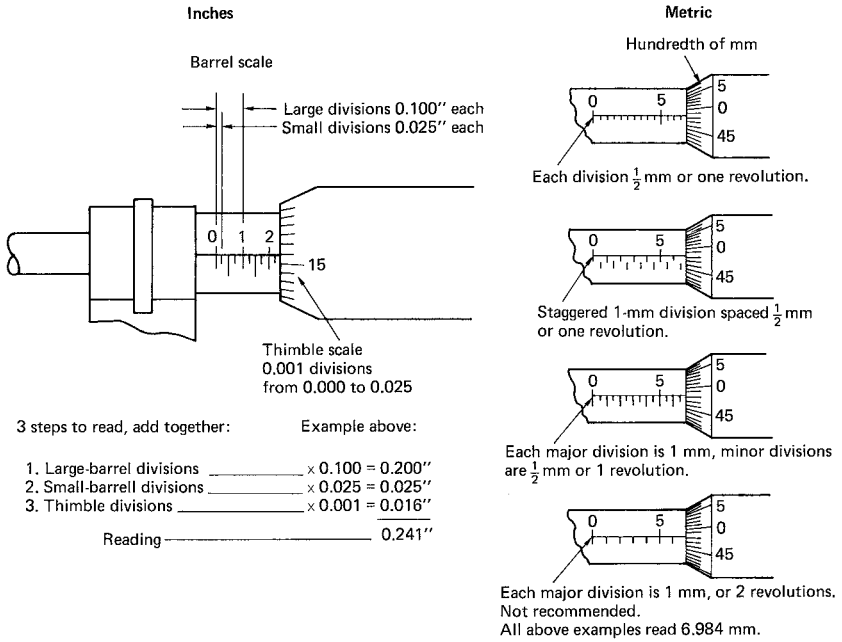


Figure 1.8. Reading a micrometer, in both English and metric units (Courtesy of DoAll.)

Also in use are electronic digital micrometers with microchip memories that give them calculating and printing capabilities. They can be used to keep dated quality control, tool calibration records, and other data that make them invaluable for operations where these are important considerations.

Since the movable face on all micrometers travel only 1 in., all large micrometers are read by adding the micrometer reading to the frame size.

## *Chapter 2*

# **CUT-OFF**

### **INTRODUCTION**

Every machining operation is performed on a blank, slug, or other workpiece rough cut to dimensions normally somewhat larger than finish sizes to allow for stock removal. Consequently, regardless of whether this operation is performed in the shop or plant or elsewhere in the facilities of a supplier, cut-off is the first step in any given machining operation.

The cut-off machine may be used to create just one piece or large numbers of blanks to begin the process of making a part. The part may be a turning screw, a gear or other integral part for a machine, an attachment or fixture for drilling holes, a die for stamping, or an aluminum extrusion. The cut-off machine is used to meet hundreds of different needs and sizes and shapes that vary from round, square, hex, angle, rectangular, or asymmetrical solids and tubing, and various types of beams. In addition these shapes may be produced in any of the hundreds of different material groups that can be parted on a given cut-off machine, each one requiring different speeds, feeds, cutting blades, and coolants to provide cost effective output.

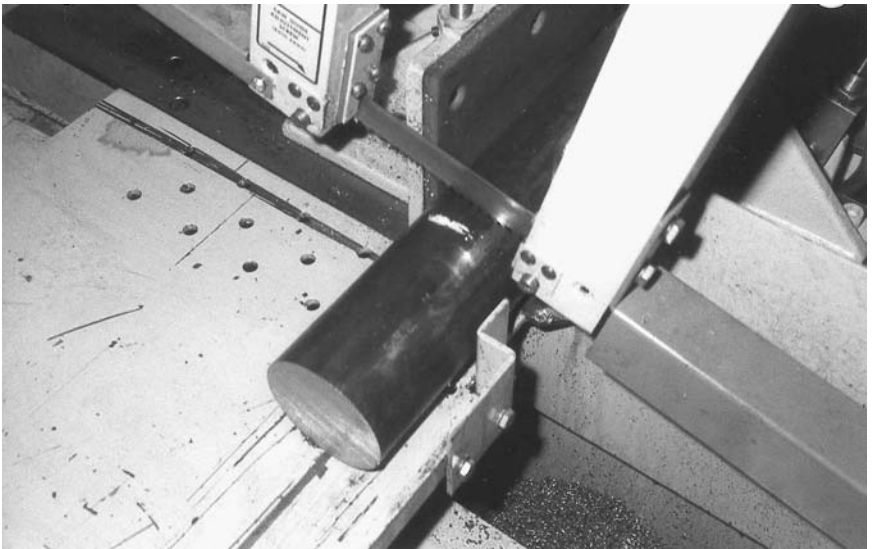
There are many types of cut-off machines: simple machines used to cut one piece at a time, production machines for many pieces, machines that can be used to make angular cuts, plate cut-offs to cut large plate materials. The machines may be horizontal cut-offs, vertical cut-offs, turning cut-offs, abrasive cut-offs, and circular saw cut-offs. Each of these types is used extensively, depending on the type of material to be used in production.

The cut-off operation also may be performed on a variety of other types of machines, including lathes, automatic screw machines, shears, presses, flame cutters, and various “high-tech” systems such as lasers, water jets and abrasive water jets, electro-discharge and electro-chemical machines, etc. Nevertheless, most cut-off work—especially high production jobs—is performed using some type of

saw—band saw, circular saw, abrasive saw, or hacksaw. Band saws normally take a smaller kerf than the alternative cut-off sawing systems, thus it is the preferred method where it is important to minimize metal removal. Holding the kerf to a minimum can mean additional blanks from a given cut-off round or other shape.

## SAWING

Cut-off of workpiece stock on a saw is normally the first step in a machining operation (Figure 2.1). A blank of adequate diameter or configuration for the part must be cut to length before other machining steps can be performed; this is true whether the job calls for one part or the manufacture of thousands of parts.



*Figure 2.1. Round of stock mounted in light-duty cut-off band machine (left-hand side), ready for parting. Operator has used chalk to mark point of cut that will result in workpiece blank of proper length. (Courtesy of DoAll.)*

## THE CUT-OFF BAND SAW

Band saws are a popular method of parting material; they are simple to operate, use inexpensive cutting tools, can handle a wide range of workpiece diameters and sizes, remove the least amount of material of any of the various types of saws when parting stock, and are relatively inexpensive. The cut-off band saw is used to meet hundreds of different needs and, of course, sizes and shapes, which can vary from round through rectangular, tubing, hex, and angle to various types of beams. In addition there are hundreds of different material groups that may be cut on the cut-off machine, each one requiring different speeds, feeds, cutting blades, and coolants in order to be cost effective for the company involved in this work.

Cut-off band saws come in two basic types (Figure 2.2).



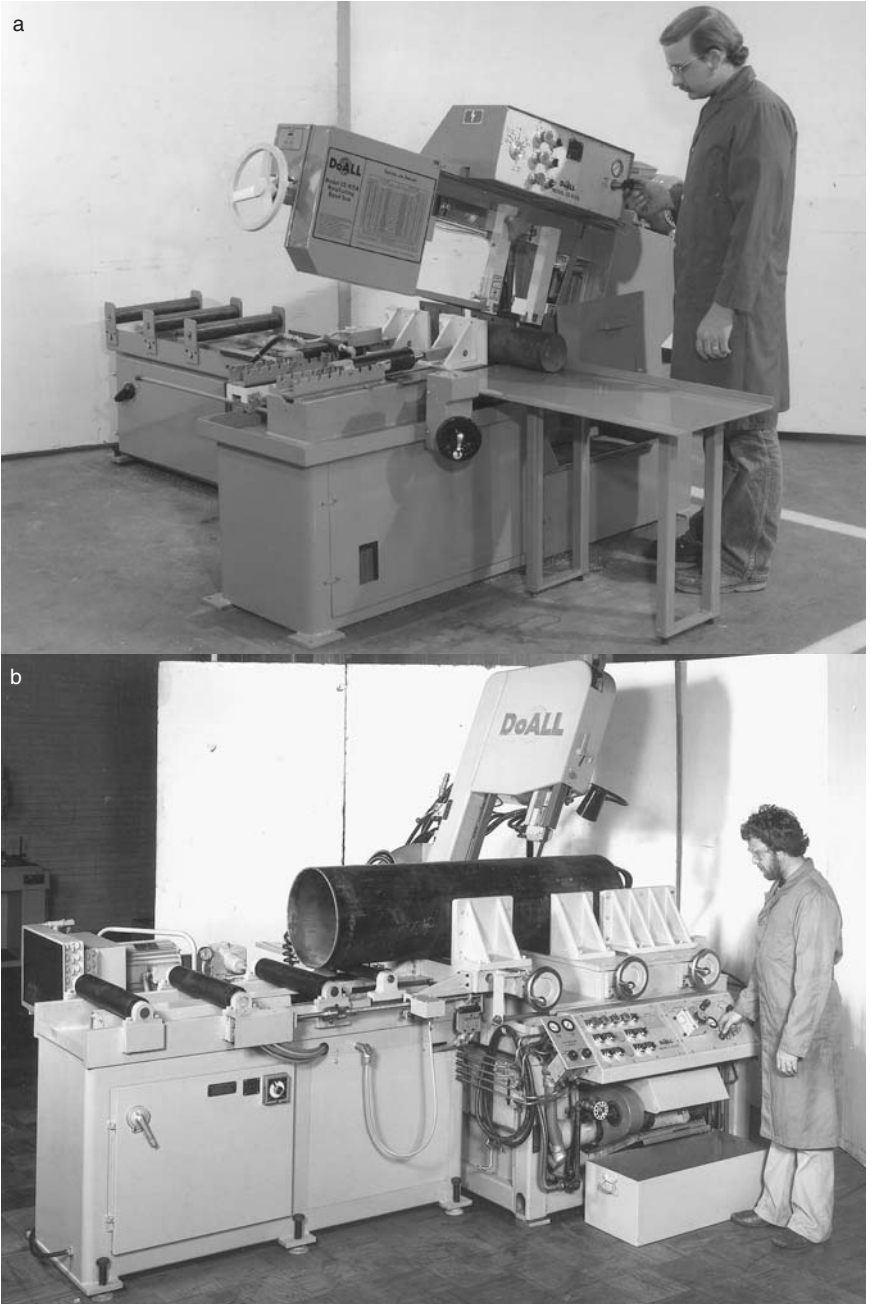


Figure 2.2. A horizontal and vertical cut-off band saw. Cut-off band saws may be (a) “horizontal” or (b) “vertical,” small or large, simple or sophisticated; they range in power and rigidity from light to heavy-duty production models. All serve the basic function of cutting stock to length, and, in the case of most vertical machines, making angle cuts. (Courtesy of DoAll.)

When used for contouring parts in which substantial portions of the workpiece are to be removed, the contour band saw removes the unwanted portions of the material in chunks rather than reducing them to chips; these solid pieces often can be used to make other parts; but even if they are not used to make other parts, they at least can be sold as scrap.

Contour band saws can be used for friction cutting, providing an efficient way to remove trim or cut difficult-to-machine materials.

In addition, filing or band polishing can be done with a few simple attachments.

Speeds for band machines are calculated in surface feet per minute directly off the indicator located on the machine and in just about all of the band cut off machines, the speeds can be changed very easily by a variable speed arrangement.

There is a rule-of-thumb that permits making appropriate band speed adjustments when shifting from one size workpiece to another in the same material and shape.

Adjusting band speed for material thickness

Increase band speed by:

15% when cutting  $\frac{1}{4}$ -inch material

12% when cutting  $\frac{3}{4}$ -inch material

10% when cutting  $1\frac{1}{4}$ -inch material

5% when cutting  $2\frac{1}{4}$ -inch material

5% when cutting 3-inch material

Feeds usually are adjusted by utilizing the weight of the head carrying the blade through the material most often hydraulically controlled. On vertical cut-offs the head is fed through the material by hydraulics, the blade must move horizontally while the weight of the head is exerted vertically and so cannot be utilized the blade must be force-fed.

The blade speed is set directly in surface-feet-per minute while feed is determined by the cross-section of the material to be cut. Therefore when speeds and feeds are recommended for a given material and shape, the cross section must be computed for optimum results. To compute the cross section for a round to be cut, the formula is

$$a = \pi r^2$$

An easy and accurate way to determine the area in square inches for various sizes and shapes of materials is to refer to *The Machinery's Handbook*, which carries tables for all of them. Blade manufacturers also typically provide extensive band speed guides for most commonly machined materials.

The computer can be a great help reducing the time and effort required to achieve each of the objectives listed above. In addition numerical control can reduce the handling time required thereby reducing the cost of producing the part. Below is an example of using the computer to determine the cost-per-cut for typical job. The example is the program for cutting a 5-inch diameter 303 stainless steel round utilizing a specific machine with appropriate settings, adjustments, blade, coolant, etc.

(A sample of a computer read-out program—Courtesy of DoALL)

Welcome To The DoALL SAWING DATABASE  
Production Power Sawing Analysis

You have 3 options:

1. Press 'Enter' to begin.
2. Enter a short I.D.
3. Enter N/A (name/address).
4. Select Material.

Main Material Group

- |                          |                         |
|--------------------------|-------------------------|
| 1. Carbon Steels         | 1008-1095               |
| 2. Free Machining Steels | 1108-1213               |
| 3. Manganese Steels      | 1320-1345               |
| 4. Nickel Steels         | 2317-2517               |
| 5. Nickel Chrome Steels  | 3115-3315               |
| 6. Molybdenum Steels     | 4017-4068               |
| 7. Chrome Moly Steels    | 4130-4150               |
| 8. Nickel Chrome Moly    | 4317-4340               |
| 9. Nickel Moly Steels    | 4608-4820               |
| 10. Chrome Steels        | 5045-5160 (50100-52100) |
| 11. Chrome Vanadium      | 6117-6152               |
| 12. Nickel Chrome Moly   | 861S-8750               |
| 13. Silicon Steels       | 9255-9850               |
| 14. Miscellaneous        | *                       |
| 15. Hi-Speed Tool Steels | *                       |
| 16. Tool and Die Steels  | *                       |
| 17. Stainless Steels     | *                       |
| 18. Copper Base Alloys   | *                       |
| 19. Nickel Base Alloys   | *                       |
| 20. Titanium Alloys      | *                       |

Enter the Number of your choice?

Material Sub-Groups

1. 201 202 302 304 130-190 Bhn
2. 303 303F 150-200 Bhn -
3. 308 309 310 330 160~1~20 Bhn
4. 314 316 317 160-220 Bhn

- 5. 321-347 154-200 Bhn
  - 6. 410 420 420F 140-185 Bhn
  - 7. 416 430F 155-195 Bhn
  - 8. 430 446 170-215 Bhn
  - 9. 440A B C 160-190 Bhn
  - 10. 440F 443 175-215 Bhn
  - 11. 17-4PH 17-7PH 150-360 Bhn
- [numbers 12-20 not used in program]

Number of your choice?

Part Shape

- 1. Round Solid
- 2. Rectangular Solid
- 3. Round Tubing
- 4. Rectangular Tubing
- 5. Hex Solid
- 6. Angle
- 7. U-Channel
- 8. I-Beam, H-beam

Enter the number of your choice

Production Power Sawing Analysis

- 1. Material Group—Stainless Steel  
Material Type—303 303F 150-200 Bhn
- 2. Material shape—Round Solid
- 3. Material Size—5-inch Diameter
- 4. Cut Surface Area—19.635 sq. in.
- 5. Cut-off Length—3 inches.
- 6. Machine Type—C-1216A
- 7. Machine Cost—\$38,711.00
- 8. Blade type—Penetrator
- 9. Blade Pitch—3.5-5
- 10. Blade Length—162 inches
- 11. Blade Width—1.25 inches
- 12. Blade Cost—\$40.86 [volume discount]
- 13. Band Speed—130
- 14. Cutting Rate—7.96 sq. in./Min.
- 15. Blade Life—7070 sq. in. [360 pieces]

- 16. Fluid Type—80
- 17. Fluid Mix—0:1
- 18. Fluid Cost—\$5.45/gallon
- 19. Operating Cost/hour—\$40.00

Would you like to change any line [Y/N]

#### Cutting Cost Analysis

Machine Cost	0.2618
Blade Cost	0.1135
Fluid Cost	0.0681
Operating Cost	<u>2.0006</u>
Total Cutting Cost/Piece	<u>2.4440</u>

Producing 19.99 Pieces/Hour

- 1. Return to Sawing Analysis
- 2. Utilities
- 3. Hard Copy
- 4. Restart
- 5. End Session

Enter the number of your choice

Note, after completing entries for the production power sawing analysis, the engineer has the option to change any item, since programs are designed to permit changes by pushing (Y/N) and calling for approval or change of each item. Thus any cut-off machine, its cost and other items and costs can be entered, and the computer will adjust for that. But also note that all of this means nothing if you fail to check and verify material hardness for the material at hand, since occasionally hardness will deviate sharply from the nominal hardness for that material.

The cutting tools used for most cut-off and also for contour band sawing today are of high-speed materials that allow the band machine to assume some of the work traditionally performed on other machine tools (Figure 2.3).

## Operating the Cut-off Band Saw

All cut-off band machines operate in much the same way, whether they are small, manually operated models used in a tool room to cut stock off to length or big, powerful, fully automated high-production machines used for volume parts production.

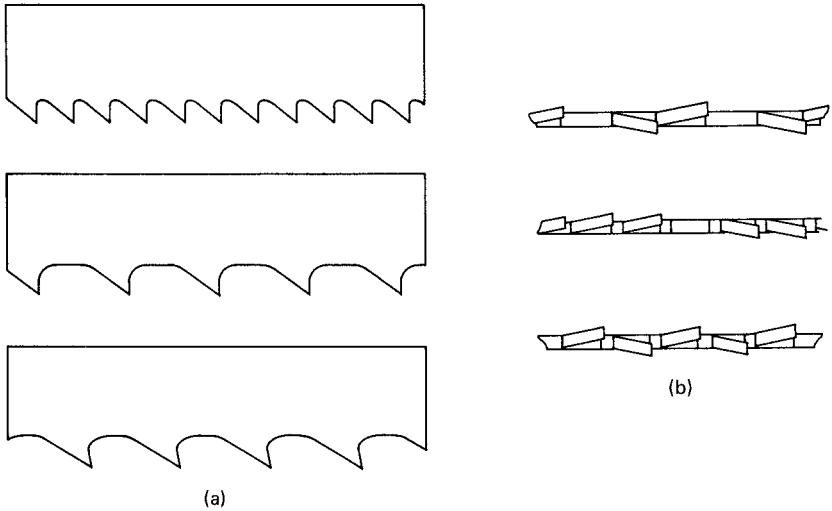


Figure 2.3. Band saw blades come in a great variety of widths, pitches (number of teeth per in.), and tooth forms, permitting selection of blade most suitable for a particular cutting operation. (Courtesy of DoAll.)

The following is the procedure for the operation of a cut-off band machine:

1. Select the band tool. The number of teeth to the inch required is determined by the thickness of the work to be parted.
2. For thinner materials, the rule of thumb is two to three teeth in the work at the same time.
3. Select the proper band speed for the material to be sawed.
4. Clamp the material in the vise. If it is a new bar of steel, always take a crop cut of about  $\frac{1}{4}$  in. The length of the crop cut is determined by the condition of the end of the bar. If the bar has been cut before, then a crop cut is not necessary.
5. Set the length of cut to be made. If only one piece will be cut, then measurement with a scale is satisfactory. If more than one piece is required and the saw is a manual type, a positive stop can be set to position the material to the proper length for each cut. If the machine is a production model, the index table can be set for the length of cut and the number of pieces set on the cut counter. All that's left to do is set the controls for automatic operation, turn the machine on, and saw until the counter turns the machine off when the job is finished.
6. When using high-speed saw bands, no more than  $\frac{1}{8}$  in. of stock needs to be left at each end of the piece for secondary operations, and even less is better. Otherwise, material is just being wasted. Sawing recommendation tables will give approximate band life in square inches per foot of blade, and multiplying that by the length of the band required for the saw being used gives total average blade life. Dividing the average number of

<b>CUTOFF BAND SPEEDS</b>		
<b>MATERIAL</b>	<b>AISI OR SAE</b>	<b>BAND SPEED SFM</b>
Carbon Steel	1008-1035	300
	1040-1064	275
	1065-1095	200
Free Machining Steel	1108-1132	325
	1212-1224	345
Manganese Steel	1108-1132	325
Nickel Steel	2317	250
	2330-2517	180
Nickel-Chrome	3115-3130	250
	3135-3150	180
Molybdenum Steels	4017-4042	260
	4047-4068	200
Chrome-Moly Steels	4130-4140	240
	4142-4150	205
Nickel-Chrome	4217-4340	200
Nickel-Moly	4608-4621	240
Chrome-Steels	5045-5160	225
Chrome-Vanadium	6117-6152	200
Nickel-Chrome Moly	8615-8750	200
Silicon-Steels	9255-9262	180
High Speed Steels	M1, T1, T2, T4, T5, T6	140
	M2, M10	
Die Steels	D2, D3, D7	90
Stainless Steels	201, 202, 321, 347,	110
	440, 303, 308, 310,	100

square inches cut per hour into average expected blade life indicates when the tool normally should be removed and discarded or resharpened.

7. Use the proper cutting fluid.

## Coolant/Cutting Fluids for Sawing

Coolant/cutting fluid both lubricates the blade/work-piece interface and prevents heat build-up. Properly selected, maintained, and used, it almost always sharply improves cut-off performance, so the selection of appropriate cutting fluids for sawing is critical to efficient, cost-effective operation. Cut-off is often viewed as a simple, straightforward, and “primitive” step that requires no more than clamping the slug and activating the saw. Yet contrary to this common misconception and the problems it generates, many materials require the proper use and care of coolant. Among the problems that ignoring proper usage entails are:

1. **Cleanliness** is the first problem. The size and the quantity of chips that typically go into the coolant tank is a crucial consideration particularly when doing production cutting. If the coolant tank is not cleaned periodically, the cutting fluids become contaminated, increasingly ineffective, and typically emit a noticeable odor.
2. **Maintenance** of the machine is important. If the brush that removes the chips from the teeth of the blade is not properly maintained, the teeth can become loaded and blade life sharply reduced.
3. **The cutting fluid** must be directed at the point at which the saw-blade teeth enter the work-piece, both to lubricate the teeth and to keep them cool for optimal sawing results. Lubrication helps the blade brush keep the teeth clean and prevents the chips from welding to the teeth.

The general rule of thumb for using the correct cutting fluid is:

- \* For speeds under 250 sfm, a sulfur based oil is recommended.
- \* For speeds up to 350 sfm, a soluble oil is recommended.
- \* For speeds over 350 sfm, a good synthetic cutting fluid is recommended.

These are general guidelines, not iron-clad rules. When, for example, you are running a series of batches of one or two pieces in different materials, it is not practical to change cutting fluid every time a change in the material must be made. For this type of operation, the most efficient course to follow is to select the fluid appropriate for the material most often being sawed.

Oil skimmers should be used to remove any surface contamination. The coolant tank should be cleaned on an appropriate regular schedule. Skimmers are generally used for removing oil that reaches the surface (floating) from water and water extended fluids. They will remove tramp oil and reduce smoke, mist and oil on parts. Choices include vertical, horizontal disk, and belt skimmers. Some are designed for hard-to-reach areas.

There are fluid testers for measuring cutting fluids, water soluble lubricants, and water soluble fluids. Other fluid testers are for measuring the concentrations of synthetic cutting and grinding fluids.

## Numerical Control and Computerized Cut-off

New model cut-off band machines typically feature numerical control and computerized monitoring and calculations. Even when the machine is manually operated or mechanically automated, computer calculations can help ensure proper settings. Consequently, the computer can be a bit help by reducing the time and effort necessary to achieve each of the objectives entailed in producing blanks. In addition, numerical control can reduce handling time, thereby reducing the cost of producing the part.

Chapter 9 has an example of calculating cost-per-cut for a typical cut-off job [See pages 289–290]. This use of a computer helps ensure efficient operation and provides reasonably accurate calculations of cost-per-cut.

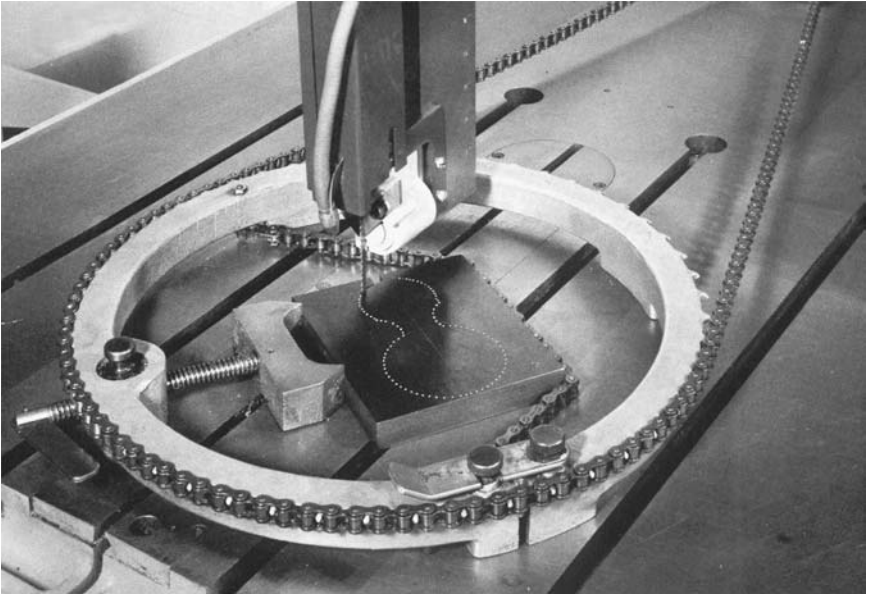
The example show the cost of cutting 5-inch round of carbon steel [1015-10-35, 160-175 Brinell hardness] using a DoAll C1216A band machine and the appropriate coolant, blade, machine feed and speed settings, etc.

Note that in the Chapter 9 example, when the calculations are completed, the operator or engineer has the option of changing any item [Y/N]. This means he can use the program to calculate the parameters for this material using any other cut-off machine, including cost per cut, and the computer will adjust for the changed inputs.

## Contour Sawing

Contour sawing with a band saw is easy. A layout line is imposed on the part to be sawed after the part is finished on the top and bottom so the piece will both lie flat on the saw table and permit the layout line to clearly show (Figure 2.4).





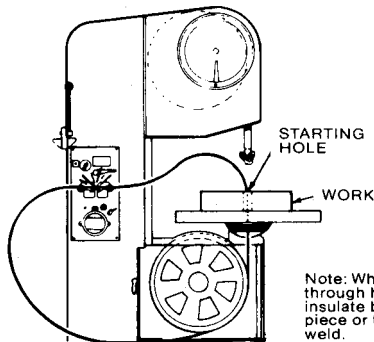
*Figure 2.4. Workpiece on contour band saw with layout line. Layout line drawn on the inked surface of the workpiece is prickpunched so that the line can still be seen and followed if the line is washed away by cutting fluid.*

If the section to be sawed requires an internal cut, then a hole is first drilled in the section to be removed to allow the saw blade to be inserted. This means that if the saw blade is already in band form, it must be cut, inserted through the drilled hole, and then welded together to form an endless blade. The contour saw is provided with such a welder for this purpose (Figure 2.5).

## Using the Contour Saw

The following is the procedure for contour sawing:

1. Select the material for contour sawing, making sure it is finished smooth on the top and bottom. Ground flat stock is available for this purpose and can be acquired from any vendor.
2. Lay out the shape to be cut. Prickpunch the outline so if the layout ink is removed while machining, the punch marks will remain to ensure a clear view of the layout line and permit accurate continued sawing.
3. The width of the saw blade used is determined by the smallest radius to be cut. For example, if a  $\frac{5}{8}$  in. radius is the smallest one on the layout, look at the chart on the machine. It indicates that the widest saw width that can cut a  $\frac{5}{8}$  in. radius without binding in the kerf is  $\frac{1}{4}$  in. This means the blade to use will be  $\frac{1}{4}$  in. wide.
4. If the cut is an internal one, drill a hole in the section to be removed large enough so the blade can be inserted and manipulated to the position wanted.



Note: When welding band which passes through hole in workpiece, be sure to insulate band from contact with workpiece or table, this will insure a better weld.

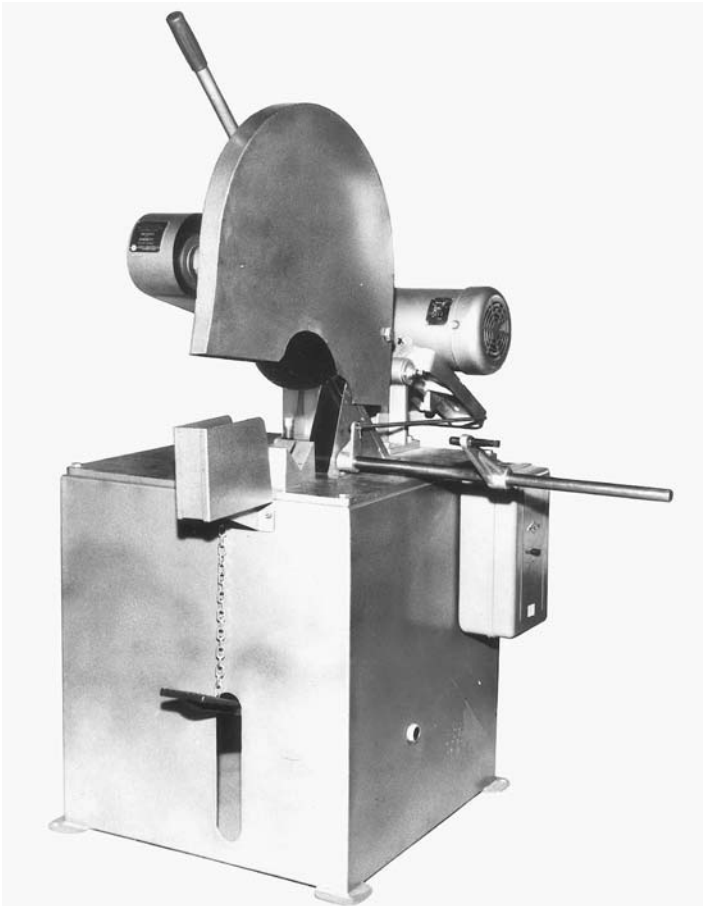


Figure 2.5. Contour saw. Most contour band saws have a welder that permits making internal cuts by drilling a hole in a segment of the portion of stock to be removed, inserting the ends of the blade through the hole, and welding them into a band.

5. Following the rule of thumb cited above, select a saw blade width and pitch suitable for the radius and thickness of the material. Insert the blade through the hole, then weld and anneal it (see “Welding Band Saw Blade,” later in this chapter).
6. Use the correct band speed and proceed to saw following the layout lines. When complete, cut the blade and remove the part.

## Circular Sawing

Circular sawing—sometimes called “cold sawing”—is a long-established and still very popular method of parting stock (Figure 2.6). There are two areas where it is used extensively. The first is for cut-off sawing of such nonferrous materials as aluminum and brass, and the other is for sawing both large-diameter rounds and thick, flat stock in sections called plates. Cut-off of aluminum extrusions is fast, accurate, and practically burr-free.



*Figure 2.6. Abrasive cut-off saw.*

There are two excellent reasons for the popularity of circular sawing. The first is that it produces a fine finish on the part (equivalent to a milled finish). The second is the high cutting speeds obtainable; it is a very fast method of removing material.

There are, however, some disadvantages. The initial investment for all but small circular saws is quite high. Because the blade must be relatively thick to efficiently cut larger-sized work, a considerable amount of material is lost in the form of chips, which is an important consideration when large expensive material is being cut. Moreover, since it must be thick to cut larger-sized materials, the circular blade is expensive and requires great skill to resharpen.

## Operating a Circular Saw

The procedure for operating a circular saw is as follows:

1. Before sawing, make sure the following conditions are met:
  - a. The saw is the proper machine for the work that needs to be done.
  - b. The correct saw blade has been selected for the material to be cut.
  - c. The blade is ground correctly.
  - d. The right speed and feed for the material has been determined.
  - e. The correct cutting fluid is being used.
2. Mount the saw blade on the spindle, making sure the cutting edges are rotating in the right direction.
3. Secure the blade firmly and inspect the machine, making sure all safety guards are in place.
4. Select the correct speed for the material to be cut. Some of the machines require a gear change, whereas others have a variable speed change.
5. Clamp the work securely in the vise.
6. If one part is to be cut, then measure the length before clamping in the vise. Stops for short-length cutting are built into just about all circular saws. If more than one piece is to be cut, then set the automatic feed, if there is one, and start cutting.

## Abrasive Sawing

The abrasive cut-off saw (Figure 2.7) resembles a cold saw, but its method of removing metal to part stock is similar to a grinding operation. At the correct speeds, with the correct support on the wheel, this type of cut-off machine can part virtually any kind of material. The resultant finish is excellent; within the limits of the machine, reasonably good accuracy can be maintained.

Speeds of the wheel at its periphery can range up to 15,000 surface feet per minute; at that speed, dry cutting is very efficient. The material gets quite soft and is removed by the grits in the wheel. Dry cutting is faster than wet cutting.

Wet cutting is done at a slower speed, 8000 surface feet per minute, and tends to cut more accurately than dry cutting.

Both methods remove metal at high rates. These machines do, however, have some definite disadvantages: The wheels wear rapidly and as their diameters decrease, the speed at the point of cut is progressively reduced and cutting efficiency



*Figure 2.7. Abrasive cut-off saw, which is fast and can part virtually any kind of material. (Courtesy of DoAll.)*

drops radically. Wheel costs and down time for frequent wheel changes result in higher costs.

A major advantage of the abrasive cut-off is its ability to cut hardened material that is difficult or impossible to cut by conventional methods. This, for example, means that some cutting tools from other machines such as end mills, and drills that have been damaged, can be retrieved by cutting off the damaged section and regrinding them to a useful shape.

Another valuable application is cutting small bars with great speed. On a production job, cut-off speed may well warrant the cost of the wheels if stock is automatically fed. For example, a 1-in.-diameter bar can be cut off in 2 sec, so a very fast automatic feed must be provided if stock indexing is to keep up with the cut-off speed.

## **Operating the Abrasive Cut-off Saw**

The following is the procedure for operating an abrasive cut-off saw:

1. Selection of the correct abrasive wheel is important. The following factors must be considered.
  - a. Accuracy required
  - b. Finish
  - c. Material to be cut
  - d. Size of material

2. To efficiently cut the material, be sure the correct machine is used. The following shows the relationship of horsepower to material and blade.
  - a. For a maximum of  $1\frac{1}{2}$ -in.-diameter material a 12-in. wheel is recommended with 2-horsepower motor.
  - b. For a maximum of 2-in.-diameter material a 16-in. wheel is recommended with a  $7\frac{1}{2}$ -horsepower motor.
  - c. For a maximum of 6-in.-diameter material a 20-in. wheel is recommended with a 15-horsepower motor.
3. Use a rubber-bonded wheel with a wet-type machine when accuracy and finish is required.
4. Clamp the work firmly in the vise after first measuring the length to be cut. Production machines have automatic feeds.
5. If coolant is used, it is vital that the mixture be correct. Cooling is the important function; a very thin mixture is recommended for this purpose. If the mixture is too rich, excessive heating may result, which tends to soften the wheel.

## Hacksawing

The power hacksaw is one of the oldest methods of cut-off used in the shop or plant (Figure 2.8). In its heyday, the power hacksaw was successful because the investment required was relatively low, it could cut materials accurately, and it could saw large blocks of steel as efficiently as it did smaller bars.

The acquisition cost of blade for the hacksaw is low and replacement is simple. The cutting of tough materials such as stainless steel is performed very efficiently on the hacksaw. Many of these machines are still in use today throughout the world.

There are some disadvantages to hacksawing: (1) It is perhaps the slowest of the various methods of sawing. (2) The low acquisition cost of the saw blade is misleading, because only two-thirds of the tool is being used while sawing. This represents an inefficient use of a cutting tool. (3) It cuts only on one stroke, losing cutting time while returning on its backstroke. (4) The structure of the machine does not allow for proper cutting speeds in many metals, thereby cutting inefficiently compared to other cut-off machines.

Selection of the appropriate hacksaw depends on the material(s) to be cut, production required, the accuracy needed, and dependability. There are hydraulic and mechanical types of hacksaws along with various designs such as open sided, closed frames, or U frames. The various types of machines will determine the rigidity for the type of application required.

## Operating a Power Hacksaw

The following is the procedure used when operating a power hacksaw:

1. Select the proper hacksaw blade according to the size of machine being used and the number of teeth required for the size or diameter of the piece to be sawed.
2. Mount the blade on the machine and apply the proper tension according to the manufacturer's recommendations.



*Figure 2.8. Power hacksaw. Simple and rugged, the power hacksaw has been a standby for cut-off for more than a century. Economical and accurate, it is well suited to low-volume cut-off but is too slow for volume cutting work. (Courtesy of Armstrong-Blum Manufacturing Co.)*

3. Determine the cutting speed. The speeds are in strokes per minute. Usually they range from 50 to 150 strokes per minute. The maximum is 150.
4. Clamp the work in the vise as follows.
  - a. Thin strips should be laid flat.
  - b. Short ends should be cut singly.
  - c. Round stock should be held flat on the table and clamped securely with the vise. Tubing should be held the same way.
5. If cutting on a sharp corner, a fine-tooth blade should be used so teeth do not get stripped out as the blade enters the material being cut.
6. Just as in band sawing the number of teeth per inch is determined by the thickness of the material being cut.
7. Use a good-quality cutting fluid when sawing to get the optimum results.
8. If for any reason the saw blade must be changed because of failure, don't start the new blade in the old cut. Start a new cut.

Some of the many other methods of cut-off in use today including such non-traditional approaches as lasers, water jet, etc.

## BAND MACHINING

Band machining combines the versatility and ease of set-up and operation of the contour band saw with the power, rigidity, and repeatability of heavy-duty production machines. Band machining is used for every kind of work; the six most common applications are depicted in Figure 2.9.

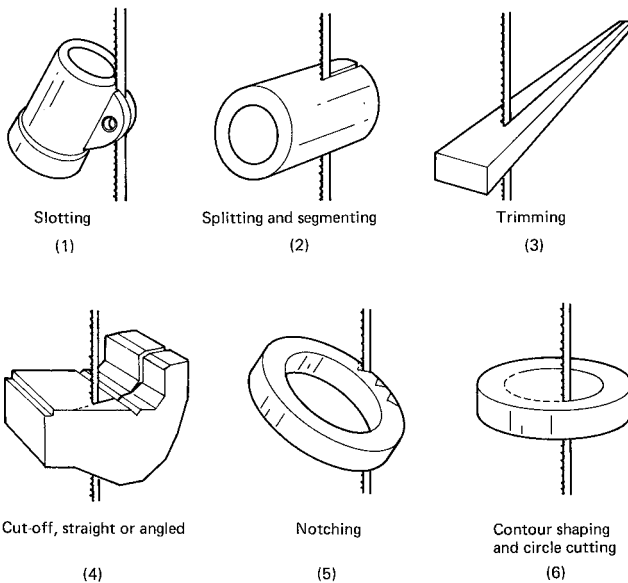


Figure 2.9. Line drawings depicting: (1) slotting; (2) splitting and segmenting; (3) trimming; (4) cut-off, straight or angled; (5) notching; (6) contour shaping and circle cutting. (Courtesy of DoAll.)

While its appearance reflects its contour band saw heritage and it retains that machine's great versatility, the "band machine" is set apart from its contour band saw predecessor by its ability to accurately and efficiently machine demanding parts in production volumes. This capability is largely due to its greater size, power, heavy-duty construction, powered tables, and the ability to utilize a wide range of high-speed steel, carbide, diamond, and other high-technology bandcutting tools, in addition to the ability to use a variety of simple fixtures.

### Solid Segments

A band saw blade removes only about one-sixteenth of an inch of material in the form of chips ("kerf") when it makes a cut; the remaining unwanted metal is removed in solid chunks. Thus the time required to perform machining is minimized



and the solid portions removed can often be used for other parts, both important considerations when performing machining work with thick cross-sections and major areas to be removed.

When the contour band saw is primarily used for die work, it often pays—given the low initial cost of carbon blade—to simply throw the band away once the die parts are completed.

Band machine applications typically are production jobs that require fast, efficient machining at metal removal and part output rates that make cutting rates, tool life, accuracy of cut, and repeatability important. Consequently, high-speed steel, carbide, and other premium tools must be used to ensure accuracy, repeatability for consistent product quality, and economical cutting and output rates.

As is true of every machining operation, the cutting tool is the determining factor in machine design. Consequently, the improvements in cutting tools (saw blades) contributed by new alloys and materials, as well as the advances in saw blade design that these foster, largely account for emergence of band machining as a modern machining method. These led to development of larger, heavier band machines that could take advantage of the newer tools. Hydraulics were added for greater tension of the band and for increased power for feeding the work into the tool. Recirculating cutting fluid systems were built into the machines to prevent excessive heat build-up, thus increasing blade life and permitting high cutting rates.

## Fixturing

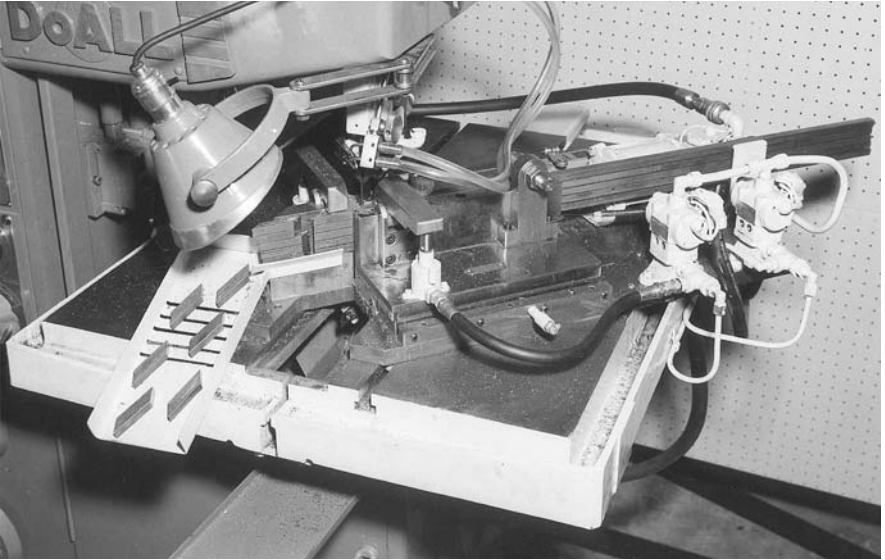
The cutting forces of the band machine cutting tool are exerted in a downward direction that pulls the workpiece down into the table. As a result, the band machine is extremely easy to fixture.

## Angle Cuts

When an angle must be cut, a simple fixture can be used to position the workpiece to the desired angle for the cut; as long as the piece is flat on the bottom, no clamping is necessary.

## Locating the Work

Simple locating stops on the side of the machine permit fast, accurate position of workpieces, and since the band machine can easily be automated to sustain high output rates, volume production of parts is readily achieved (Figure 2.10). With the downward cutting action in effect holding the part to the table, all that's needed is some way of preventing it from moving away from the blade. An arrangement as simple as positioning a bar to prevent the material from moving with the blade will do the trick. When the bar retaining the part is located perpendicular to, and square with, the blade, the part will be cut square. If the portion of the workpiece that will rest on the table is not flat, then a simple top-mounted clamp such as a quick clamp will hold it securely in position for the cut.



*Figure 2.10. Band machine set up for fully automated production run, with automatic stock feed and clamping for the cut, sawing, and discharge of completed parts. (Courtesy of DoAll.)*

## The Squaring Bar

A work-squaring bar is used on a powered table band machine to square work to the blade and to hold it in position for straight line cutting (Figure 2.11). The squaring bar eliminates the need to scribe layout lines, so work can be done directly and accurately to the required dimension.

## Machining Accuracy

The contour band machine can maintain a machining accuracy of  $\pm 0.002$  in. whereas, depending upon the blade used, the finish will be in the  $400 \mu\text{-in.}$  range. This means that when secondary operations are required, the material that must be left to perform them can be minimal, whereas for jobs where high-precision dimensions and finishes are not needed, no further machining is necessary.

## Using a Work-Squaring Bar

To mount the work-squaring bar, the following procedure should be followed:

1. Set the socket head screws loosely in the T-nuts that project up from under the bar.
2. Place the T-nuts in the T-slots of the table and slide the backup bar forward on the table.
3. Position the backup bar at a distance from the front edge of the saw band equal to the length of cut wanted. Allow sufficient clearance for positioning the workpiece.



*Figure 2.11. The squaring bar (shown on the left) is used to accurately position work relative to the blade for precision sawing.*

4. Square the backup bar to the table by aligning one edge of the movable workstop with the edge of the T-slot or the edge of the filler plate on the work table. For greater accuracy, use a solid square with one edge carefully aligned to one of the T-slots and the other to the bottom of the work-squaring bar.
5. Set the zero mark on the calibrated bar in line with the saw band and tighten the socket-head screws to anchor the backup bar firmly to the table.
6. Shift the movable workstop to one side of the saw band and run the table forward until the front of the stop passes the saw band. Then move the workstop toward the saw band until the edge just touches the set teeth.
7. Close the locking lever to hold the workstop in that position.
8. Adjust the pointer on the workstop so it lines up with the zero mark on the scale.
9. Now set the workstop to the size of the part to be cut.
10. Machine the part.

## **Servo Feed Attachment**

When doing contour sawing on a band machine, a servo contour feed attachment is very helpful and, for multiple parts, is all but indispensable. This device, usually built into machines with hydraulic tables, features a servo valve that

prevents the feed rate and force from exceeding preset amounts. The valve counteracts both the increases and the decreases in feed force exerted on the band that occur as it passes through the work, especially when a workpiece is being manipulated to produce curves, radii, and contours. In addition to helping ensure efficient machining, it reduces the possibility of blade damage due to overfeeding.

## Handwheel Control

The work is guided along a layout line by means of a handwheel that positions the workplace in relation to the blade. As the workpiece is manipulated into the blade, the lower saw guide senses changes in the feeding force and automatically adjusts the table speed and direction to maintain a nominally constant feeding force. When heavy workpieces are to be contour cut on a band machine, the recommended practice is to place the work on ball-transfer sections to minimize drag.

## Air-Powered Tables

Air-powered tables have a controlled amount of air pressure driving the table. When contouring lighter work using the air-driven tables, the table reacts much as the hydraulic table equipped with servo-feed mechanism does. When overfeeding occurs, air pressure reaches its limit as applied by the operator; this has the effect of adjusting the table speed automatically.

For smaller work the part is held by a round fixture and controlled by a handwheel through a chain driven by a series of sprockets. For larger parts the chain is connected either to pins located in the part or small V-shaped pins easily connected.

## Workholding Devices

Angle plates and machine speed clamps are used to facilitate holding parts of various shapes to the table.

## Disk-Cutting Attachment

A disk-cutting attachment is very handy when circles or arcs are required. When using the disk-cutting attachment, drill a hole or else make a center punch mark to position the centering pin and hold the work steady.

The following procedure is used to install a disk-cutting attachment.

1. Bolt it to the saw guide post with two sockethead cap screws and washers. Place the washers against the post to prevent damage to the post.
2. Lower the post until the upper saw guide is about  $\frac{3}{8}$  in. above the table.
3. Loosen the clamping bolts holding the clamp on the radius arm.
4. Keep the pivot arm parallel with the radius arm, then slide the unit along the radius arm until the centering pin is at a distance from the saw band equal to the radius of the circle required.
5. Hold the assembly at that point and tighten the bolt.

6. Square the center of the centering pin with the tip of the saw teeth. This can be done by placing one edge of a square on the saw slot while the edge of its perpendicular side is simultaneously touching the saw teeth. At this point loosen the clamping bolt and swing the centering pin until it touches the edge of the square, then retighten the bolt.
7. Move the saw guide post up or down to the thickness of the work and position the centering pin into the hole or center punch mark located in the center of the workpiece.
8. The part can be located in the center fixtures or pins can be mounted on it such that, with the chain wrapped around it, they will engage the links of the chain. Once located, the part can then be sawn to the circle or arc required.

If a number of circles or arcs are required, a powered rotary table can be used. In this case, the table is locked in position to prevent movement forward or backward, and cutting power is produced by the motion of the powered rotary table.

A wide variety of production jobs can be machined on a hydraulic contour saw using simple or quite sophisticated fixtures and devices. When designing a fixture or selecting an existing device, the major factors to be considered include: (1) the size, shape, and accessibility for holding the workpiece; (2) the clamping and durability of all moving parts; (3) the possibility of combining several operations into one fixture by interchanging stops, clamps, and nesting blocks; (4) the use of low-cost, basic, fixture building material—steel, epoxy, kirksite, wood, aluminum, or plastic. (5) Nearly all band machine fixtures consist of a base plate (for mounting the workpiece and clamps), positioned and held to the worktable by T-bolts. (6) Clamps are usually unnecessary because of the downward cutting action of the blade. When stack-cutting, however, clamps should be used to keep parts in alignment and to avoid vibration.

With its simplicity of setup, the band machine often can complete jobs in less time than it takes to make the initial setup for other machining methods.

## TROUBLE SHOOTING BAND PROBLEMS

Saw band problems usually result from improper adjustment, incorrect feeds or speeds or incorrect choice of blade. These general guidelines or adjustment considerations apply to all types of band machines, including cut-off band saws, contour band machines, band plate saws, etc., and also apply in general to such other types of sawing machines as hacksaws and cold saws.

To correct a given sawing problem, the following procedures can help isolate the problem and ameliorate or eliminate it:

1. Band vibration
  - a. Decrease or increase band velocity
  - b. Increase Feed Pressure
  - c. Check band guide inserts for adjustment
  - d. Check band guide inserts and backup bearings for wear.
  - e. Hold works firmly.
  - f. Check cutting fluid (it must be equally applied to each side of the blade)
  - g. Use finer pitch blade.

2. Premature dulling of teeth
  - a. Failure to break in band properly.
  - b. Band speed too high, causing vibration.
  - c. Too light a feed pressure (causes work hardening of material)
  - d. Cutting fluid is not covering the band at cutting edge.
  - e. Check to see if you are using the correct cutting fluid, mixture, and cleanliness.
3. Band teeth stripping.
  - a. Readjust brush, replace if worn.
  - b. Use finer pitch blade on thin work section
  - c. Reduce feed pressure.
  - d. Increase band speed slightly
  - e. Chip-welding, check coolant application.
4. Crooked cuts
  - a. Decrease feeding force
  - b. Increase velocity
  - c. Dull Band; replace
  - d. Coolant application; make sure it is not feeding only on one side of the blade.
  - e. Check band tension.
  - f. Check guide arms; should be as close to the work as possible.
  - g. Check inserts for wear.
5. Cutting rate too slow.
  - a. Increase feed pressure.
  - b. Check pitch; a coarser pitch blade may be needed.
  - c. If the proper pitch has already been selected, increase the band velocity.

## TROUBLE SHOOTING MACHINE PROBLEMS

The following is a simplified description of the proper sequence of automatic operation.

Horizontal Cut-Off Machines,

1. Machine is set up on “auto-cycle” and running.
2. Work is in position for the first cut.
3. Head lowers and completes cut.
4. Vise transfer (fixed opens, Index clamps).
5. Head raises.
6. Wand clears work, head stops at “up” position.
7. Index vise moves work forward.
8. Vise transfer (fixed vise clamps, index vise opens).
9. Head lowers repeating the cycle and making cuts until
  - a. Out of stock feature stops machine.
  - b. Preset cut counter stops machine
  - c. Operator stops machine.

### Machine Starts But Will Not Continue to Run

1. Check band tension and limit switch
2. Check preset counter. The machine will not run if it shows zero.
3. Is the index vise clamped?
4. Is it out of stock?
5. Check head down limit switch.

### Machine Runs But Band Will Not Turn:

1. Check band tension.
2. Check drive belts.

The above are a few of the problems that may occur that the operator can check by himself. Most problems are related to the operation description listed above and can be checked out in the order given. If these steps do not solve the problem, then the only recourse is to call in a service man.

## WELDING A SAW BAND

### Welding Band Saw Blades

Band saw blades are welded on machines that are designed specifically for that purpose (Figure 2.12), and it is common practice to weld a band saw blade on the welder mounted on most contour band machines to permit making internal cuts. Whereas most tool rooms and machine shops weld all or nearly all of their saw band on the machine, the main purpose of the welder is to allow the operator to saw internal sections. To make an internal cut in a workpiece, blades already in band form must be sheared, threaded through a hold drilled in the section to be removed, and then welded together to form an endless band.

### Blade Width

Contour band saws and band machines use a variety of blade widths, with the width of the blade for a specific job being determined by the size of the radius of any contours to be machined. The rule of thumb is to select the widest band that can cut the smallest radius required (when no contours are required, use the widest blade available). If, for example, the smallest radius in the layout is  $\frac{5}{8}$  in., the widest blade capable of turning that radius is  $\frac{1}{4}$  in. Most contour sawing machines have a chart right on the machine that shows the relationship between the width of a blade and the smallest radius it can turn.

### Welding Procedure

To weld a blade, the sequence to follow is:

1. Select the saw blade to be used. New blade comes rolled in a container and can easily be removed in a strand. **Caution: Wear safety glasses and protective gloves.**

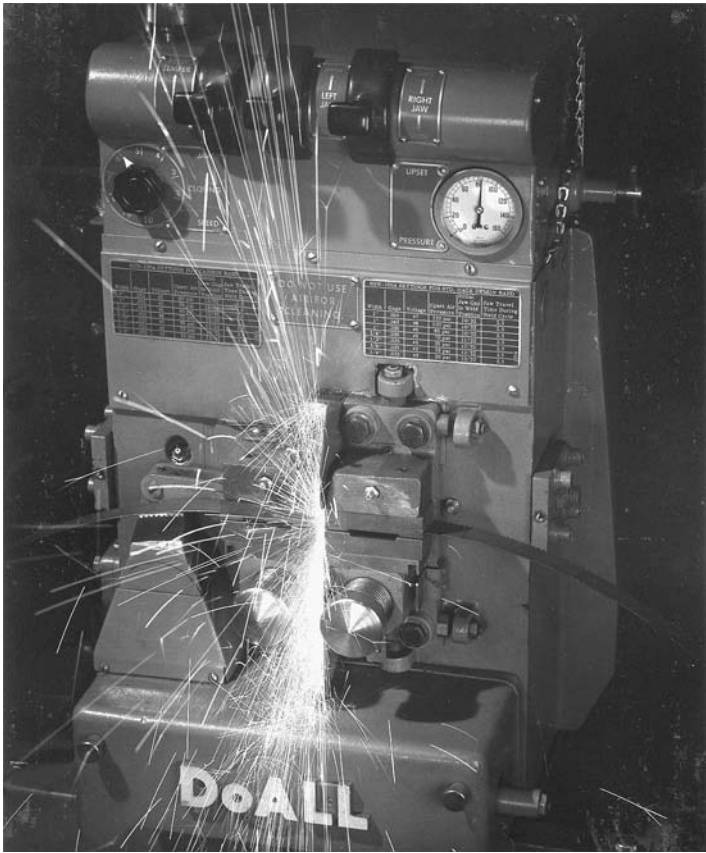


Figure 2.12. Welding band saw blade. (Courtesy of DoAll.)

2. Pull out the section of band to be used to the correct length. Cut it off.
3. Wipe the oil off the ends of the blade so they are clean, then reverse the ends so that the teeth are opposite each other and line the ends up, one on top of the other.
4. There is a grinder on the welding machine. Turn it on and grind the ends of the blade while they are being held together. Grind the ends slightly. They will now butt against each other with 100% contact.
5. Adjust the knob on the welder for the width of blade to be welded. On one type of welder, this knob indicates the range of width of blade. This width ranges in scale from  $\frac{1}{4}$  in. to 1 in. It provides a spring tension to exert force on the jaw as it moves forward when the welding lever is depressed. The lever is set for the width of the blade, to determine the travel distance of the jaw. Older machines only have a knob for setting spring tension.
6. After making the settings, place the ends of the saw blade in each jaw with the ground ends abutting each other in the center of the opening. Before clamping the jaws, line up the backs of the saw band along the straight edge on the jaws.



7. Press down on the upper lever to actuate the welding process. Hold the lever down until the cycle is complete. Release the jaws and raise the lever to its upmost position.
8. After welding, the weld section is very brittle and has a section on the weld, called the flash, that must be ground off to allow the blade to pass through the guides of the machine. Before grinding this section to size, it must be annealed to bring the blade back to the proper temper so it can flex around the band saw wheel.
9. Set the welding lever in the annealing position indicated on the machine. Place the welded band in the jaws and tighten securely. An annealing setting is on the machine for blade widths under  $\frac{1}{2}$  in. and over  $\frac{1}{2}$  in. Set it to the width of the blade being annealed. Press down on the annealing button and hold it down until the color of the weld is a dull cherry red. This color is good for just about all blades to be welded, fostering weld strength and the ability to flex properly. Do not permit the weld to get any hotter than this; otherwise, weld strength deteriorates and the blade will not hold together well. After the right color is obtained, jog the annealing button to cool the band slowly.
10. Release the jaws and remove the blade. Grind the flash off on each side, being careful not to let the teeth touch the grinding wheel. Use the thickness gage on the machine to check for proper thickness of the blade after grinding. Be careful not to apply too much force while grinding; allow the grinding wheel to do the work.
11. After grinding to size, put the saw band back into the jaws and temper the blade once more. This time, hold down the button only until the color of the weld is blue. This step relieves the stresses that occur during grinding.

## Checking the Weld

Check the weld:

1. The band back should be straight.
2. The weld area should be clean and free of inclusions.
3. The weld should be the same thickness as the band itself.
4. The teeth should have proper spacing and offset.

The weld area on the band is shown in Figure 2.13.

The band is now ready to be mounted on the machine.



Figure 2.13. Weld area of properly welded band saw blade. (Courtesy of DoAll.)

## Chapter 3

# TURNING AND THE LATHE: DEFINITION AND HISTORY

## INTRODUCTION

This chapter briefly discusses turning and defines the “basic lathe,” complete with a brief history, description, nomenclature, and typical operations, including the use of accessories. Cross-references to broad or general topics are listed after the Index to Chapter Segments whereas cross-references to *Machinery's Handbook* (hereafter also referred to as *MH*) for subjects related to a specific step or operations are cited where that step or operation is presented.

## Definition of Turning

*Turning* is the process of rotating or turning a workpiece against a cutting tool to impart a new shape, hence the name. Machines performing this operation may bear different names (engine lathe, chucker, automatic screw machine, turret lathe, turning centers, etc.), but all are *lathes*.

The lathe can face, turn, chamfer, neck, taper, drill, bore, ream, spin, file, polish, hone, buff, knurl, cut internal and external threads, and cut blanks (cut-off), making it one of the most versatile machine tools in use today.

## History of Turning

Turning is one of the earliest techniques developed by man for imparting new shapes to materials: Illustrations made by Egyptian artists more than 5000 years ago depict lathes in clearly recognizable and relatively advanced form. Consequently, the lathe is likely to be much older than that. The lathe was the second of man's “complex tools” to be invented (the bow-drill was the first), and it

still ranks among the most important basic methods for giving various alloys and materials new shapes.

## The Basic Metalworking Lathe

The basic metalworking lathe was one of the first of the “true” machine tools. It was introduced early in the Industrial Revolution. Historians credit the *screw-cutting lathe*, developed by Henry Maudslay around 1800 to produce threaded metal parts, with being the first lathe to possess all of the basic features of the modern machine tool (Figure 3.1). With its ability to produce duplicate screws of a variety of varied pitches from a master leadscrew through the use of change gears, as well as to turn cylindrical parts, the lathe played a major role in the Industrial Revolution that created modern industry. Often the only machine tool on hand in the largely handcrafting metalworking shops of the early Industrial Revolution, the lathe continues to rank among the most useful machine tools found in modern toolrooms, machine shops, and repair and maintenance shops (Figure 3.2).

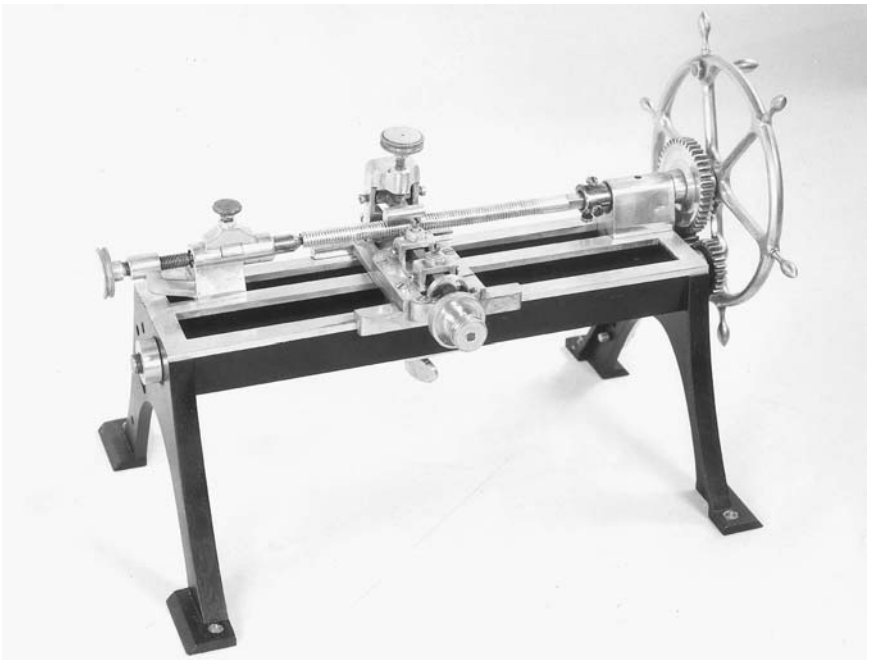


Figure 3.1. Maudslay screw-cutting lathe, the first of its kind. (Courtesy of DoAll.)

### Lathe Design

Specific lathe designs vary considerably, but the basic features of all versions are similar. Thus, basic lathe nomenclature applies to every variety of turning

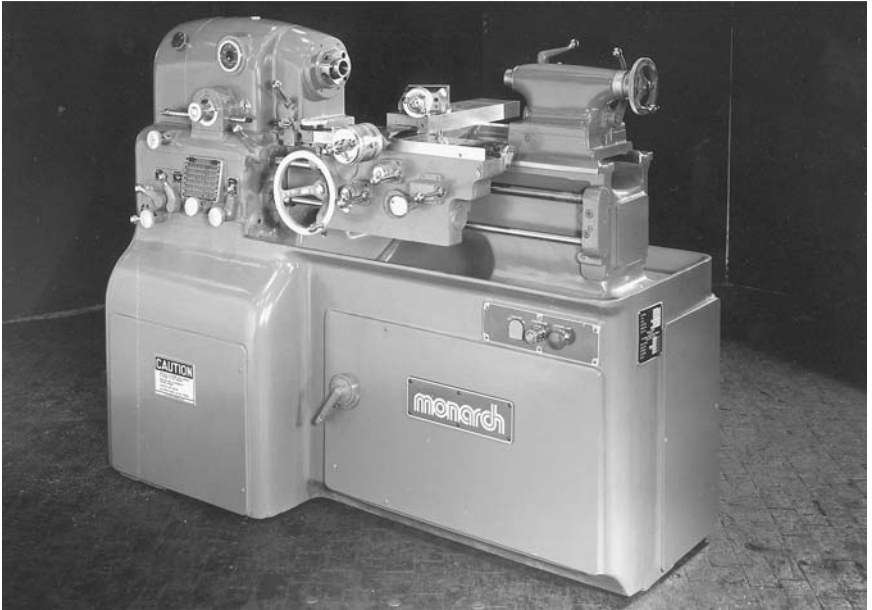


Figure 3.2. A typical toolroom lathe. (Courtesy of Monarch Machine Tool Co.)

machine. This characteristic of the “basic lathe” makes it possible to develop a general description of the typical sequence of steps—*what part* to move at *what time* to accomplish *what purpose*—required to produce a part on a lathe.

## Basic Lathe Designation and Nomenclature

Lathes bear such distinguishing designations as “toolroom lathe,” “light powered lathe,” “standard engine lathe,” “vertical turret lathe,” and “heavy-duty, large swing lathe.”

Nevertheless, they have the same basic elements and use the same nomenclature (Figure 3.3), varying only in accuracy, size, rotational speed range, horsepower, and the particular form that their specific design features and controls take. This is also true of variations on the basic lathe-design machines that do not have “lathe” in their designation, including “screw machines,” “chucking machines,” and “boring machines”—they are all basically lathes. Moreover, putting the method of driving and controlling the application of cutting tools to the workpiece aside, even highly sophisticated numerically controlled turning centers are lathes.

## Lathe Model Designations

By convention, lathe names or designations typically include a two-component model number that indicates the machine’s maximum workpiece capacity in terms of (1) *swing*, or the diameter of the largest workpiece it can rotate

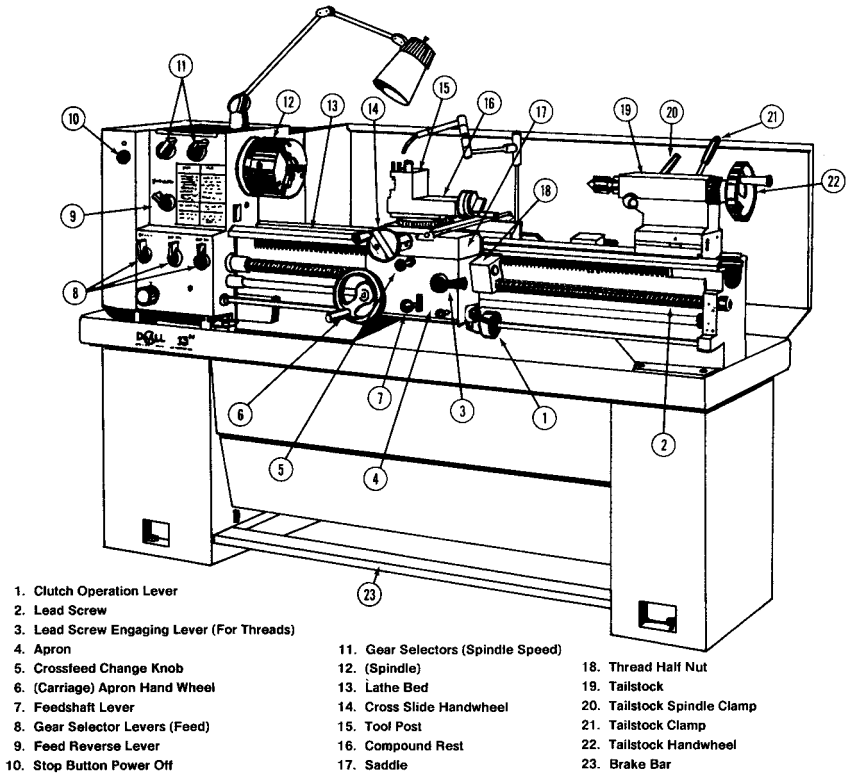


Figure 3.3. Lathe nomenclature. (Courtesy of DoAll.)

in a chuck or between centers without hitting the bed; and (2) *distance between centers*, or the dimension, normally in inches, or the longest workpiece that can be mounted and turned between centers on the machine. This is sometimes referred to as *length* or “*bed length*” For example, XYZ Co.’s Model “X-1340” lathe would have a *swing* of 13 in. and a *distance-between centers (bed length)* of 40 in. The “13” means the centerline of the spindle is located a little over  $6\frac{1}{2}$  in. from the bed, thus permitting a workpiece up to 13 in. in diameter to turn about that centerline without touching the bed. “Gap-bed” lathes—ones with a removable bed section that permits turning work larger in diameter than the particular machine’s nominal swing—provide a way to get around this basic dimensional restriction. They were commonly used in the heyday of conventional machining to handle an occasional need to turn a workpiece that was larger than those typically machined at the shop, but are rarely used today because of the excessive time required for changeover. The “40” means that when (1) the tailstock is moved as far as it will go toward the end of the bed opposite the spindle and (2) a part is supported between a center mounted to the spindle and one mounted to

the tailstock, the length of the workpiece, when measured between the centers, will be 40 in.

## Controls

The location of the operating *controls* and the form they take may vary, but the functions they perform—starting, stopping, braking, reversing rotational direction, setting RPM, etc.—are the same in all versions, which results from the fact that all “lathes” do basically the same thing.

## What the Lathe Does

A basic, unadorned lathe does just one thing: turn cylindrical parts. With attachments, however, it becomes very versatile, performing such additional functions as tapering, boring, tapping, and reaming.

## Horsepower

The type of alloys and typical volumes of metal to be removed determine the horsepower a lathe must have, and the horsepower largely determines its construction—the greater the horsepower required, the more rigid the machine must be to permit taking full advantage of the potential stock removal capability provided by that horsepower.

Accuracy also influences the rigidity a specific lathe must possess—the greater the accuracy required, the greater the rigidity needed.

## Size

The basic determinant of machine size is the size of the largest work to be turned. Standard lathes range from models bearing a “6–12” swing/length designation—6-in. swing (3-in. centerline to bed) and 12-in. bed length—up to ones boasting a “400–240” model number, or 400-in. swing and 240 in. (20-ft) bed length. Special lathes may be much larger—for example, those designed to turn housing rings for hydroelectric power generating turbines or propeller shafts for supertankers.

## Lathe Axes

Lathes are driven along the work by a feed shaft or screw, with the direction of movement being referred to as *longitudinal*. That direction is also known as the *Z* axis, since under standard conventions the direction in which a driven component—on the lathe, the cutting tool—moves toward or away from the spindle is designated as the *Z* axis. The feed shaft feeds the tool into the workpiece through three components: (1) the end gearbox, (2) the quick-change gearbox, and (3) the carriage and apron assembly.

The carriage and apron assembly simultaneously moves the cutting tool along the work in the *Z* axis and across the work in the direction perpendicular to the *Z* axis.

This cross-feed direction is known as the *X* axis, and it, too, is accomplished by means of the end gearbox, quick-change gearbox, and the carriage and apron assembly. In most operations, feed—the distance the cutting tool is advanced into or along the workpiece—is set so that cross-feed or travel in the *X* axis per spindle revolution is twice the longitudinal feed or amount of travel per revolution along the *Z* axis.

Levers on the quick-change gearbox permit selection of a wide range of feed rates. In addition, the quick-change gearbox makes it possible to select threads by changing the feed from the feed shaft to a lead screw, which furnishes the required lead for the thread to be turned. Both English and metric threads can be selected with ease on most modern lathes.

The carriage holds the compound and the compound rest. The compound can be fed across the work manually or under power. The compound rest holds the toolholder and can only be fed manually; it can be set at any desired angle. Machining accuracy's vary with the type of lathe, and the need for accuracy varies with the work to be done.

Most lathes used in a toolroom, for example, are used for primary work. Since stock must be left on the part to allow bringing the workpiece to finish dimensions in the secondary operations, the accuracy of the cross-feed screw and of the readouts are normally not critical.

In contrast, in typical job shop and other machining operations in which a substantial portion of the work includes finished parts, the machine accuracy—the accuracy of both the cross-feed screw and of the handwheel or, preferably, digital readouts—is critical, and the machine must be capable of holding the tightest finish tolerances routinely encountered.

## Setting Feeds and Speeds

Rates of feed are set in relation to rates of speed. The selection of settings for a specific combination of speed and feed on a gear-head lathe is normally accomplished quickly and easily by means of levers mounted on the headstock. To aid in making proper settings selections, the speed settings within each of the available speed ranges are displayed in numerals printed in the same color. To set the speed, the range containing the desired speed is selected, and the selection level is then set to that desired speed setting, with the color serving to reduce the chance of incorrectly making lever settings. Feeds are marked by letters and typically require a four- to five-letter setting to achieve the feed rate desired for all but threading operations. For threading, an additional lever that actuates the lead-screw is required. *MH* reference: "Turning speeds and feeds."

## Accessories

There is a broad range of accessories and attachments available for use on lathes. Technically, an "attachment" is affixed to the lathe at the factory or by a "field installation" procedure and, in effect, becomes part of the machine.

In contrast, an "accessory" simply is clamped to, or mounts onto, the machine without machining or other special steps and thus does not become part of the

Material	Depth of Cut	High Speed Steel Tool		Carbide Tool	
		Feed IPR	Speed SFM	Feed IPR	Speed SFM
1015–1035	.300	.020	300	.020	525
1040–1095	.300	.020	300	.020	475
1132–1151	.300	.020	300	.020	525
1335–1345	.300	.020	300	.020	400
2330–2345	.300	.020	300	.020	425
3115–3130	.150	.015	65	.015	350
4032–4042	.300	.020	80	.020	400
4130–4140	.300	.020	80	.020	400
4130–4141	.300	.020	65	.020	425
4337–4340	.300	.020	60	.020	400
4615–4621	.300	.020	80	.020	480
5130–5135	.300	.020	80	.020	400
50100–52100	.300	.020	70	.020	400
6145–6152	.300	.020	75	.020	400
8647–8660	.300	.020	75	.020	400
9255–9260	.300	.020	75	.020	400
T1-T2-T6	.300	.020	50	.020	325
M1-M2-M6-M10	.300	.020	50	.020	325
T15-A7-D3-5	.300	.015	30	.015	200
201–304	.300	.020	70	.020	300
303	.300	.020	80	.020	300
5056–7178	.300	.030	800	Aluminum Wrought	
A332–B850	.300	.030	800	Aluminum Cast	

The above speeds are just a few examples of appropriate speeds and feeds for a variety of materials. Of course, speeds and feeds will vary depending on the amount of material to be removed when finishing, taking into account the hardness of the material. The chart above assumes the hardness to be about 200 brinell.

### *Lathe Feeds and Speeds: Cutting Tool Material*

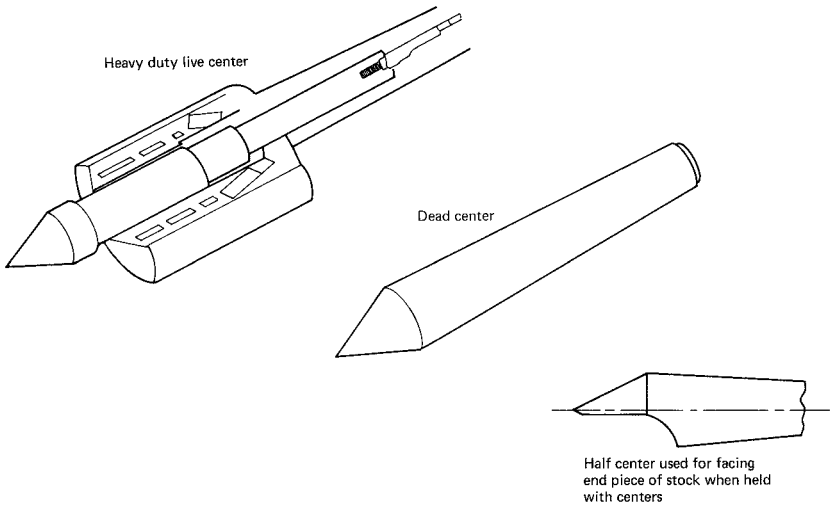
machine. The distinction between an attachment and an accessory normally makes little practical difference, so in normal shop usage, all are simply referred to as “accessories.”

## Typical Accessories

### Centers

Although centers are considered to be accessories, it is impossible to accomplish many routine operations without them. There are two basic types of centers: *live* and *dead* centers (Figure 3.4). When mounted in the tailstock, a dead center





*Figure 3.4. A sketch of a live, dead, and half-center used to support work extending out from the chuck.*

remains fixed while a part mounted onto it revolves around it. In contrast, a live center mounted in the tailstock turns with the work. If a dead center is mounted in the spindle, it turns with the work and consequently, in that arrangement, functions as a live center.

## Lathe Dog

When a part is mounted between centers, some method must be used to link the part directly to the spindle so that its rotation is transmitted to the part. Otherwise, when the cutting tool enters the work, instead of taking a chip, it will bite into the work and prevent it from rotating. The function of the lathe dog is to transmit the spindle rotation to the workpiece. The dog is mounted on the part and clamped securely to it, and then an extension or “leg” is set against a stud or other interlocking feature of a face plate mounted on the spindle. With the dog in place, as the spindle rotates, it drives the part, rotating it and allowing the cutting tool to remove metal from it.

## Other Accessories

These include a variety of choices of:

- \* Toolholders—quick change, rocker arm, open-sided, and turret block.
- \* Chucks—three-jaw (Figure 3.5), four-jaw (Figure 3.6), six-jaw, collet, and air-actuated.
- \* Face and drive plates.
- \* Taper attachments (Figure 3.7).



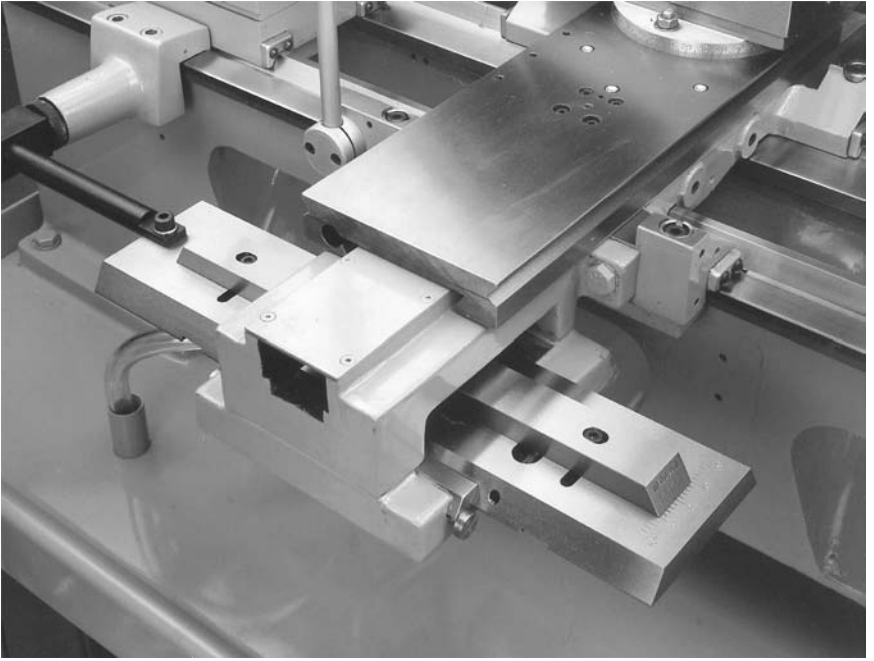
*Figure 3.5. A typical three-jaw (universal) chuck. (Courtesy of DoAll.)*



*Figure 3.6. A typical four-jaw (independent) chuck. (Courtesy of DoAll.)*

- \* Six-position turret.
- \* Rests—steady (Figure 3.8) and follower (Figure 3.9) rests.
- \* Mandrels and milling attachments.
- \* Digital readouts.

Each of these will be covered in specific discussion of operations where its use is critical or useful.



*Figure 3.7. Taper turning attachment. (Courtesy of DoAll.)*



*Figure 3.8. Steady rest, which is used to support heavy bars. (Courtesy of DoAll.)*



*Figure 3.9. Follower rest, which is used to support long thin shafts. Otherwise, the shaft would bend due to the pressure exerted by the tool. (Courtesy of DoAll.)*

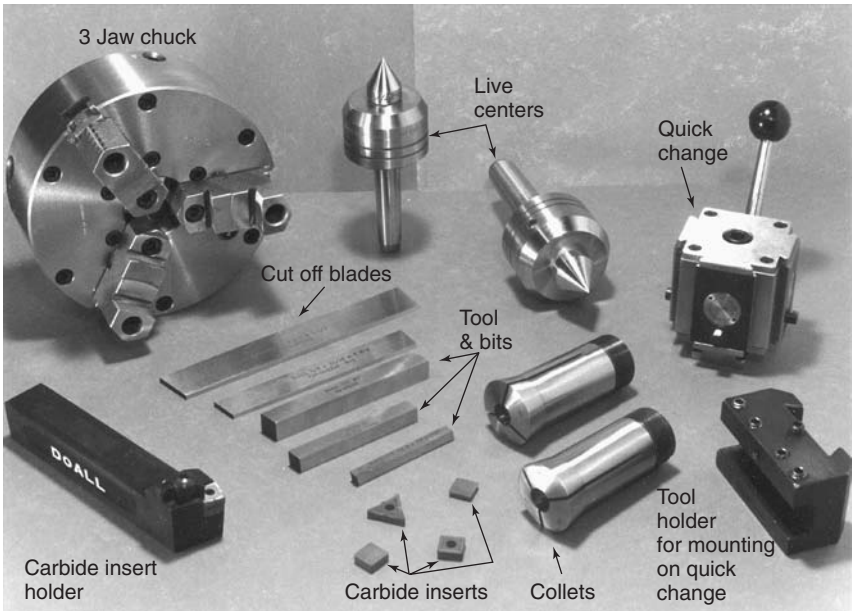
## FACING

The basic purpose of facing is to square the ends or faces of workpieces with the axis of the work as it turns. Thus in facing, the workpiece is held firmly in ways that permit it to run true about its axis or centerline. This is accomplished with the work either held in a chuck or mounted between centers.

### Facing in a Chuck

The steps required are as follows (see Figure 3.10 for an array of chucks, toolbits, toolholders, etc.).

1. Mount the work in a chuck. Use a three-jaw chuck for round work and a four-jaw chuck for irregular shapes. Work extends about 1 in. from chuck for rigidity.
2. Clamp work securely in the chuck.
3. Select a left-hand toolbit (when mounted, the cutting edge is closest to the *tailstock*; if we are dealing with a right-hand toolbit, cutting edge is closest to the headstock).
4. Place tool in toolholder—rocker-arm, quick-change, open-end, or variations of these.
5. Center the tool. This can be done by:
  - a. Centering by eye to the part, then adjusting to center after making the first cut.



*Figure 3.10. An array of chucks, toolbits, toolholders, centers, etc. Specifically shown are: a three-jaw chuck, live centers, quick-change toolholder, collets, carbide inserts and holder, cut-off tools, and high-speed toolbits.*

- b. Centering the tool to a dead center held in the tailstock.
- c. By centering tool. Readily made and kept for later use, a centering tool positions the toolbit in relation to a known location. The tool is positioned against that location, and the toolbit cutting edge is set to a fixed point on the centering tool.
6. Start the lathe. Lathe must turn counterclockwise (toward operator).
7. Establish tool/work contact. Bring the tool up close to the workpiece surface as it is turning and “kiss” or touch lightly. Work may be faced from the center out or from side to center; either one is acceptable practice.
8. Engage the automatic cross-feed and take a light cut to check for proper centering of the toolbit.
9. Once setup has been confirmed by light cut(s), remove from 0.050 to 0.125 in. of stock per pass until desired size is achieved.
10. File burrs from sharp edge.

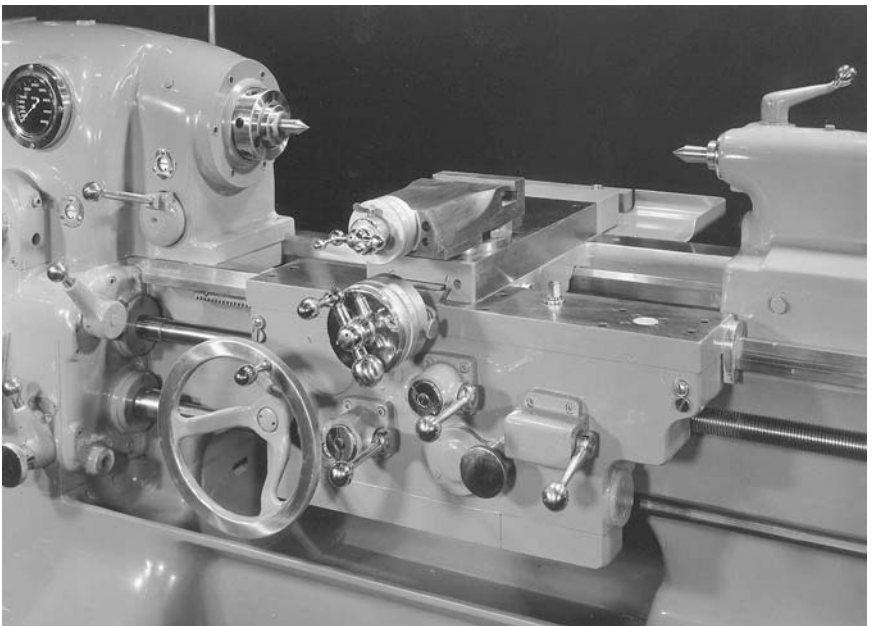
## Facing between Centers

Since for this operation the workpiece is secured at both ends, facing between centers ensures that the face will be perpendicular to the axis of the workpiece. It requires initial facing in a chuck, so it is used for high-precision work. This work

can be performed on either a “dead” center (tip of center does not rotate with the work) or a “live” center (tip of center rotates with the work).

The sequence of steps is as follows.

1. Center Drilling. Center holes to accommodate the centers must be drilled. To do this:
  - a. Mount workpiece in chuck for facing. The reason for facing in a chuck before facing between centers is to ensure that the face of the workpiece will be exactly perpendicular to the centerline of the body of the work.
  - b. Face (see “Facing in a Chuck”).
  - c. Mount drill chuck in tailstock, insert a center drill, and tighten securely.
  - d. Select correct speed for the size of the center drill.
  - e. Start lathe, drill to a depth one-half the taper length of the center drill.
  - f. Repeat sequence for the other end of the workpiece.
2. Remove center drill and drill chuck from the tailstock and the chuck from the spindle.
3. Put a dead center in the spindle (see Figure 3.11) and mount a drive plate on the spindle. Tighten drive plate securely (method—camlock, screw, or taper lock—depends on lathe’s spindle holder).
4. Mount a straight dog to the workpiece.



*Figure 3.11. Centers mounted in lathe, spindle and tailstock. (Courtesy of Monarch Machine Tool Co.)*

5. Put a live or dead center into the tailstock.
6. With the dog attached, mount the workpiece between the centers, adjusting the tailstock so that workpiece is held securely in place.
7. Adjust the dog so it engages the driving screw on the drive plate and tighten.
8. Mount a *right-hand* facing tool to face to the center on the tailstock. A half-round dead center can be used to allow extra clearance for the toolbit.
9. Touch the toolbit to the face.
10. Start the lathe and take a light cut.
11. Machine to desired size. If a dead center is being used, proper tension of the center against the workpiece is essential, and sufficient lubrication must be applied between the center and the workpiece.

## Turning Work on a Face Plate

The primary purpose of a face plate is to serve as a device to which a workpiece—often of irregular (noncylindrical) shape—can be mounted for machining. When a face plate is used, the work is clamped directly to it.

### Face Plate, Drive Plate?

The primary function of the face plate—to serve as a rotating base to which work is directly mounted—differentiates it from the drive plate with which it is often confused:

1. The drive plate is used primarily to turn a part mounted not on the plate itself but, rather, between centers. Most drive plates are designed so that a straight driving dog mounted on, and clamped to, the work will engage a protruding bolt mounted to the drive plate. Consequently, when the spindle revolves, turning the drive plate, the bolt engaging the dog spins it, turning the workpiece.

In some versions, a slot machined on the outer periphery permits mounting a bent dog (Figure 3.12) whose angled end engages the slot. The bent dog is typically used to drive larger work, whereas the straight dog is normally the choice for smaller workpieces.

2. Although the *face plate* can, if necessary, be used as a drive plate, it is not as convenient for such usage as a drive plate because of its size and weight. Consequently, except when a suitable drive plate is not available, the face plate normally is used only as a mounting base. Either bent or straight dogs can be used with a face plate, but in the main, the bent dog is normally the most appropriate one to use.

Because a face plate mounts directly to the spindle, if the spindle runs true and the plate is properly mounted, it will also run true. Consequently, use of a face plate ensures high accuracy, and thus it is particularly suitable when the periphery to be turned must be precisely perpendicular to an end or portion of an end surface already machined.



*Figure 3.12. Bent lathe dog, used when turning on centers.*

To permit proper clamping of the workpiece to the face plate, the part size must be appropriate for the size and weight of the face plate. The face plate also can be fixtured to handle light production work as a secondary operation.

The procedure for turning a workpiece on a face plate is as follows,

1. Mount the face plate to the spindle; it is mounted very much in the same way as chuck (see page 60). Make certain it is securely tightened to the lathe spindle.
2. Put bolts and clamps in each of the slots required, to achieve secure clamping of the work to the face plate. Bolts and clamps must be of a size appropriate to the work; use the shortest bolts that can be securely mounted, to minimize the possibility of “crashes” with the cutting tool, its holder, or the carriage of the machine.
3. Clean the face plate carefully, making sure no dirt, metal particles, etc., remain that might interfere with accurate alignment of the part.
4. Clamp the work to the face plate; locate the work as close as possible to the center point of the face plate.
5. Clamp an indicator to the machine or the carriage, and indicate the part, making sure it is in the position required for the job. If the periphery of the part to be machined is smooth, the indicator can be safely used. If, however, the periphery has a rough finish, then use the chalk method to avoid damage to the indicator.
6. In lining up the part in proper relationship to the face plate center, use a soft hammer to tap the work in the right direction until:
  - a. “Zero” is recorded on the indicator all around the workpiece periphery (The Indicator Method).
  - b. You get a complete, even-density chalk mark completely around the workpiece periphery (The Chalk Method).



7. Using the method for mounting a part to a four-jawed chuck, find the high point and then carefully and very slightly loosen the clamp on the *opposite side* (the low point). Now gently tap the work with the soft hammer in the direction of the low point. Tighten the clamp and indicate the part again. Continue the process until the part is properly aligned.
8. If the part is large and heavy—particularly if it is also irregularly shaped—it may be necessary to affix a counterweight to control or eliminate vibration caused by an out-of-balance weight distribution.
9. **Caution.** Before starting the machine, it is good practice to turn the work one or more complete revolutions by hand to make sure everything is properly secured and that the work, clamps, and driving dog clear the carriage, tool and toolholder, and machine ways.
10. **For safety,** when starting the machine, stand to one side of the revolving face plate to preclude the possibility of injury in the event that something lets loose.

The part can now be turned as in any other turning operation performed on the lathe.

## Steady and Follow Rests

Steady and follow rests are typically taken very lightly but, in reality, they represent two of the potentially most important accessories available for use on a lathe.

The *steady rest* (Figure 3.13) is primarily used for supporting long, heavy parts to be turned. When the steady rest is used, a center mounted in the tailstock assists it in keeping the workpiece properly aligned.

The term “heavy” is relative to the capacity of the machine: A part that is 4 in. in diameter and 10 in. long may be heavy for a 13-in. lathe, but light if machined on a 17-in. lathe. If the part were to be machined on the 13-in. lathe, it would be heavy enough to require a steady rest, whereas on the 17-in. lathe, it would be light enough that it could readily be machined cleanly and accurately without one.

The steady rest is indispensable when a long part must be center drilled at one or both ends or when long stock must be drilled and bored.

As the name implies, the steady rest, once mounted, does not move but remains at the point where it is initially positioned.

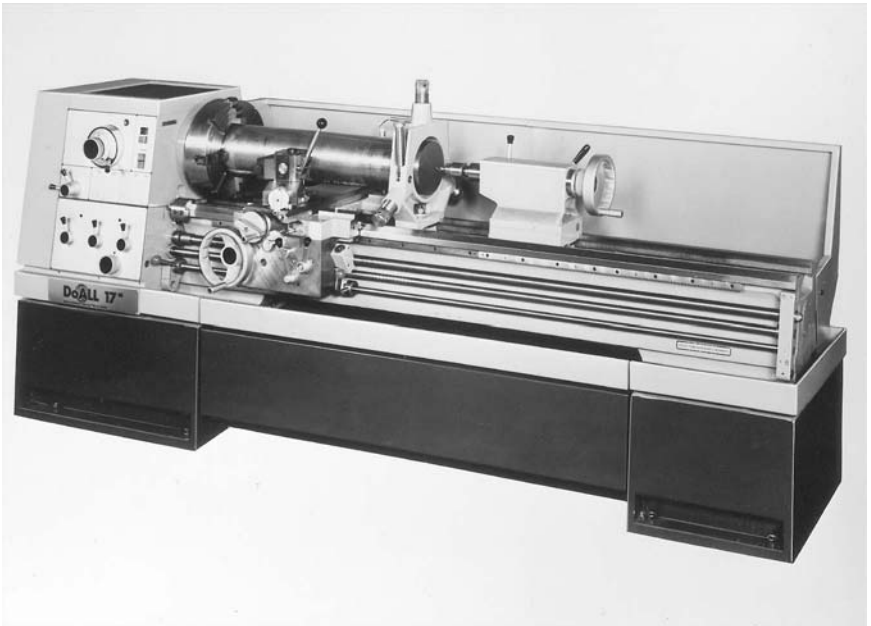
The *follow rest* is normally used to support work that has a tendency to deflect under the pressure of the tool being fed into it.

In contrast to the steady rest—and as its name indicates—the *follow rest* mounts to the carriage and thus moves along with or “follows” the tool as it makes its cut along the periphery of the work.

This tool is extremely helpful if not absolutely indispensable when long narrow shafts must be machined.

The procedure for using the steady rest is as follows.

1. Mount the steady rest on the bed of the lathe.
2. Mount the work in a chuck.



*Figure 3.13. An example of a steady rest supporting large work.  
(Courtesy of DoAll.)*

3. Center drill the end facing the tailstock. To do this, support the work with the steady rest, tighten the chuck, then adjust the jaws of the rest so the part turns true when the machine is running.
4. Put a drill chuck and center drill in the tailstock, select the proper turning speed, and drill the center hole. The center drill should enter the work to a point about half way up the tapered portion of the tool.
5. Remove the drill chuck with the center drill from the tailstock and replace it with a center. Adjust the tailstock and tighten it to the bed. Using the tailstock handwheel, feed the center into the center hole until it snugly and firmly supports the work.
6. Re-adjust the steady rest, positioning it so that it does not hinder the turning operation, yet provides secure support to the work.
7. In adjusting the steady rest jaws, make sure the work will turn easily without binding but still provide firm support, free of any “play.”

The procedure for using the follow rest is as follows:

1. Mount the follower rest to the carriage of the machine; mounting holes are located in the back of the saddle.
2. Adjust the jaws of the follower rest so that the work can turn freely, yet is held securely so that it cannot deflect.

Because the follower rest is designed to move with the tool and toolholder, providing constant support to the work in the immediate vicinity of the point

of cut, it is particularly helpful when turning long and narrow rods, shafts, and bars.

## Turning Round and Irregular Shapes

“Straight turning” on a lathe creates a new shape by removing material from the periphery of a round or irregularly shaped workpiece. Material may be removed at more than one level or diameter.

*Round* parts can be turned in a three- or six-jaw chuck. The three-jaw chuck is most commonly used; it is called a “universal chuck” because all three jaws move simultaneously.

*Irregularly* shaped parts can be turned in a four-jaw chuck. The four-jaw chuck is known as an “independent chuck” because each jaw adjusts independently, permitting the chuck to firmly hold parts with portions of their periphery at varying distances from the centerline of the part.

## Lathe Chucks

Workpieces mount to lathe spindles in one of three basic types of spindle mounts: (1) cam lock spindle nose; (2) tapered spindle nose; (3) threaded spindle nose. The threaded spindle nose mount is the least popular of the three because it is less accurate than the cam lock and tapered mounts.

## Chuck Cradle

It is standard practice to use a wood cradle (into which the chuck will fit) that can be placed on the lathe bed under the spindle when mounting and dismounting a chuck. This protects the operator and machine in the event a chuck slips during the mounting or dismounting process.

## Mounting the Chuck

The first step in mounting any of the three types of chucks is to lock the spindle or put it in the lowest speed range so that the spindle will not turn while the chuck is being mounted.

When mounting a *cam lock* chuck, make sure the cams of the chuck are lined up with the holes in the spindle. When these are lined up, slide the chuck, resting on the cradle block, so that the cams are in the holes, then use the wrench to tighten the cams evenly. Marks on the chuck show exactly the point to which the wrench should be tightened.

When mounting the *tapered spindle* chuck, line up the key on the taper portion with the keyway on the spindle. Slide the chuck onto the spindle and with the proper spanner wrench, securely tighten the chuck.

When mounting the *threaded spindle* chuck, slide the chuck to the spindle and then screw the chuck on. When the chuck is on the spindle, back off about an eighth of a turn, and with a snap, tighten the chuck.

## Turning a Round Part

The procedure for turning a round part is as follows.

1. Mount a three-jaw chuck and clamp the material in it.
2. Mount a right-hand toolbit.
3. Set the proper cutting speed.
4. Set the feed rate.
5. Determine the depth of cut.
6. Center the tool.
7. Turn the lathe on, and touch the work with the tool.
8. Move the tool to the end of the piece.
9. Set depth for roughing cut.
10. Make roughing cut.
11. Change speeds and feeds for finishing cut.
12. Make finish cut.

## Turning an Irregular Shape

The procedure for turning an irregular part is the same as for turning a round one except that the irregular part must first be aligned.

## Aligning Irregular Part for Turning

An irregularly shaped part is aligned in one of two basic ways, the chalk method and the indicator method.

The chalk method is accomplished in the following manner

1. Mount the work in the chuck. Use the lines of the chuck to evenly space the jaws for the diameter to be turned.
2. Put the material in the jaws and tighten each jaw equally.
3. Set a slow spindle speed, turn on the machine, and with a piece of chalk held in one hand, with that hand supported on the carriage, touch the work lightly so as to hit the high spot as the spindle rotates.
4. Loosen the jaw opposite the high point, then tighten the high-point jaw to move the part down and retighten the low-point jaw. Repeat until chalk leaves one solid line around the part.
5. The part can now be turned, using the procedure outlined above for turning around parts.

The chalk method is a fast and simple rough-and-ready way to line up the irregular part for turning.

If the diameter is to be turned in relation to a finished diameter, then in addition to using chalk to line up the part, an indicator is used. The indicator will ensure a much more precise line-up of the chuck jaws.

The process is as follows.

1. Align the part using the chalk method.
2. Mount an indicator on the bed of the machine.
3. Touch the indicator to the finished part of the workplace as a reference.
4. Loosen and tighten the jaws until the indicator reads “0” all the way around.

## TAPERS

Any object that changes size gradually and uniformly along its length is said to be “tapered.” A *wedge*, for example, decreases at a regular and steady rate from its apex to its base and therefore it is “tapered.” Although wedge-type square and rectangular tapers are utilized as keys to fasten machine parts together, most tapers used in the shop are cylindrical. Since workpieces revolve about their centerlines, tapers turned on a lathe normally will take a cylindrical or conical shape—round at any point, but increasing or decreasing in size (diameter) at a uniform rate from one end to the other.

In the machine shop, tapered parts and components are both vital elements of the machine tools and accessories used in day-to-day operations and constitute the kinds of work the machinist is routinely called upon to handle.

The spindles of lathes, drill presses, and milling machines as well as other devices feature internal tapers that permit a perfect fit and firm grip between it and such items as drills, reamers, arbors, centers, and cutters of all kinds with matching external tapers. In addition to promoting a good fit and a tight friction-grip between the mating surfaces, matching tapers automatically align the centerlines of the spindle and the device being inserted, ensuring that tool alignment will be “true.”

For many machining operations—aligning and using drills, taps, reamers, and end mills, for instance—friction alone is sufficient to hold the tool in place for machining. Some tapers, however, have too sharp an angle to be held by friction alone, so these are designed to be pulled into position by a threaded device known as a draw bar. Tangs that fit into machined recesses also are used to ensure a positive drive.

There are a great many “standard” tapers, but the ones most commonly used in the shop are:

- \* Morse tapers, which are most commonly used on drill presses and lathes.
- \* Brown & Sharpe tapers, used mainly on milling machines, grinding machines, arbors, and machine tool spindles.

Other tapers encountered in shop work are:

- \* Jarno tapers, used on profiling machines, die-sinking machines, and for the spindles and tailstock tapers of lathes.
- \* American Standard machine tapers. Mainly found in milling machines, these are held in place by means of a draw bar.

- \* British Standard. Used by British shops for self-holding, quick-release, and chuck tapers; it uses some metric, Brown & Sharpe, and Morse sizes.
- \* Jacobs tapers, used for drill chucks and spindles.

The various tapers are presented in depth in *Machinery's Handbook* (Note: pages vary with edition. These are from the 22nd edition): Amer. Std., pp. 1734, 1740–4; Brit. Std., p. 1736; Brown & Sharpe, pp. 1734, 1738; Jacobs, p. 1747; Jarno, pp. 1734, 1739; and Morse, pp. 1734, 1737 (Morse stub taper shanks, p. 1736).

## Taper Turning

When setting the lathe for one of the above-mentioned tapers, refer to the appropriate *MH* citation plus pages 1594 and 1595, “Taper turning, with combined feeds” (21st edition).

There are three basic ways to turn a taper on a lathe:

1. Tailstock. Offsetting the tailstock to create the desired taper.
2. Compound Rest. Setting the compound rest at the angle desired.
3. Taper Attachment. Using a special taper attachment.

The *tailstock* method has several major drawbacks: (1) The tailstock center is offset and therefore is no longer in line with the center of the headstock, thus time must be spent using indicators to realign the tailstock after cutting the taper. (2) Only gentle tapers can be accurately cut; if the taper is more or less “sharp,” inaccuracy occurs. (3) Since the workpiece is in effect being purposely “wobbed,” wear can occur in the center hole, also causing inaccuracy of cut. (4) It cannot be used for internal tapers. For these reasons, this method should only be used when it is essential to cut a taper and neither of the other basic methods is available.

The procedure for this method is as follows

1. *Center* the stock with a center drill (see page 55) and turn the stock to the required dimension for the largest end for the part, as listed in *MH*.
2. *Calculate* the taper. Tapers are calculated either in taper per foot or taper per inch; see *MH*. Example: To machine a No. 2 Morse taper, look at the chart in *MH*, page 1735 (22nd edition). The small end is listed as 0.572 in., the large end at 0.700 in. The length of the plug, including the tang, should be 3.125 in.; the tang itself is 0.4375 in. That puts the taper length, not including the tang, at  $2\frac{9}{16}$  in.)

Another method of making the calculations for the tailstock offset approach is to use this mathematical formula:

Tailstock offset =  $\frac{1}{2}$  taper per inch  $\times$  Length. The *MH* table lists the taper per inch as 0.04995 in.

Length, without tang, is  $2\frac{9}{16}$  in.

Substituting these values in the formula gives:

Tailstock offset =  $\frac{1}{2}$  (0.04995)  $\times$  2.562

Tailstock offset = 0.64 in. or  $\frac{1}{16}$  in.

3. *Offset* the tailstock. There is a set screw on the bottom of the tailstock at the front and one opposite it at the rear. Loosen the front one and release the back one, and move the front of the tailstock on in to achieve the required offset (0.64 in., in our example). Check the offset by moving the tailstock close to the headstock and comparing centers. When the correct offset distance has been achieved, snug up the set screws so the tailstock cannot shift during machining.
4. *Mount* the work between centers, center your tool, and make a roughing cut.
5. *Determine* amount of stock to be left for secondary operation. If the part is to be used directly from machining, then leave a few thousandths for polishing with an emery cloth. If the part is to be hardened, leave about 0.010–0.015 in. for finish grinding.
6. *Check* the work. Make any adjustments in taper that are necessary, and when that is satisfactory, make the finishing cut to bring the work to size.
7. *Realign* the tailstock when the work has been completed, indicating it in for proper accuracy.

Tailstock realignment may be done in several ways:

1. The fastest, easiest, and surest method is to use a straight mandrel 12 in. long. Mount it between centers and indicate it in from the tailstock to the headstock, adjusting the tailstock set screws until a zero reading is obtained throughout the length of the mandrel bar.
2. Put a part in a chuck and turn it to the diameter of the tailstock spindle. Position the tailstock 12 in. from the headstock, then indicate the tailstock spindle with an indicator mounted to the carriage. Move the indicator to the headstock and indicate the part turned. Repeat this procedure, moving the tailstock by loosening and tightening the set screws until both readings are “0.”
3. Turn a piece 12 in. long and indicate it using the method listed in No. 2, above, adjusting the tailstock until the indicator reads “0” along the entire length of the piece.

The *compound rest* method is routinely used for cutting tapers, particularly when making centers, chamfers, and other types of sharp angles. The method is fast and accurate, the work can be mounted either on centers or in a chuck, and internal angles can readily be machined by this method using a rigid boring tool.

The procedure is as follows.

1. *Mount* the work in a chuck or between centers.
2. *Select* the proper cutting tool and center it.
3. *Adjust* the compound to the angle (taper per inch) desired, using the graduated scale on the compound.
4. *Turn* the machine on, and, feeding the tool into the work with the compound feed handle, remove the unwanted material until the piece achieves correct size.
5. *Finish*—leave a few thousandths and polish with an emery cloth or leave 0.010–0.015 in. for finish grinding after hardening.

The *taper turning attachment* is an excellent tool and is the preferred method when a wide variety of taper types and sizes are routinely cut, especially long, gradual tapers. It cannot, however, be used for very sharp angles; these must be cut using the compound rest method, above.

The procedure for using the taper turning attachment is as follows.

1. *Engage* the taper slide bar to the clamp on the back way of the machine. This holds the taper attachment in place, preventing it from moving with the carriage.
2. *Set* the taper attachment at the angle of taper per foot desired. Most taper attachments have scales at each end, one showing the angle in degrees, the other in taper per foot.
3. *Mount* the cutting tool, center it, and machine the taper leaving a few thousandths for polishing with an emery cloth or 0.010–0.015 in. for hardening and finish grinding.

## THREADS

The word “thread” comes from the Old English term *thread* (*thrawan*—to wind, to twist) and is defined as: The prominent helical part of a screw—a helical ridge of uniform cross-section formed on the inside or the outside of a cylinder or a cone.

### Functions

Threads take a variety of forms, but all serve one of three basic functions: (1) *fastening*—screws, bolts, mounting lugs, pipe fittings, etc.; (2) *power transmission*—leadscrews, driving gears, jack lift screws, etc.; (3) *measurement*—micrometers, other measuring devices.

### Types

The six basic types of threads used today are:

1. *American*—A “V” thread with a 60-degree included angle; the upper crest and base are both sharp. Commonly used as a *fastening* or “locking” thread.
2. *Whitworth*—A British thread with a 55-degree included angle; slightly rounded crest and base. *Fastening* uses, similar to those for the American.
3. *Unified*—A thread developed for common use by the United States, Canada, and Great Britain, replacing American and Whitworth; used by the armed forces of these nations to ensure interchangeability of threaded parts. *Fastening* and *measurement* uses, similar to Whitworth and American.
4. *Acme*—A thread featuring a 29-degree included angle; flat crest and base. *Power transmission* uses such as feed screws, jack and vise screws, etc.



5. *Buttress*—A thread with a “driving” side that is straight while the rear portion is at a 45-degree angle. For special *power transmission* uses such as screws for heavy-duty jacks where power is to be applied in one direction only.
6. *Square*—A thread with a 90-degree angle. Sturdy, capable of handling very heavy loads, but hard to machine; Buttress or Acme now typically are used for its *power transmission* applications.

## Thread Characteristics

Threads (see Figure 3.14) are formally defined in terms of the following specific features: (1) outside diameter; (2) number of threads per inch; (3) pitch; (4) pitch diameter; (5) class of fit (Class I to V), and external (A), internal (B).

The *outside diameter* and *number of threads per inch* are pretty much self-explanatory.

The *pitch* equals 1 in. divided by the number of threads per inch. For example, a screw with 8 threads per inch has a pitch of  $\frac{1}{8} = 0.125$  in.

The *pitch diameter* is the diameter of the point at which the male thread contacts the female thread as the thread advances along its helix.

The *class of fit* is alternately known as the *class of thread*. The basic pitch diameter for a given thread is determined by thread standards and is outlined in a government publication (H-28). These standards are shown in the *Machinery's Handbook* and can be found by referring to the “Threads” listings in the index.

The *class of thread* is then determined from the basic pitch diameter. In general, these are as follows.

Basic Pitch Diameter (in.)	Class	Comment
+0.0005	I	Very loose fit
+0.0010	II	Fine fit
+0.0015	III	Close fit
+0.0020	IV	Tight fit
+0.0025	V	Interference fit

When tapping threads, the tap designation generally will include an indication of the fit; examples would be H-2 or H-3, meaning a high Class II or Class III fit, or L-1, meaning a fit lower than Class I for the basic pitch diameter. Care must be taken, however, when selecting a tap for a class fit. For example, since class 2 is looser than class 3, a tap with an H-3 size is to be used because it will cut a larger thread.

## Measurement

When gaging threads with thread gages, the gages will indicate their sizes by class fit and a symbol next to the class fit, such as “A” for an external thread or “B” for an internal thread.

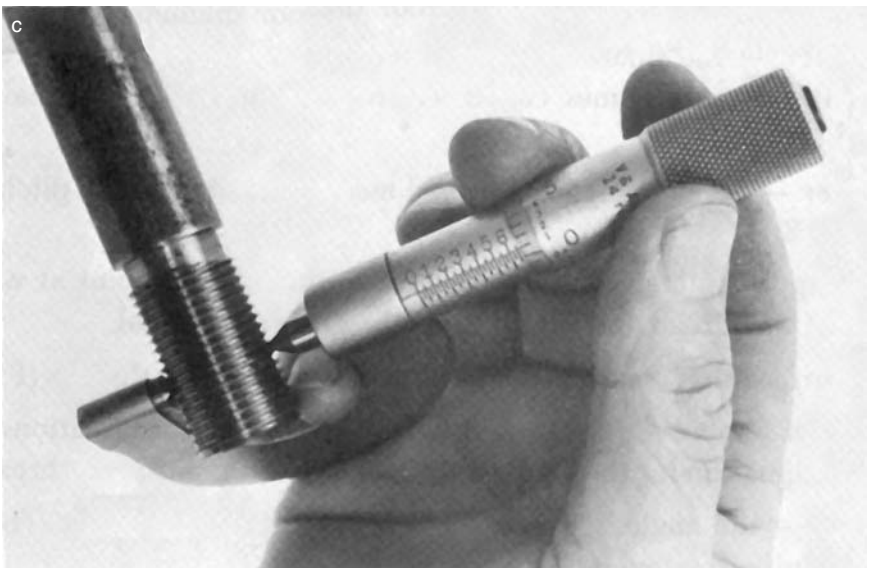
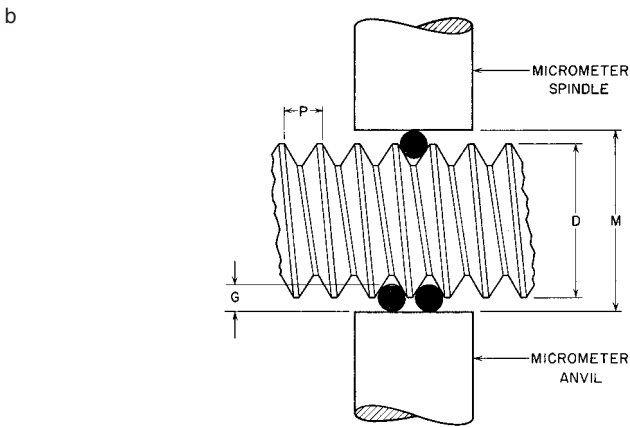
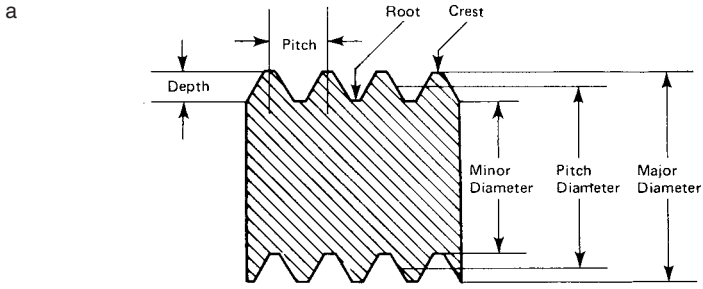


Figure 3.14. (a) Thread nomenclature; (b) checking the thread via the three-wire system; (c) measuring threads with a thread micrometer.

Threads can be gaged in various ways:

1. *Thread Micrometer.* This instrument features a conical spindle and a swivel V-shaped anvil. The swivel compensates for the thread lead. The micrometer comes marked for a series of threads it is capable of measuring.
2. *Three-Wire System.* The wire is purchased as a set for measuring a range of threads. A formula is provided on the case housing each set of wires that allows for the proper height gage (if missing, refer to *Machinery's Handbook*). The proper gage pressure is important when using this approach. This is the most accurate method of checking threads.
3. *Optical Comparator.* This is a most convenient method when using a thread overlay chart, with the threaded part mounted between centers.
4. *Threaded Ring Gage, Threaded Plug Gage.* This method is excellent for checking threads, particularly on production lines. It is the only way that internal threads can be checked. Gages are readily available from vendors and can be obtained with any degree of accuracy required. When using a thread ring or plug gage, a “go” and “no go” gage is used. This type of gage requires no extra training or skill to use.

## THREAD CUTTING ON A LATHE

Because it rotates cylindrical parts about their centerline axes, the lathe is ideally suited for creating threads—internal as well as external.

Threads can be cut on a lathe in a number of ways:

1. By *single point turning*:
  - a. *External.* Threads are created on the periphery of the workpiece.
  - b. *Internal.* Threads are turned on an inner surface created by drilling, reaming, boring, etc.
2. By *tapping.* Threads are created on an internal surface, normally following the drilling of a slightly undersized pilot hole—that is, a hole with a diameter approximately the same as the “pitch diameter” (diameter at a point on the thread about midway between the outer or “major” diameter and the inner or “minor” diameter of the completed thread).
3. By *die chasing.* Threads are “chased” (“chase” is the shortened form of the term “enchase” from the Old French *enchasser*—to engrave, to adorn with raised or embossed work) and finished using a die.
4. By *rolling.* Similar in important aspects to die chasing, except that instead of cutting and removing metal to create threads, metal is displaced into the bottom of the die grooves to form the upper portion of the threads. This approach, obviously, is best suited for malleable materials. Dies take two forms:
  - a. *Flat thread-rolling dies.* Cylindrical work is rolled through the grooves of the flat die; the grooves are forced into the surface, displacing metal into the die grooves to create the threads.

- b. *Cylindrical thread-rolling dies*. These work much the same as flat, except that the dies are cylindrical and two of them normally are mounted in a special attachment for the rolling operation.

## Threading on a Lathe

Threading on a lathe requires a somewhat different procedure than that used for ordinary turning. A single point tool is used and it is generally more fragile than a turning tool. At least two or more procedures may be necessary to have the finish a satisfactory thread requires.

As noted, there are various types of threads. The V 60 degree American thread, the 55 degree Whitworth thread, the acme thread for power, the square thread [not used very much but still utilized for special applications], and the pipe thread used for sealing. Below are some examples of speeds and feeds for single point threading.

Material	Thread Dia	Infeed		Hi Speed Tool	Carbide Tool
		1st Pass	Last Pass	Speed	Speed
Carbon Steels	1–6 inches	.020	.0005	55 FPM	250 FPM
Free Machining	1–6 inches	.020	.0005	55 SFM	250 SFM
Nickel Alloys	1–6 inches	.015	.001	30 SFM	150 SFM
Stainless Steels	1–6 inches	.015	.001	20 SFM	110 SFM
Titanium Alloys	1–6 inches	.020	.0005	10 SFM	80 SFM
HI Temp Alloys	1–6 inches	.010	.001	15 SFM	70 SFM
Copper Alloys	1–6 inches	.010	.001	100 SFM	250 SFM
Aluminum Alloys	1–6 inches	.020	.001	175 SFM	250 SFM

Cutting fluids should be used when cutting threads at these low speeds. Oil can be used for fine finish. When in production runs, a heavy-duty synthetic cutting fluid is recommended. When threading aluminum, chip welding is a common problem, so do not try to thread aluminum alloys without a good cutting fluid.

## Turning Acme and Pipe Threads

These are differentiated from other threads in part by their use. Acme threads typically are used for machine tool leadscrews and similar applications.

Pipe threads are typically tapered and are used to connect pipes carrying liquids or gases: they also have construction uses (scaffolding, fences, handrails, etc.).

## Turning Worms

Technically—depending on how one interprets it—this may or may not constitute “threading” since instead of threads the operation produces “worms” (not to be confused with ones suitable for fishing—these worms are used with meshing,

tooth-type “worm gears” in speed reducers, and other heavy-duty applications requiring high-speed ratios). On the other hand, dictionaries define a “worm” as “anything helical: the thread of a screw,” so from that perspective, maybe a “worm” is a “thread.”

## Turning Threads

Since single-point turning is the most common approach used in cutting threads, it’s the logical place to begin.

### The Threading Tool

Turning tools used in various other methods of machining may vary with the personal preferences of the machinist. The threading tool, however, cannot vary in its requirements because the surface finish on threads cut with a single-point thread-cutting tool is influenced by the relief angles on the tool.

### Relief Angles

The angles that form the side and front of the tool must incorporate an adequate amount of relief—too little relief results in rubbing and produces a rough finish, whereas too much relief weakens the tool and can cause both poor finish and premature tool failure.

### Other Factors

Other factors governing the tool are the machine, the material to be cut, the cutting speed, the cutting fluid used, and, of course, the condition of the tool itself.

The tool can be ground by the machinist or a commercially purchased tool can be used.

### Thread Selection

Lathes vary in their abilities to cut threads *without* changing gears. Depending on the size of the machine, some lathes may have as many as 97 choices of thread without replacing gears, whereas others provide as few as 16.

The quickest way to complete the process of turning a thread is to use the same tool for both roughing and finish cuts. Not surprisingly, that procedure, described below for cutting external threads, is the one most commonly used.

## Cutting External Threads

The procedure for cutting external threads is as follows.

1. Select the setting on the lathe for the thread to be cut (some lathes require four settings, others five).
2. Mount the material to be machined in a chuck or between centers, depending on the part to be produced.

3. Set the compound on the carriage to an angle that is about 1 degree less than half of the included angle of the thread. For example, if the thread to be cut is a unified or American “V” thread, the included angle is 60 degrees. Half of that is 30 degrees, so the angle would be set at 29 degrees. This eliminates any possibility of rubbing and drag by the rear portion of the tool, permitting the cutting edge of the tool to remove metal freely and cleanly. When turning an Acme thread (see Figure 3.15), half of the included angle is  $14\frac{1}{2}$  degrees; in this case, setting the compound to 14 degrees is sufficient to preclude drag.



*Figure 3.15. Turning an Acme thread.*

4. Insert the cutting tool into the toolholder and center it. Holding the center gage against the straight portion of the work held in the chuck or between centers, align the tool. This is accomplished by holding the center gage to the part to be machined and shifting the toolholder to position the tool so that it fits snugly into the center gage notch.
5. The center gage can be used to check the angles on the cutting tool as well as to align the tool to the work. It is calibrated with a scale that permits checking the number of threads per inch. Separate center gages are made for use with 60-degree threads and with Acme (29-degree included angle) threads.
6. The thread depth is calculated from the crest to the root of the thread. About 75% of the thread pitch is the depth of cut for most threads.
7. Set the gear control, located on the machine panel, for thread cutting. This (1) connects the leadscrew so it will turn when the machine is running and (2) disconnects the feed shaft. A half-nut lever is used to engage the leadscrew; proper lead for the thread to be machined is selected by engaging the thread dial, located next to the carriage, at the

proper setting. When machining even-numbered threads per inch—8, 10, 20, etc.—any line on the thread dial can be used as a guide for engaging the half-nut lever. When, however, odd-numbered threads per inch are to be cut, then the lever can only be engaged when the odd-numbered line—1, 3, 5, etc.—lines up properly. The thread dial on all lathes is marked to clearly show when the half-nut is to be engaged for a particular thread per inch setting.

8. Turn the lathe on, and just touch the threading tool lightly to the work to establish the starting point. After touching the work, loosen the cross-feed screw micrometer-collar set screw, set the collar to zero, and retighten the set screw. This provides the reference point for subsequent machining.
9. Assuming all settings are correct, running the lathe at the correct spindle speed will provide the best finish attainable on that machine. Note, however, that the half-nut must be disengaged at the end of the cut, a deceptively simple maneuver that requires excellent hand coordination and an adequate amount of practice. Consequently, both the beginner and the experienced machinist who has not cut threads for any appreciable period of time should make at least a few practice runs using a slow spindle speed until they gain or regain confidence in their ability to disengage cleanly and accurately. Note also that when turning a thread to a shoulder, a relief must be machined to permit cutting the thread all the way to the shoulder.
10. After touching the work lightly with the tool, move the tool out past the end of the work and then advance it a few thousandths of an inch. Now, engage the half-nut, timing it to a proper line (any line for even-numbered threads, *odd only* for odd-numbered threads) for the thread to be cut, and using cutting oil or cutting fluid, liberally applied, make the first cut. Take care to disengage properly to prevent damage to the tool and/or workpiece and machine.
11. After completing the initial cut, check the threads with a thread gage to ensure that the number of threads cut per inch is correct.
12. After confirming that all is correct, make successive passes to complete the thread. When each cut is completed, turn the cross-feed screw out one turn so the tool will clear the work on the return path. Return the carriage to the beginning of the work, dial the cross-feed screw back to zero (its original position), move the compound in 0.0020 in., and continue this process until the thread reaches its proper depth.
13. Check the thread, using a micrometer thread gage, a thread ring gage, or a mating part.

## Cutting Internal Threads

Internal threads are turned in a manner very similar to the way external threads are cut. The cutting tool used and the setup of the compound is the same as for external threads. The hole to be threaded, however, must be either bored or else drilled and reamed prior to threading. Depending upon the accuracy of finish thread required, the threaded hole can be finished to size using a tap to chase the

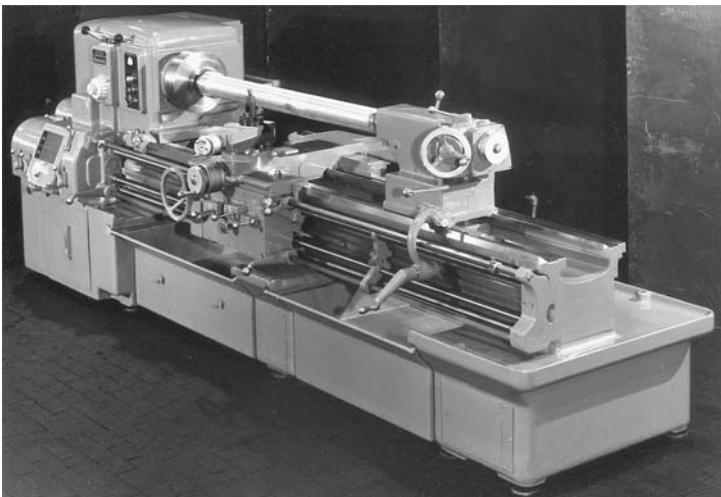
thread or, alternately, by machining it and checking the thread with a thread gage, a “go/no go” gage, or a mating part.

## Cutting Acme Threads

The procedure for turning an Acme thread is similar to that for single-point thread cutting. The compound, however, should be set at a 14-degree angle, and the tool is centered to the work using an Acme center gage. The tool is fed into the plane of the work using the compound, and the cross-feed is used to locate the reference point of the tool.

The procedure is as follows

1. Grind the tool to an included angle of 29 degrees.
2. The back rake angle, as is true of all threading tools, will amount to zero degrees. The toolholder will determine when a back rake angle must be ground:
  - a. When a quick-change or open-side holder is used, the back rake angle is zero.
  - b. When a rocker-type toolholder is used (see Figure 3.16), then the angle of the back rake must match the angle of the rocker holder so that combined result equals a zero back rake.
3. Set the compound at 14 degrees and center the tool with the Acme center gage.
4. Zero the cross-feed handle after touching the work with the cutting tool. Move the carriage to the end of the work. Feed the tool in with the compound, and then machine the thread.



*Figure 3.16. Turning on a lathe using a rocker-type toolholder. (Courtesy of Monarch Machine Tool Co.)*



5. Check the thread with a mating part or else use the one-wire method. The wire size can be determined by multiplying the constant, 0.4872, times the pitch. (Example: Assume there are 8 threads to the inch. Since the pitch is 1 divided by the number of threads per inch, the result is:  $\text{Pitch} = \frac{1}{8} = 0.125$ . Consequently, the wire size will be:  $0.4872 \times 0.125 = 0.0609$  in. in diameter.)

## Turning Pipe Threads

Pipe threads are tapered because tapered threads permit tighter connections than are possible with a straight thread.

Pipe threads are used for devices and systems designed to contain and transfer liquids, chemical fluids, and gases of all kinds with no leakage.

The thread angle is the same as that used for the Unified and for the American “V”—60 degrees—and pipe thread dimensions and characteristics are governed by the American Standard pipe thread series.

Since the diameter of a tapered piece varies, the pitch diameter of a tapered thread is measured at the small end of the taper.

When grinding the threading tool, the back rake on the threading tool should be kept at zero degrees. For the tool to be properly ground, the crest and the root must be slightly flat. The tip of the threading tool is generally equal to a width 0.033 in. times the pitch, which amounts to the distance from one thread to the next and is calculated by dividing 1 by the number of threads per inch.

The procedure for turning a pipe thread is as follows.

1. The taper is 0.75 in./ft. The taper is guided by the taper attachment; set the taper attachment to  $\frac{3}{4}$  degrees.
2. The included angle is 60 degrees, the same as for Unified threads. Set the compound to one-half of the included angle less 1 degree—that is, to a setting of 29 degrees.
3. Center the cutting tool and align it along the straight edge of the work with a center gage.
4. Take a light cut at first, and check the number of threads per inch. The taper attachment will correctly guide the tool, but there may be backlash in the attachment. To eliminate any backlash, after each cut, be sure the carriage is returned far enough past the end of the work to remove the “play” as the carriage is engaged and moves the tool into the work for the next cut.
5. Follow the procedure for all threading operations: After touching the work with the tool, zero the work with the tool, zero the cross-feed micrometer gage, make a cut, back off one turn so the tool will clear the work on the return movement, then set it back to zero to prepare for the next cut. Now move the compound in a few thousandths and remove the material as required. Repeat until the thread is completed.
6. Check the part, using either a mating part or a pipe thread gage. *Machinery’s Handbook* specifies the length for a hand-tight engagement—that is, approximately four threads past the first few imperfect threads.

7. Whenever machining threads, a lubricant or heavy cutting fluid must be liberally applied to ensure good finish and satisfactory tool life.

## Tapping Threads

It is common practice to tap threads on a lathe. It is much more practical, for example, to tap smaller holes—up to about  $\frac{3}{4}$  in.—than to cut internal threads. To achieve the same accuracy as is obtained by turning internal threads, the hole must first be bored out to correct size on the lathe so that the tap will be guided by the hole and maintain its accuracy.

The procedure for tapping threads on a lathe is as follows.

1. Mount the part in a chuck and a center drill in the tailstock. Drill a pilot hole in the work. The pilot hole can be half the diameter of the hole the next operation will create.
2. Determine the accuracy required for the tapped hole. If the requirement is exacting, then bore the hole with a boring tool. This is generally practical for holes of  $\frac{1}{2}$ -in. diameter or greater.
3. If no boring is required, drill with the tap drill following drilling of the pilot hole. A tap drill is a drill size smaller than the tap, to leave the proper amount of metal to permit cutting the specified thread size. The tap drill is selected using *Machinery's Handbook* tables or charts provided by tap manufacturers.
4. Chamfering the hole first will ensure very accurate alignment and prevent creation of a burr at the start of the thread.
5. Larger taps have a center hole in the shank of the tap that can be used to guide it with a center in the tailstock. A tap wrench is mounted on the tap before centering the tap with the tailstock. Smaller taps do not have a center hole, so a T-handle tap holder should be used to mount on the tap end. This T-holder has a center hole in its end that can be used with the center in the tailstock for accurate alignment of the tap. Project No. 1, located at the end of this chapter, is for machining a T-tap wrench. The step-by-step procedure is given, along with drawings of each component.
6. The tap should be introduced into the hole and given a few turns by hand while applying pressure against it with the tailstock. It is easier to start with a taper tap (this tap has a seven-thread taper lead), so if the hole to be tapped goes completely through the work and thus has clearance for the tapered section, the taper tap can be used. If the hole goes through the work, and a taper tap is not available, then a plug tap (it has two- to three-thread lead) can be used.
7. After making a start by hand, back off the tap one or two turns to clear the chips from the flutes.
8. Now, using lubricant, liberally applied, and the lathe set at its lowest spindle speed setting, locate the position of the tap wrench so that the handle is braced by the compound and sufficient pressure is applied by the center in the tailstock. With spindle speed at lowest setting, start the lathe, and turn the tailstock handle to keep pressure on the tap as the part

turns. The tap will draw itself into the work while the machine is running, and the wrench handle will keep the tap from revolving while it is held in place by the compound. If the depth of the hole is more than  $2\frac{1}{2}$  times the diameter of the tap, stop the machine and reverse it frequently to clear the tap of chips and prevent breakage.

9. After finishing the thread, reverse the machine to remove the tap. Be sure there is support on the opposite side of the carriage for the tap wrench.

## Threading with a Die

The procedure for threading with a die is similar to that for tapping, except that the threads are created on the outer periphery instead of internally.

Lubrication is essential, and die threading normally is performed mainly on small rod stock. Care is necessary to prevent the rods from bending because of the high torques created; in view of this tendency to twist under the applied torque forces, no more than 0.005 to 0.010 in. of stock should be removed at a time.

Dies are also often used to chase threads that have previously been machined.

## Cutting Worms

Worms and worm gears are used to transmit motion and power, with common uses including speed reducer applications. Fine-pitch worms are primarily used for motion, while coarse ones are used for heavy-duty power applications.

Whereas toothed worm gears are made with hobs, the worms that drive the gears can be made in several ways, including turning them on a lathe.

The pitch diameter of the worm is defined as diametrical pitch (the American standard) or modular pitch (the metric standard). Most lathes will have these listed near the thread selections posted on the lathe.

The procedure for turning worms is as follows.

1. Grind the turning tool to an included angle of 40 degrees.
2. The width of the tool point is critical. The width is listed in *Machinery's Handbook* after the diametrical pitch; determine the width and then grind it on the tool. For high accuracy, the angle and tip can be measured on an optical comparator.
3. Select the gear ratio. This is accomplished by changing the gears in the rear of the machine. Usually it is merely a matter of dismounting one gear, putting a spacer in its former place, then remounting the gear and adjusting it until it meshes with its mating gear.
4. Set the quick-change gear box for cutting the pitch required.
5. The lead on a worm is very fast, so consequently it may be necessary to adjust the gear lever for a fast lead. On some machines, when the worm ratio is set, the gear lever is automatically shifted over.
6. Proceed to turn as in regular threading. A worm is most often turned on a lathe to replace one worn or damaged in use. In that case, the "fit" is

judged by how smoothly the worm meshes with the worm gear it moves. The pitch naturally should coincide exactly with the pitch on the gear itself.

## DRILLING ON A LATHE

About 75% of all machining consists of putting holes into the work. Holes produced by lathes are extremely important because they are not only necessary for straight fits but are the first step in creating threads and are needed to make cams and for a multitude of other applications.

*Drilling* is frequently the first operation to be performed on a lathe, whether the step is to use a center drill prior to mounting between centers, drilling a pilot hole with a small drill to direct a larger drill, or to set the stage for such further operations as boring, reaming, tapping, and threading.

### Center Drilling

The center drill—more formally known as a drill-and-countersink combination—is the drill most commonly used on the lathe to create starter holes, so we'll start there.

The workpiece is held in one of two basic ways:

1. By a lathe chuck. A three-jaw chuck normally is used to hold round stock when the starter hole is to be drilled in the center of the piece. A four-jaw chuck is usually used when the hole is to be placed anywhere in the workpiece *but* the center, such as when a cam will be made.
2. Mounted to a face plate. The work is mounted to a face plate when accurate alignment with the face of the work is critical; the hole can be created anywhere on the part, i.e., in the center or elsewhere.

### Drilling Safety

Safety precautions must be taken when drilling on a lathe. Because of the nature of the tool, chips come out of the flutes in long, stringy strips. Stopping the drilling action at frequent intervals breaks the chips into smaller pieces that flow more easily out of the flutes, minimizing potential problems with the chips. If chips come out long and stringy because they have not been broken, the safest course is to remove them with a pair of pliers. **Do not use hands to remove chips!**

### Lubrication

Lubrication is critical when drilling deep holes, both to help the chips move freely up through and out of the flutes and to cool the tool-workpiece interface. Because of the confined working conditions in which the drill operates, much heat is generated at the cutting edge of the drill.

## Drilling Procedure

To perform a drilling operation on the lathe, use the following procedure.

1. Mount a center drill in a drill chuck and place the drill chuck in the tailstock. If the hole is to be used as a pilot hole for a larger drill instead of for mounting between centers, be sure the pilot hole size will be slightly larger than the web of the large drill to follow.
2. Look up the cutting speed for the workpiece material. It is given in SFPM (surface feet per minute), so using the formula,  $\text{RPM} = \text{cutting speed} \times 12 \text{ divided by } \pi \times \text{diameter of the drill}$ , calculate the RPMs to be set. (A more detailed explanation of the formula for converting SFPM to RPMs is contained in Chapter 7.)
3. Determine the depth of the hole. For a center drill hole, the depth should be about half of the countersink section of the drill.
4. Apply cutting fluid liberally and drill to indicated depth. *If that hole is to be used as a pilot hole, go on to the following steps.*
5. Substituting the diameter of the larger drill, recalculate the RPM setting and set it on the machine.
6. Applying cutting fluid, feed the drill into the work at a slow, steady rate using the handwheel of the tailstock. If the feed pressure is too great, the drill will break; if the RPM setting is too low, that also can cause the drill to break.
7. Continue until the drill exits the work or reaches the depth required.

## Boring on a Lathe

Boring is a secondary operation, performed following drilling, that provides a way to enlarge and true a hole.

Normally, only a small amount of material is left for the boring operations—about  $\frac{1}{16}$  to  $\frac{1}{8}$  in. of stock.

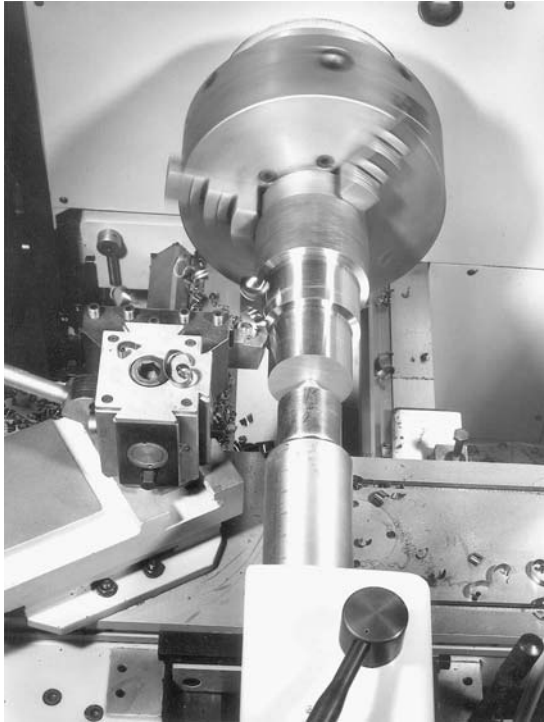
There are a number of styles of boring bars and Toolholders for them that are used on lathes. One popular type has end caps that fit on the end of a boring bar and utilize insert toolbits that are ground like all other toolbits used for turning. The various holders allow for setting various angles to accommodate boring requirements.

Toolholders are positioned at 90-, 45-, and 30-degree angles and come in a variety of diameters depending on the size of the hole to be bored.

In addition to the foregoing, there are one-piece forged boring bars of various diameters; there are also solid bars that accept insert toolbits. All of these can accept carbide, high-speed steel tools or cast alloy tools.

The procedure for boring is as follows.

1. Set the boring tool so that it is aligned with the centerline of the workpiece. The easiest way to do this is to line it up with a center mounted in the tailstock. With a quick-change holder (Figure 3.17), the tool can be swiveled to line it up. With a rocker-arm holder, the centerline will have



*Figure 3.17. Overhead view of turning using a quick-change holder.  
(Courtesy of DoAll.)*

to be estimated with the hole to be bored and then corrected if not exactly in the center.

2. Mount the work in a chuck or to a face plate, depending on the requirement. Be sure the boring tool is suitable for the work.
3. Set the correct RPM for boring. The speeds are the same as for straight turning. The diameter of the hole determines the cutting speed required and thus the RPM setting.
4. Set the tool for a light cut, and with the machine running, advance the tool into the work about  $\frac{1}{8}$ – $\frac{1}{4}$  in.
5. If the setup is satisfactory, set a depth of cut of 0.015–0.020 in. for stock removal—remember to use plenty of cutting fluid for good lubrication and a fine finish. Check size with a telescopic gage and micrometer.
6. Turn the material and remove the stock to the required length. Check and, applying lubrication liberally, machine the work until the specified size is attained. Do not take too heavy a cut—that can cause the boring tool to deflect, leaving a tapered hole.

Boring requires using the most rigid holder that can enter the hole to be enlarged, since deflection can distort the finished hole. That, along with achieving fine finish, is why a minimum finish cut is necessary.

Material	Depth of Cut	High Speed Steel Tool		Carbide Tool	
		Feed IPR	Speed SFM	Feed IPR	Speed SFM
1015–1035	.050	.005	115	.005	660
1040–1095	.050	.005	95	.005	650
1132–1151	.050	.005	90	.005	525
1335–1345	.050	.005	110	.005	455
2330–2345	.050	.004	40	.005	260
3115–3130	.010	.002	35	.003	190
4032–4042	.050	.005	90	.005	420
4130–4137	.050	.005	110	.005	455
4140–4340	.050	.005	110	.005	455
4615–4621	.050	.005	110	.005	510
5130–5135	.050	.005	110	.005	455
50100–52100	.050	.005	95	.005	435
6145–6152	.050	.005	110	.005	510
8647–8660	.050	.005	105	.005	455
9255–9260	.050	.005	105	.005	455
T1-T2-T6	.050	.005	65	.005	365
M1-M2-M6-M10	.050	.005	60	.005	350
T15-A7-D3-5	.050	.005	60	.005	350
201–304	.050	.005	90	.005	375
303	.050	.005	105	.005	490
5056–7178	.050	.005	800	Alum. Wrought	
A332–B850	.050	.005	800	Alum. Cast	

### *Boring on the Lathe: Cutting Tool Material*

The above speeds are merely a few examples of cutting speeds and feeds for a typical variety of materials. Naturally, speeds and feeds will vary depending on the amount of material that is to be removed when finishing, taking into account the hardness of the material. The chart above is based upon a material hardness at about 200 Brinell

## **Reaming on a Lathe**

Reaming is a machining operation designed to bring a hole to exact size (see Figure 3.18). If the reamer is a standard supplied reamer size, it provides the quickest and most accurate way to finish a hole when used following a boring operation.

The procedure is as follows.

1. In boring a hole to be finished by reaming, leave from 0.005 in. to 0.010 in. of stock to be removed by a Rose reamer.



*Figure 3.18. Reaming a hole on the lathe, using ample cutting fluid.  
(Courtesy of DoAll.)*

2. Although almost any reamer will do an acceptable job in providing a fine finish, a Rose reamer is usually selected for finishing. A Rose reamer cuts primarily on its cutting edges set at an angle of 45 degrees and provides the best results when removing about 0.005 in. of material.
3. Start the machine and run it at a slow speed—about half of the RPMs indicated for a drill of the same diameter in that material. (See Chapter 7 for conversion of speeds in surface feet per minute into RPMs.)
4. After the reamer has been mounted into the tailstock, either in a drill chuck or, if it has a taper shank, directly into the tailstock holder, feed it into the work slowly and with plenty of lubrication.
5. Run it through the work, then remove it with the chuck still running in a clockwise direction. **Do not reverse the motion of the chuck.** If the chuck is reversed, this can damage the cutting edge of the reamer.
6. If reaming a hole that has been drilled first, use a drill about 0.015 in. undersized, then ream the hole with a standard reamer to impart the final size. A right-hand spiral reamer helps remove the material and will impart a nice finish to the hole. Use plenty of lubrication, and again, **do not reverse the spindle.**

## Counterboring on a Lathe

Counterboring is a method used to enlarge an existing hole to permit it to accept either a screw or a pin that has the diameter of the hole and a head that is larger. In the latter case, the enlargement permits the head of the screw to be



inserted into the hole below the surface of the work, allowing parts to mate with or pass close to the surface without interference by the screwhead.

Cap screws, sockethead screws, shoulder screws, and pins of various types may be inserted into the hole. The pilot hole and counterbore are  $\frac{1}{64}$ – $\frac{1}{32}$  in. larger than the screw fitting into the recess.

There are three types of counterbores: (1) the pilot and counterbore are in one piece—generally used for standard size screws and heads; (2) counterbores with replaceable pilots—used for unusual sizes; (3) a bar that is the standard size of the screw and has replaceable blades to use for unusual sizes of heads.

The procedure for Counterboring is as follows.

1. Select the counterbore to be used.
2. Drill the pilot hole. This should be the size of the screw plus  $\frac{1}{64}$  in.
3. Insert the counterbore in the pilot hole and using cutting fluid, drill to a depth of  $\frac{1}{64}$  in. deeper than the length of the head. The cutting speed is the same as for any other drill of that diameter.

## Spotfacing

A spotface is similar in appearance to a counterbore, but the larger portion of the tool is used to smooth a surface upon which a part will be seated. A pilot hole is first drilled to guide the spotface pilot, and the part to be seated on the surface is held to the surface by a screw that fits through the pilot hole.

To spotface, the same basic steps as for counterboring are followed, and a fourth step is added to complete the task. In step 1, select a spotface that has the correct diameter to accept the part that will be seated on the surface.

4. Perform steps 2 and 3 above. In step 3, the cutting speed will be determined by the diameter of the spotface; again, use cutting fluid. Drill to a depth sufficient to clean the surface flat on which the part will be seated, usually referred to as a “boss.”

Drilling, reaming, and boring on a lathe are common but demanding practices and should be approached with care and patience.

## KNURLING ON A LATHE

The term “knurl” is derived from the Middle English word *knorre*, meaning rough stone, knot in wood.

The two meanings are appropriate, since knurling serves two primary functions that, respectively, utilize these characteristics:

- \* *Gripping*. Knurling creates a “rough” surface that ensures a firm grip on small handwheels, thimbles, handles, etc.
- \* *Expanding*. Knurling expands the diameter of the part being knurled slightly by displacing metal and creating, in effect, tiny “knots” that contribute to a tight union with a mating part.

## Types of Knurls Used

For *gripping*, both a properly roughed surface that ensures a good grip and appearance are important, and the diamond knurl typically is used. For applications where *expanding* function is key, then a straight knurl normally is machined.

## Knurling Tools

These are made either with replaceable rollers or with a series of interchangeable revolving heads; both approaches permit selection of coarse, medium, and fine knurling patterns for the diamond knurl and for the straight knurling patterns.

## Knurling Procedure

The procedure for knurling is as follows.

1. After the part has been turned to the required dimension, based on the application, select the appropriate finish—coarse, medium, and fine. The choice of knurling tool normally is determined by the application:
  - a. For *gripping* choose the finish (also, based on the number of raised segments per inch, known as the “pitch”) that provides the combination of appearance and sureness of grip required.
  - b. For *expanding* select the straight knurl finish required. Typically this is done to create a snug press fit for a mounting arbor or similar part either inadvertently machined undersized or rendered so by wear. The greater the expansion required, the coarser the finish needed.
2. Mount the knurling tool firmly in the toolholder. Pressure on the tool will be higher than on a cutting tool; thus it is necessary to achieve greater rigidity in clamping the knurling tool than is required for cutting tools.
3. Center the knurling tool with the work, making sure that the upper and lower knurling surfaces hit the work simultaneously.
4. Spindle speed should be slow—between 100 and 150 RPMs normally is recommended—and feed per revolution should be about 0.025 in.
5. Move the carriage to the end of the piece being turned, and as the spindle is turning, feed the knurling tool into the work. When feeding in, start with about one-eighth to one-fourth of the knurl overlapping the work and touch the part lightly to make sure that both knurling segments are turning. Once the tool has been properly set, feed the tool into the work about  $\frac{1}{64}$  (0.064) in.
6. At the same time, start the longitudinal feed so that the knurling tool will travel across the work. Lubrication is important, so apply cutting oil as the part is being machined.
7. Stop the lathe when the knurling tool reaches the end of the part, **but do not allow the knurl to run off the end of the workpiece**. It is vital to stop the lathe and not just the longitudinal travel, since that would result in a double cut that will spoil the appearance of the finished part.

Moreover, since synchronization will have been lost, the double cut will occur across the entire face of the piece when the lathe is reversed for the return pass.

8. Before starting the lathe again, reverse the travel. Now start the lathe and, at the same time, feed the knurling tool into the work another 0.025 in. with the cross-feed. The tool is now traveling toward the tail-stock; when it reaches the point where it was initially started, stop the lathe.
9. Reverse the lathe, then start it up and feed it in, repeating this process until the desired knurl is achieved. Use lubrication liberally on all passes.

In general, the straight knurl requires fewer passes than the diamond knurl for a given finish and depth of knurl. In both patterns, the coarse knurl is cut deeper than the medium knurl, and the medium knurl is cut deeper than the fine knurl.

Because the ends of the part are rough when the knurling operation is completed, to finish the job, their edges should be touched lightly with a file to remove burrs and sharp edges.

## FILING AND POLISHING ON A LATHE

When a part has sharp edges, burrs, nicks, and tool marks, files and emery (abrasive) cloths can be used to clean them up for safety and appearance—*unless* the part is designed for a permanent and specific fit. In the latter instance, the part is usually hardened and then cylindrically ground.

### Removing Burrs and Sharp Edges

It should be standard practice to routinely use files to remove burrs and sharp edges that can cause scratches and deep cuts to anyone handling the part following machining. Normally, no more than about 0.002 in. to file can keep the part straight if the length is a consideration.

### File Toothforms

Files used for turning come in several toothforms, including single, double, and curved. The single-cut file has a single row of teeth parallel to each other at an angle that runs straight across the face of the file, while a double cut file has two rows of teeth crossing each other at an angle.

The *single cut file* will remove stock faster because it is a coarser file. Files come in a variety of types that vary in the finish they impart: coarse; bastard; second cut; and mill (smooth).

### File Length

The length of the file is measured from the base of the file to its end; the tang is never included in the file length. Because the tang comes to a fine point, it can

injure the user if the tang is not mounted in a wooden handle when filing. **Always mount the handle on the file!**

## Polishing Cloths

These come in either aluminum oxide or silicon carbide, and vary in the size of the gains (grit) used. Relative coarseness or fineness of the grit determines the finish, so obviously, the finer the finish desired, the finer the grit that must be used.

## Spindle Speeds

The spindle speed must be slow when using a file to remove edges, burrs, and tool marks, whereas it can be quite high when polishing with an abrasive cloth. Stock removal during polishing with an abrasive cloth ranges from about 0.0002 to 0.0005 in. or only about one-tenth of the amount removed during filing.

## Filing Procedure

The procedure for filing is as follows.

1. Hold the wooden handle in the right hand and grasp the opposite end between the thumb and forefinger of the left hand. (*The reverse of that—holding the handle in the left hand and grasping the other end between the thumb and forefinger of the right hand—is often recommended, especially for beginners, since it reduces the likelihood of accidentally being hit on the hand by the revolving chuck while removing stock. If the piece is long and supported by centers, either method of holding the file is satisfactory.*)
2. For ferrous metals, set the spindle speed at about 100 RPMs. When filing nonferrous metals, the spindle speed can be increased fourfold.
3. First file the sharp corners smooth so there is no danger of injury. Hold the file exactly as described in step 1 and lightly exert pressure to the edge, moving the file forward while removing stock. Repeat until the corners are smooth and rounded satisfactorily.
4. Next, start at the end closest to the spindle and hold the file as recommended in step 1, with tip held slightly to the right.
5. Turn the machine on, and applying light pressure, guide the file to the right, making the upcut and then the downcut. Note that when using a double-cut file, the upcut is usually deeper than the second or downcut. After each complete movement of the file, lift it up and, starting at the point just finished, continue until the end of the workpiece or of the portion to be filed is reached.
6. Take long strokes and, after each complete filing operation, check the size with a micrometer. Continue until the required finish and size are achieved.
7. When filing a workpiece mounted between centers, use a live center in the tailstock to prevent burning the work. This precaution is necessary

because pressure from filing is transferred to the center and if this is even moderately excessive from time to time, it can result in scoring or burning damage to the dead center and the part.

## Procedure for Polishing

The procedure is as follows.

1. Set the highest RPM on the machine.
2. Select the grit size of the abrasive polishing cloth for the finish required or desired. Grit sizes vary from 80 to very, very fine; normally, any grit size from about 180 to 240 will result in a good finish. Use a strip of abrasive cloth about  $1\frac{1}{4}$  in. wide and about 10 to 12 in. long.
3. Place the cloth around the part, holding one end of the cloth in the right hand. Start the lathe and, grasping the other end of the cloth with the left hand, draw it across the workpiece surface as the part rotates. Apply pressure while doing this; normally, no more than 0.0002–0.0005 in. of stock is removed during the operation. Friction will cause the part to become quite warm, so it is best to avoid touching the part with bare hands until the operation has been completed and the part has cooled off.
4. Extra force can be applied during the polishing operation by wrapping the abrasive cloth around a file and applying it in this manner to the workpiece. This method requires a deft touch plus skill and practice, so it should not be attempted on “money” work until the technique has been perfected on practice stock.

Both filing and polishing are “finishing” operations, which means the sequence of use should begin with the coarser files and abrasive grits and progress to the finest files and grits selected for the finishing step.

## GRINDING TOOLBITS

The grinding of toolbits (Figure 3.19) has been an art since the first lathe was developed. Generally speaking, toolbit grinding in the shop has largely meant grinding high-speed steel toolbits.

During World War II and after, however, when brazed carbide became popular, it became necessary to grind carbide. Because of this hard new material, toolbit grinding took on a new form and carbide toolbit grinding helped develop the diamond industry.

Inserted carbides, cast alloys, and ceramic tools, by virtue of their greater rates of stock removal coupled with the ease of changing tools on modern machines, are rapidly taking over many of the applications formerly handled by high-speed steel tools. Because of this development, the grinding of high-speed toolbits is on the wane; however, a considerable amount is still being done in the shop and thus will be covered here.

It is helpful to know the nomenclature of tools for most efficient use of them as well as grinding them, so tool nomenclature will be briefly reviewed here. In



*Figure 3.19. Array of carbide tool inserts and toolholders, high-speed toolbits, center drills, and brazed carbide toolbits. Top row: carbide toolholders; second row: cut-off tools and cast toolbit; third row: center drills and high-speed toolbits; right side: cemented brazed carbide toolbits; bottom row: carbide inserts, ceramic insert. (Courtesy of DoAll.)*

in addition to reviewing procedures for grinding high-speed steel tools, the grinding of carbide and cast alloy tools will also be described.

Tool geometry is important because it gives the cutting tool the clearances necessary for removal of metal, provides “beef” behind cutting edges for heavy chip-removal capabilities, and allows imparting special shapes as needed.

The geometry is governed to a great extent by the alloy to be machined, by the alloy used in the tool, and by the direction in which the cutting edges will be applied (right hand, left hand). A right-hand toolbit is one whose cutting edges face toward the spindle and, when machining, moves in the direction of the headstock; cutting occurs to the machinist’s left when in the normal operating position. A left-hand toolbit has its cutting edges facing in the direction of the tailstock and, when machining, it moves toward the tailstock; the machining then takes place to the right of the operator.

## Nomenclature

As illustrated in Figure 3.20, the major features or nomenclature of toolbits are as follows.

- \* *Back Rake.* This angle is the slope away from the front of the tool. If it slopes downward, it is a positive rake angle; if it slopes upward, it is a negative rake angle.
- \* *Chipbreaker.* A feature (normally a groove ground parallel to the side cutting edge) that forces the chip as it is taken into a very sharp curl, causing it

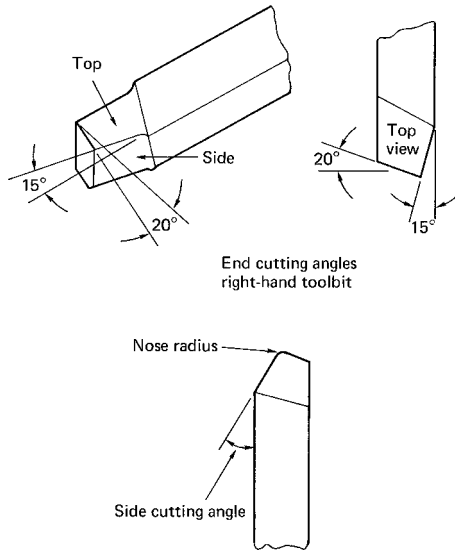


Figure 3.20. Angles to be ground on a high-speed toolbit.

to break rather than assume long, ribbonlike shapes. The chipbreaker serves two purposes, namely, promoting safety and helping keep the work and point of cut clear of chips.

- \* *Side Cutting Edge.* The angle that allows the cutting edge to cut freely. It is the slope from the side cutting edge at the top inward toward the bottom.
- \* *End Cutting Edge.* The angle from the front of the tool starting at its intersection with the side cutting edge and back toward the opposite end. This angle is critical because of the support it gives to the nose of the tool.
- \* *Nose Radius.* This radius on the tip of the cutting edge gives support to the tip itself. The nose radius is always specified when purchasing a carbide insert.

## The Pedestal Grinder

The pedestal grinder—also known as a bench grinder—comes in a variety of types and forms. This section will only review the more or less standard pedestal grinder, used for just about every kind of toolbit grinding.

Pedestal grinders used for grinding high-speed steel tools have two wheels with supports for the tool located approximately at the center of the periphery of the wheel. These supports are adjustable inward and outward, and normally should be positioned within  $\frac{1}{8}$  in. from the wheel. This protects the operator by ensuring that the tool or bit being ground cannot slip down between the wheel and support.

Normally, the wheel to the left of the operator facing the machine is the roughing wheel and has a grit size of 40, whereas the wheel to the right is the finishing wheel and has a finer grit size of 60.

Above the wheels on each side are protective guards, mounted to the body casting, that are called *spark protectors* (since they limit the direction in which sparks can fly during grinding to safe ones). Above the spark guards are safety shields—

usually with lights—mounted on adjustable brackets that further protect the operator. Wheel guards surround both wheels.

## Safety Precautions

When operating the pedestal grinder, the following safety precautions should be taken.

1. Always wear safety glasses when operating the machine.
2. Stand to one side of the machine, out of the planes of wheel rotation, when turning on the grinder. **Let the machine run for a minute or so before using.** The reason for this is that if a wheel is damaged, it will explode toward the front since it is contained in all but the area open for grinding in the front. While this is extremely rare and unlikely to happen, the safe course is to take the proper precautions.
3. Check a new wheel for damage before mounting it on the machine by tapping it *lightly* with a metal rod. The resulting ring should be clear; if the ring is muted and dull, discard and **do not use!**
4. The wheels must have blotters on each side. If one or both do not, get blotters for both sides or get a different wheel. **Do not use without a blotter on each side!**
5. Mount the wheel on the spindle. It should fit perfectly. If it is loose, get another wheel, and *never* try to force one on. **If a wheel does not fit properly, do not use it!**
6. After mounting the wheel on the spindle, snug the nut so that it holds the wheel properly. **Do not use excessive force to tighten the nut, because this can crack the wheel.** The nuts are left- and right-handed so that they turn counter to the direction of spindle rotation and thus tighten rather than loosen when the grinder is running.
7. Make sure the diameter of the wheel is the correct size for the grinder being used. If a wheel too large for the machine is mounted, the spindle speed will be too fast for the wheel. All wheels specify the maximum safe speeds. **If the safe speed for a wheel is exceeded, the wheel may explode.** If the wheel is too small in diameter, grinding will be slow and inefficient but it will not pose a safety hazard.
8. The pedestal grinder for high-speed steel grinds only on the periphery of the wheel. **Do not grind on the side of the wheel.**

The pedestal grinder for carbide tools differs from that for high-speed steel tools. It is made to grind on the side of the wheel. The wheels used are usually diamond wheels and should be used wet. In the past, a silicon carbide wheel was popular and it is still used on occasion, but it is not, however, very efficient or economical in comparison with diamond wheels.

The supporting-tools rests can be adjusted for the correct angle. All rake, end, and relief angles for carbide grinding are critical and must be ground precisely.

The grinding wheels can be revolved clockwise or counterclockwise to allow for ease of removing stock when holding the tool at its proper angle.

The carbide pedestal grinder has a much heavier design than the pedestal grinder for high-speed steel tools. The rigidity of the grinder allows for greater precision.



Grinding of cast alloys can be done on the pedestal grinder used for high speed. Although water is used to cool off high-speed steel tools as grinding is done, it should not be used when grinding cast alloy toolbits. They should be air-cooled; water will cause too much shock and, as a result, the tool will either break or develop cracks.

High-speed toolbits come in various alloys ranging from relatively soft alloys in the molybdenum range to very-difficult-to-machine alloys in the tungsten range.

## Grinding High-Speed Steel Bits

To grind high-speed steel tools, use the following procedure.

1. Adjust the work rests to within  $\frac{1}{8}$  in. of each grinding wheel. Start the grinding with the roughing wheel located on the left side of the grinder.
2. Make sure the water trough is filled with water so the tool can be dipped into it from time to time to keep it cool. If the machine does not have a trough, use a separate container.
3. Hold the tool firmly with the right hand and support it with the left. Grind the front angle and the slope at the same time. The commercial toolbits are already sloped back. This part can be touched lightly with the grinder.
4. The novice should hold the tool against the wheel lightly and keep it there while grinding until the section is completely ground. This will give a nice, clean finish to the tool. With greater experience, it is preferable to keep moving the tool back and forth across the wheel to prevent wearing it unevenly.
5. Next, grind the side cutting edge angle and the side relief angle at the same time. Do not apply too much pressure against the wheel. Excessive pressure can burn the tool and destroy its hardness.
6. The angles ground will be determined by the material to be machined. Materials are reviewed in Chapter 7.
7. When finished with the rough grinding, repeat the procedure using the finishing wheel. Not much stock should remain to remove, so all grinding on the finish *wheel* should be done with a very light touch.
8. Finally, grind a  $\frac{1}{32}$ -in. radius on the nose to give strength to the tip.

When grinding cast alloys, use the same procedure but do not use any water to cool the tool. If the tool must be cooled let it cool in the air. This type of material can get red hot but it will not anneal when air-cooled. If dipped in water when hot, it will crack and often break.

## Grinding Brazed Carbide

The following procedure should be used to grind a brazed carbide toolbit.

1. Use a carbide pedestal grinder. Use the diamond wheels recommended for roughing and finishing. Apply coolant freely to the wheel while grinding.

2. Set the work rest at the recommended angle for the end cutting edge.
3. The end relief angle determines the cutting edge. Note that when the tool is placed on the work rest, the end relief angle will be facing away from the operator for a right-hand tool on the finishing wheel. On the roughing wheel, it will face toward the operator. This means the wheel must rotate counterclockwise to complete the grinding. Turn off the machine, then reverse the switch to obtain the counterclockwise rotation. Grind as above.
4. Next, grind the back rake angle as required by the material to be machined. Remember to set the correct work-rest angle.
5. Finally, grinding “free-hand,” grind a  $\frac{1}{32}$ -in. radius on the nose to give strength to the tip.
6. For finish grinding, repeat the above steps, using the grinding wheel to the right after making sure it has the finer grit wheel mounted. Diamond grinding wheels are usually identified by color to make it easier to select the correct wheel.

## FORM TOOL TURNING

### Turning Cutoff and Form Tools

Turning a part includes turning a shape on that part with a tool that has the shape already formed in it. Forming the shape requires skill on the part of the machinist and requires careful selection of speeds and feeds to impart the shape to the part itself. For production runs, the formed tool may not be needed because a machine controlled by an appropriate computer program can easily generate the tool shape required. The only requirement in connection with the tool bit is it must be narrow enough to be manipulated without hitting the back of the tool. Therefore, the speeds shown below are only for high-speed tools formed by the machinist.

Cutting off of a part on the lathe is a common practice both for production and for individual pieces; it can be used to produce a single part or for the first part made by the machinist for a possible production job. The first tool for cutting off may be made from a high-speed tool bit. While a cut-off tool can simply be bought, if only one or two pieces are needed, it doesn't pay to buy one.

There are many good commercial cut-off tools available and these should be for production runs. Despite higher tool cost, the versions providing the most economical results for production are carbide. The speeds and feeds shown below are for high speeds tool bits.

The speeds and feeds shown here are only examples of speeds and feeds used and are helpful as starting points when considering a job. Tool manufacturers typically provide recommended speeds and feeds for various materials when using their products, and, of course, these should be utilized.

## Feed IPM

Material	Sae	Speed	Cutoff Tool Width							
			.062 .125 .1250			Form Tool Width				
			.500	.750	1.00	1.50	2.00			
Carbon Steels	1008-	500	.001	.001	.002	.0015	.0012	.001	.001	.0007
	1095	255	.001	.001	.001	.0013	.0011	.001	.001	.0008
Free Mach Steels	1108-	400	.001	.002	.002	.002	.0018	.002	.001	.0012
	1213	550	.003	.003	.004	.003	.0027	.002	.002	.0018
Manganese Steels	1330-	305	.001	.002	.002	.0015	.0016	.001	.001	.0011
	1345	250	.001	.002	.002	.0018	.0015	.001	.001	.001
Nickel Steels	2320-	280	.001	.002	.003	.0025	.0018	.001	.002	.001
	2330	150	.001	.002	.002	.0021	.0016	.001	.001	.0008
Nickel Chrome St.	3115-	250	.001	.002	.002	.0023	.0017	.001	.002	.0011
	3315	150	.001	.002	.002	.0021	.0016	.001	.001	.0008
Moly Steels	4017-	325	.001	.002	.002	.0018	.0016	.001	.001	.0011
	4047	250	.001	.002	.002	.0017	.0015	.001	.001	.001
Chrome Moly Steels	4130-	250	.001	.002	.002	.0017	.0015	.001	.001	.0011
	4140	250	.001	.002	.002	.0017	.0015	.001	.001	.001
Nickel Chrome Moly Steels	4320-	325	.001	.002	.002	.0018	.0016	.001	.001	.0011
	4340	305	.001	.002	.002	.0018	.0016	.001	.001	.0011
Nickel Moly Steels	4615-	325	.001	.002	.002	.0017	.0015	.001	.001	.001
	4626	305	.001	.002	.002	.0018	.0016	.001	.001	.0011
Chrome Steels	5015-	325	.001	.002	.002	.0017	.0015	.001	.001	.001
	5160	225	.001	.002	.002	.0017	.0015	.001	.001	.001
Chrome Steels	50100	275	.001	.002	.002	.0018	.0016	.001	.001	.0011
	52100	275	.001	.002	.02	.0018	.0016	.001	.001	.0011
Chrome Vanadium	6118-	325	.001	.002	.002	.0018	.0016	.001	.001	.0011
	6150	290	.001	.002	.002	.0016	.0014	.001	.001	.001
Nickel Chrome Moly Steels	8617-	325	.001	.002	.002	.0018	.0016	.001	.001	.0011
	8742	250	.001	.002	.001	.002	.0017	.001	.001	.001
Silicon Steels	9255-	225	.001	.002	.002	.0017	.0015	.001	.001	.001
	9535	175	.001	.002	.002	.0023	.0018	.001	.001	.001
Hi Speed Tool Steels	T4 T8	175	.001	.001	.002	.0015	.0013	.001	.001	.0008
	M7-46	SAME								
Hi Speed Tool Steels	T15 M	150	.001	.001	.002	.0015	.0013	.001	.001	.0008
	M3-2	SAME								
Tool & Die Steels	A7 D2	150	.001	.001	.001	.0011	.001	.001	.001	.0007
	D3 D7									
Tool & Die Steels	O1 O2	205	.001	.001	.001	.0013	.001	.001	.001	.0007
	H12	255	.001	.001	.001	.0014	.001	.001	.001	.0008
Stainless Steels	201-	275	.001	.002	.002	.002	.0016	.001	.001	.001
	202-	SAME								
	302-	SAME								
	304-	SAME								
	303	325	.001	.002	.002	.0025	.002	.002	.001	.001

*Form Tool Cutoff Speeds And Feeds*

## Lathe Chatter

Chatter and vibration in machining are not always problems. Some machinists even want vibration, adding it intentionally to break chips. Internal grinding operators sometimes like to have a little chatter in rough grinding to make the grinding wheel break down and do a better job of cutting. By itself, chatter is seldom a serious problem, but it can cause other problems. If it is high frequency chatter, it may produce irritating noise. It can leave marks in the workpiece so deep that the finish will be out of tolerance, but more often it produces a shallow pattern that that may be unacceptable “because it doesn’t look good.” And when using carbide tools, chatter can cause serious reduction in tool life.

Here are things to try when encountering chatter while turning:

- \* Lower the stock removal per pass.
- \* Try a heavier feed if presently turning below 0.10”
- \* Reduce width of cut when using a form tool.
- \* On slender bars use a steady rest.
- \* Use a smaller nose radius on the cutting tool to keep cutting forces away from the radial direction.
- \* Modify side cutting edge angle to be more radial—keep force away from radial direction.

When boring try the following:

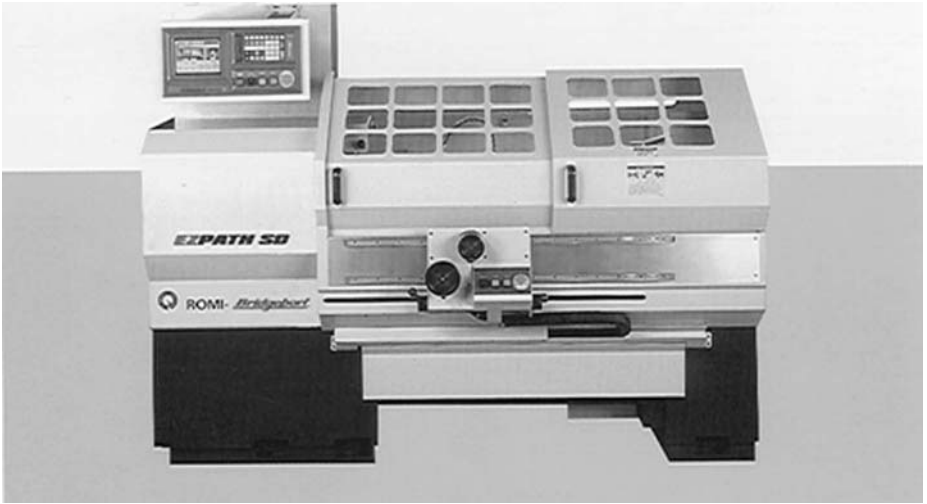
- \* Use the largest bar diameter possible.
- \* Use the shortest bar length possible.
- \* Make the tool nose radius as small as possible to minimize radial force.
- \* Make the side cutting edge angle as radial as possible to reduce radial force.
- \* Reduce stock removal in roughing.
- \* If possible consider a carbide boring bar.
- \* For final boring problems, consider reaming after a semi-finish bore.
- \* For roughing, consider multi-tooth bars.

## The Turning Centers

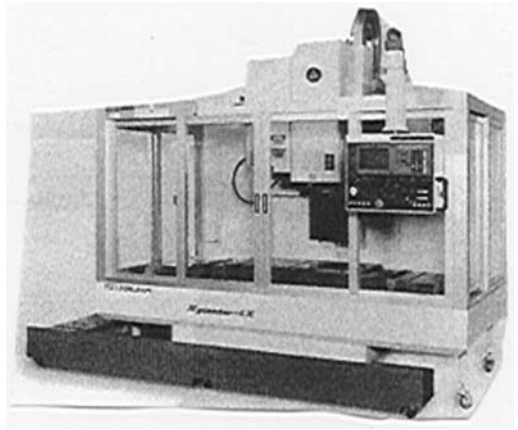
From the basic lathes with all of the attachments just shown evolved the turning centers with the advent of the computer.

CNC lathes like the Bridgeport shown greatly simplify turning operations and sharply boost productivity. But make no mistake. Take away the computers, strip away the covers of the machine and you will see the basic lathe under it all. This is not to deprecate or devalue these sophisticated turning centers but rather to show how a sound grasp of the basic machine is essential to understanding both its evolution to today’s high levels of output and accuracy and to its potential for further refinement and development.

With added features, controls, and expanded capabilities, computer-controlled lathes earn the designation of turning centers, either vertical or horizontal,



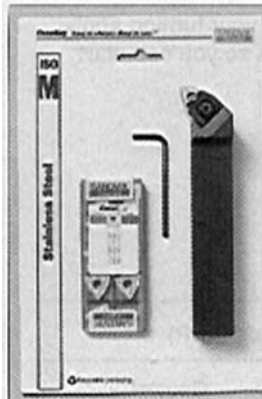
*Bridgeport Ezpath CNC Lathe. (Courtesy of Bridgeport)*



*Pictured are, left, a horizontal machining center, and, right, a vertical machining center. (Photos courtesy of Kitamura Machinery of USA, Inc.)*

depending upon whether the spindle is mounted vertically or horizontally, as depicted in the following photos.

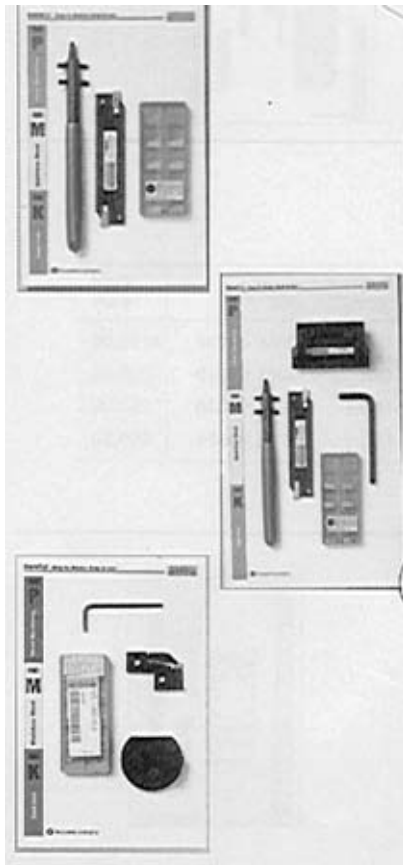
What has been added to create a machining center is a highly rigid machine to handle higher horsepower, ball screws or linear guideways to increase accuracy, an automatic system to control a cutting fluid system, a chip removal system, and a computer to control the action of the machine. In addition many versions of



*Stainless Steel Machining Kit*



*Cast Iron Machining Kit*



*Cut-off Kits. These kits simplify parting and cut-off operations. Once the work-piece material is known, the tool is selected for it and the operation: parting of a bar or tube, internal or face-grooving, etc. The final step is selection of the grade, based upon whether conditions are good, average, or difficult. (Photos of kits courtesy of DoALL AND SANDVIK)*

these machines will have a parts catcher, a bar feeder, a bar puller, programmable tailstock, a tool pre-setter and multiple automatic tool positions.

Underneath it all, however, the turning or machining center is still basically a lathe. The purpose of the design is to increase production and hold down cost by reducing part handling. Prior to the advent of the computer, the highest cost of turning and other functions now handled on the turning center was the cost associated with the operator loading and unloading the part, changing the various tools for making the part, cleaning the parts and the machine, and allowing for rest periods. Today most of that has been minimized.

The computer becomes the driving force. It not only can direct all of the functions cited above but, with appropriate software, it can readily estimate the cost of producing a part. Nevertheless, the basic principles still apply, including how fast a part can be machined in any given material and configuration and how much will it cost to produce that part.

Rules of thumb still provide beginning points for machining various materials. A mild carbon steel, for instance, can be machined at the rate of 1 cubic inch a minute for each horsepower on a machine. This of course means the greater the horsepower the greater amount of stock that can be removed per pass and the more rigid the machine must be. Using a program such the one for the cutoff saw [see chapter 9] and substituting the lathe for the saw and its cubic cutoff rate indicates how much each pass of the turning tool is needed to machine a particular part.

If, for example, three passes are necessary, to determine the cost per cut for each pass, take the cost of the machine, the cost of the tool, coolant, the cycle of machine movement, etc., into account. This approach will readily determine productivity. Tool manufacturers offer basic kits designed to facilitate specific machining operations.

With as many as 12 tool positions, some of the examples of accessories provided can automatically be used when turning a piece. With the proper speeds and feeds applied, whether cutters are high speed tools or carbide, costs can be determined.

### Project Number 1: T-Tap Wrench

#### INTRODUCTORY INFORMATION:

The T-tap wrench or socket wrench is essential to get into hard-to-tap places. It is not as sensitive as the regular tap wrench and the handle can be moved back and forth when tapping close to a shoulder on the work.

#### STOCK:

Cold rolled steel

#### SAFETY AND PRECAUTIONS:

1. Always make sure that drills are sharp when using.
2. Observe all shop safety rules.

## Item 3 - Handle

STOCK: C.R.S.

## TOOLS:

6" Steel rule	Emery cloth
O.D. Caliper	Oil
File	

## PROCEDURES OR STEPS:

1. Secure 5/16" diameter X 4-5/8" long, C.R. steel.
2. Chuck in lathe collet. Turn both ends to 3/16" diameter X 3/8" long.
3. Groove center of handle with round nose tool X 1/4" wide.
4. Polish all over. (Polish enough for handle to fit 5/16" reamed hole.)

## Item 4 - Handle Ball

## STOCK:

C. R. S.

## TOOLS:

6" Steel rule	Countersink	Oil
File	3/16" Drill	Ball peen hammer
	Emery cloth	

## PROCEDURES OR STEPS:

1. Secure 3/8" diameter C.R. steel (2 required).
2. Place in lathe collet, machine and file 3/16" radius.
3. Drill 3/16" hole X 1/4" deep.
4. Countersink edge of hole.
5. Cut-off to 5/16" length.
6. Polish with emery cloth.
7. Rivet to handle.

## Item 5 - Body

## STOCK:

C.R.S.

## TOOLS:

6" Steel rule	17/32" Drill
Center drill	Thread center gage



1/4" Drill	1/8" Drill
Vee block	File
1" Micrometer	Emery cloth and oil

## PROCEDURES OR STEPS:

1. Secure 13/16" diameter X 3-5/16" long.
2. Place in lathe collet, face one end and crown other end.
3. Center drill 13/16" diameter.
4. Drill 1/4" X 1-3/8" deep hole.
5. Reverse in collet, turn .748" diameter and 19/32" diameter.
6. Drill 17/32" X 1-1/4" deep hole.
7. Thread in lathe 3/4" -NF X 3/4" long.
8. Drill 1/8" cross hole as per drawing.
9. File and polish surfaces.

Item 7 - Collet

## STOCK:

C.R.S.

## TOOLS:

I. D. Caliper	1/2" Drill
6" Steel rule	Forming reamer
11/16" Drill	File
Center drill	Emery cloth and oil
3/4"-16 NF Tapset	O. D. Caliper

## PROCEDURES OR STEPS:

1. Secure 15/16" diameter X approximately 4" long.
2. Chuck in lathe collet, face end.
3. Center one end, support work with tail center.
4. Medium knurl as per drawing.
5. Drill 5/8" hole X 1-3/8" deep.
6. Bore hole to 11/16" X 1-3/8" deep.
7. Tap hole 3/4" - 16 NF X 1" deep in machine.
8. Drill through with 1/2", drill, approximately 2" deep from edge.
9. Use forming reamer for 30° inside bevel of hole.
10. Bevel out side diameter, nose 11/16" diameter at small end X 30° taper.
11. File and polish.
12. Cut-off to 1-7/8" length.

## Item 8 - Chuck

## STOCK:

C.R.S.

## TOOLS:

6" Steel rule	3/16" Drill	1/16" Mill saw
Center drill	File	3/16" Mill saw
O. D. Caliper	Emery cloth	Oil

## PROCEDURES OR STEPS:

1. Use 5/8" diameter X approximately 4" long.
2. Place in collet, center drill small hole, support with center.
3. Turn 1/2" diameter X 1-7/8" long.
4. Cut-off 2-3/8" length.
5. Place in 1/2" collet, center drill.
6. Drill hole A as per drawing X 7/16" deep.
7. Bevel end 1/2" diameter X 14°.
8. File and polish.
9. Place in mill dividing head chuck, saw the 1/16" slots X 1-3/4" long.
10. Saw slot in other end 3/16" wide X 1/4" deep.

## Chapter 4

# THE MILLING MACHINE

The formal definition of the generic milling machine runs along these lines: a *generalized* machine tool that employs rotary multiple-toothed cutters to remove metal from a workpiece secured to a table whose motion relative to the cutting tool is controlled by a combination of vertical and horizontal feeds.

A less formal description of the milling machine pegs it as the only machine tool capable of reproducing itself without the aid of any other machine tool or specialized machine. (This is true only of horizontal mills with universal tables that permit them to cut gears, worms, etc.)

## HISTORY

Eli Whitney developed the milling machine in the early 1800s. The exact year he began using his revolutionary new machine is somewhat uncertain, but since the first recorded reference to it dates back to 1818, historians use that year as an acceptable approximation.

Whitney's machine is both the oldest milling machine known to possess all of the basic features needed to earn the title of "machine tool" and the most significant in its influence on the design of later milling machines. (See *Studies in the History of Machine Tools*, Robert S. Woodbury, M.I.T. Press, Cambridge, MA, 1972.)

Virtually all important elements of design found in the modern milling machine were developed in the United States. Builders were initially all located in New England, but these new machines were also later produced and refined in Cincinnati and other cities in the Midwest. Beginning with the model Whitney invented, early milling machines were used to manufacture firearms, clocks, steam engines, farm equipment, and, following their commercial introduction at mid-century, sewing machines.

American-made milling machines and pirated copies—some so slavishly duplicated that they carried the name of the manufacturer of the model they imitated—were used in metalworking operations in Great Britain and in the United States, but did not come into general use in Europe until the 1890s.

But even in the United States during the latter part of the nineteenth century, only specialized shops had any kind of milling machine; today, the smallest shop—even the “home machinist”—finds some sort of milling machine necessary, and this useful machine and its many spin-offs have evolved to the point where they are indispensable to metalworking and now routinely perform metal-cutting operations previously difficult or impossible to do.

## BASIC TYPES

The milling machine comes in two basic types:

1. *Vertical*. The cutting tool is normally inserted into a mounting device on the spindle and is held vertically.
2. *Horizontal*. The cutting tool is either mounted directly into the spindle and held horizontally, or it is mounted on an arbor supported and rotated by the spindle at one end and supported at the other end by a detachable device.

Both machines have similar operating controls, features, and metal-removal characteristics, but the horizontal machine typically is structurally much heavier and more powerful than its vertical counterpart. For its part, the vertical mill is more versatile and costs less, with basic, yet highly attractive, characteristics that help account for the fact it is much more popular than its muscular relative.

The milling machine is second only to the lathe in utility and usefulness for toolroom and machine-shop applications.

Project No. 2, located at the end of this chapter, is for machining a wheel puller. The step-by-step procedure is given, along with drawings of each component.

## BASIC FUNCTIONS

The primary functions of the milling machine are: (1) to machine slots and (2) to face materials (create flat, smooth surfaces).

In their most rudimentary forms, these functions are linear in the sense that any given point on the centerline of the cutting tool traces a straight line as the tool progresses through the work. By the use of attachments, however, the tool can be manipulated so that a given point on the tool traces a complicated path, creating, in the process, such complex forms as: circular, elliptical, concave, or convex shapes; and arcs, helix angles, hexagons, or other multifaceted geometry's.

These capabilities readily translate into drills, reamers, and a host of special cutting tools as well as gears, worms, rack-and-pinions, housings, shafts, and similar components, including all of the parts needed to reproduce the milling machine itself.

In brief, the milling machine is an extremely versatile power tool.

## THE VERTICAL MILLING MACHINE

The basic vertical milling machine has a characteristic, readily recognizable shape (Figure 4.1). Its basic components are the milling head, ram, base and column, knee, and the table.

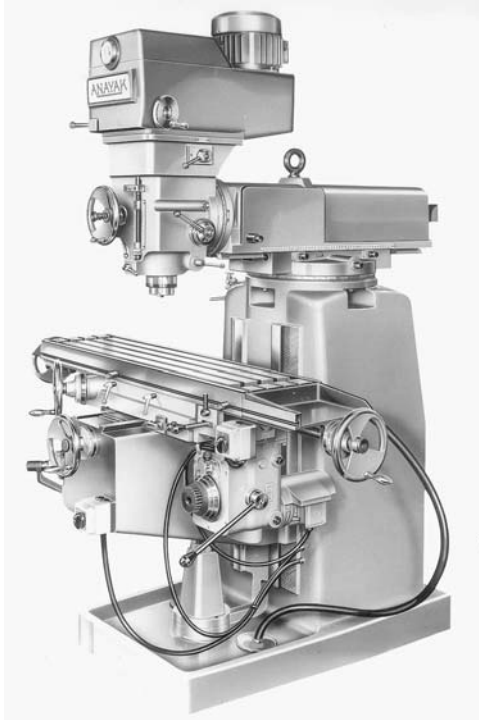


Figure 4.1. Vertical milling machine. (Courtesy of DoAll.)

Major features of these basic components include:

1. The *milling head*, a self-contained unit (Figure 4.2). It houses the spindle drive and motor, belts and pulleys or gears, quill and quill housing, etc., needed to power the cutting tool. Mounted on a movable ram, it has the following features: (a) The head normally is very accurately aligned so that a mounted cutter will be exactly perpendicular to the table, but it can be set at an angle for special machining steps. (b) Spindle speeds are set and changed by either a variable speed pulley or, if the machine does not have that feature, by selecting the correct diameter pulley and manually shifting the drive belt to it. (c) On both of the above types, there is a choice of three downfeed speeds that are coordinated with the spindle speed for drilling or boring holes. If desired, feed can be manipulated by hand. (d) The milling head can be tilted up to 45 degrees to the right or left or to the front or the back. On larger mills, the head may tilt only to the right and left, and an aligning pin may be used to help ensure proper vertical positioning.



*Figure 4.2. Milling machine head showing the angles it can be tilted, exhibiting its independence of movement. (Courtesy of DoAll.)*

2. The *ram* is mounted on the column of the machine and both rotate in a full circle and, guided by accurately scraped dovetails, slides back and forth. This allows the head to be positioned anywhere within the 360-degree arc; this provides access to all portions of the table. To reposition the head, the operator loosens the bolts locking the ram in place, moves the ram until the head is in the new position, then tightens the bolts once more.
3. The *column* is a heavy-duty casting that carries not only the ram but also the knee, a heavy casting that projects at a 90-degree angle from the column. The lower portion of the column forms the base of the machine that, in addition to supporting the machine, normally contains a reservoir for the recirculating coolant system. The knee, accurately positioned by means of an acme screw rotated either under power or manually with a handle, slides up and down the column on handscraped ways. The screw, hardened and ground for accuracy and long useful life under heavy-duty use and equipped with dials divided into increments of 0.001 in., moves the knees up and down 0.100 in. per revolution.
4. A *table* capable of moving from side to side (longitudinally or along the *X* axis) or in and out (traverse or “cross-feed”—along the *Y* axis) is mounted to the knee. The vertical movement up or down (toward or away from the spindle) of the knee provides the third or *Z* axis, permitting a workpiece mounted on the table to be positioned anywhere within the three-dimensional “working space” under cutting tools mounted in the spindle quill.

Table movement is effected by acme screws actuated by handwheels equipped with dials graduated in 0.001-in. increments; one full revolution of a handwheel moves the table 0.250 in. Movement longitudinally (*X* axis) is accomplished by

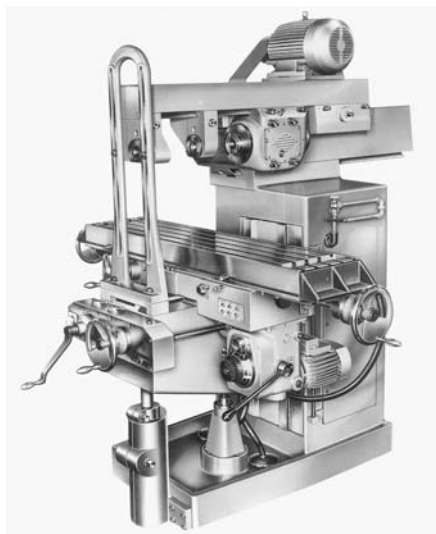
handles, one at each end of the table. Cross-feed ( $Y$  axis) is performed using a handwheel mounted on the front of the table.

On most milling machines, feed along all three axes— $X$ ,  $Y$ ,  $Z$ —can be accomplished under power controlled by a motor actuating each axis. When the machine is operated under power, the rate of feed must be coordinated with the speed or rate of rotation of the cutting tool driven by the spindle.

If desired, each of the table and knee acme screws can be firmly locked to preclude unwanted movement along that axis.

## THE HORIZONTAL MILLING MACHINE

The horizontal milling machine (Figure 4.3) has basically the same components and controls as the vertical milling machine, but with a few differences. Ram and casting construction are similar to those of the vertical mill, and table ( $X$ ,  $Y$ , and  $Z$  axes) and knee travel are the same. The spindle, however, is mounted in the column and other than the rotary turning motion it imparts to the cutting tool or to arbors mounted in the spindle, it does not feature any motion.



*Figure 4.3. Horizontal milling machine. (Courtesy of DoAll.)*

The ram is used to support an arbor inserted in the spindle upon which a cutter is mounted. Other cutters can be mounted directly in the spindle when the arbor and ram are not utilized. Feed into the tool is always controlled by table movement. With the massive support provided to the ram by the basic design of the horizontal milling machine, greater horsepower can be used than is feasible on vertical mills of similar size, making possible higher rates of metal removal.

In addition to conventional milling operations, gear cutting, machining of complicated helix angles, and production of special tools are readily performed on the horizontal milling machine.

Horizontal milling machines come in various sizes and horsepower capabilities to accommodate varying shop requirements. If the table of a milling machine is fixed (i.e., it cannot swivel back and forth at angles to the longitudinal axis, the machine is a plain milling machine. If the table can swivel, it is known as a universal milling machine; the latter is required when helix angles are to be machined.

Drills, reamers, boring tools, grinding bits, and similar cutting tools used on other machines can be used on milling machines, but the cutting tools unique to milling machines include dovetail, shell, T-slot, double-ended, inserted carbide, and shell end mills. Plain milling cutters, side milling cutters, Woodruff cutters, and a host of other cutters also are widely used on milling machines. These are covered in greater detail in the following sections on cutting tools.

## MILLING MACHINE CUTTING TOOLS

The great variety of milling cutters available permit the milling machine to perform an extremely broad range of machining steps and functions, making it by far the most versatile of all of the basic machine tools for the kinds of things it can do.

As is true of every machine tool, a milling machine without cutters is useless. Moreover, machines are designed to take advantage of specific capabilities of the cutting tools it uses—that is, the machine is designed around the cutting tools and not the other way around.

Milling machines pre-dating the introduction of high-speed steel cutting tools, for example, feature relatively low spindle speeds because the carbon steel cutting tools then available could not withstand the heat and abrasion associated with high speeds and feeds.

Once high-speed steel tools were introduced, however, more powerful machines were designed and built to take full advantage of all of the improved heat resistance and retention of the new cutting edges under higher friction and application of power they provided.

This basic approach permits realization of the maximum stock removal rates specific types of cutting tools provide, and applies to other types of cutting tool materials and designs. With the advent of carbides and coated cutters, greater horsepower and rigidity are required to permit achieving the high metal-removal rates they are capable of sustaining.

With ceramics now entering a new stage of development in which more horsepower and numerical control is required, cutting tools are being developed that are more diversified and capable of creating shapes previously difficult or impossible to achieve on a practical machining basis.

## TYPES OF MILLING MACHINE CUTTERS

There is a virtually endless roster of milling machine cutting tools, but by far the most popular and commonly used are end mills and milling cutters.



## End Mills

The end mill is the most popular type of milling machine cutter in use today. End mills are widely used on vertical milling machines in toolrooms and in machine shops, but, in terms of volume of cutting used, machining centers and numerically controlled milling machines probably account for the bulk of profitably expended end mills.

Among the considerations and features that make the end mill so versatile are the virtually infinite variety of diametrical sizes and design characteristics that provide an almost unlimited ability to impart new shapes to metal. Some of the design features include:

1. *Flutes.* End mills have flutes with cutting edges (teeth) on the circumference of the tool body and on its extended end. The flutes on the body are usually helical (spiral) in design, and the spiral can be either right- or left-handed. Although straight flutes can be used, the spiral ones promotes faster, easier chip removal.
2. *Flute Length.* End mill flutes extend from the “business end” of the tool along the flanks of the tool, anywhere from  $\frac{1}{2}$  in. to 2 or 3 in., depending on the diameter of the tool. The flute ends at the portion of the tool that mounts in the machine quill, the shank. Most end mills have straight shanks, but some have shanks with a Morse or a Brown and Sharpe taper; these, however, are not as popular as tools with a straight shank. End mills with a straight shank have a flat milled into them to accommodate a set screw mounted through the toolholder to lock them securely in place.
3. *Number of Flutes.* End mills are available with differing numbers of flutes (Figure 4.4). Smaller end mills typically feature two, three, or four flutes, whereas larger-diameter tools may have six, eight, or more flutes. There are good reasons for the variety:
  - a. Two-flute end mills are used for milling slots. With center-cutting edges, these end mills can “plunge cut,” penetrating into the work pretty much the same as a drill does. In some European countries, these two-flute end mills are called “slot drills” because they permit plunge cutting to depth and milling and slotting with the same tool. Two-flute end mills typically are made slightly undersized because they normally are used for slots and keyways; tolerances run plus zero and minus 0.002 in.
  - b. Four-flute end mills are generally used for finishing operations, and are almost exclusively used for machining on the workpiece peripheries. Although some four-flute end mills have center-cutting capabilities, they are not widely used. Because they typically are used for finishing, these tools are usually made slightly oversized: tolerances typically run minus zero, plus 0.002 in.
  - c. Three-flute end mills are used for plunge milling, and they typically provide a finish nearly as smooth as four-flute end mills. Tolerances typically are the same as for two-flute end mills (plus zero, minus 0.002 in.).



Figure 4.4. Two-flute and four-flute end mills, the most popular used in milling machines.

- d. Larger end mills are available with various numbers of flutes up to six or more, depending on the requirements of the job.

The more specialized types of milling cutters include the T-slot; angular; Woodruff key seat; concave; convex; gear cutters; corner-rounding; and other special cutters (see Figure 4.5).



Figure 4.5. A dovetail cutter (a), used for cutting angles, and a Woodruff key cutter (b), used for keys inserted in shafts.

Their individual designations pretty well explain what they are used for—T-slot cutter, for instance, is used to machine the T-slots in machine tool tables used for workpiece holddown and mounting of fixtures and various accessories; angle cutters are used to machine ways and other parts that must be machined to very precise angles; and the Woodruff key seat is used to machine the seat for half-round shaped keys.

## Carbides

All types of end mills except miniature end mills are available in solid carbide or with brazed carbide cutting edges or inserted carbide teeth.

Brazed carbide and inserted teeth carbides are the most rigid and the least prone to breaking. They typically are mounted on high-speed steel substrates.

## Roughing Cutters

Because of its ability to sharply slash machining times when large volumes of unwanted material must be removed, roughing cutters are very popular types of end mills. Available in sizes ranging from  $\frac{1}{4}$  in. to 3 in. in diameter, the roughing end mill usually has a 30-degree helical angle with notches machined in its flutes and cutting edges that act as “chipbreakers” that prevent formation of long strings of chips. Among other things, this conserves horsepower, but the main advantage is the fact that breaking up the chips permits using feeds up to four times higher than regular end mills, sharply boosting machining productivity. The materials used in manufacturing roughing and end mills feature high heat resistance, permitting somewhat higher cutting rates than finishing end mills can maintain.

## Milling Cutters

While “end mills” tend to be useful for a variety of applications, other milling cutters normally are developed for a specific use (see Figure 4.6).

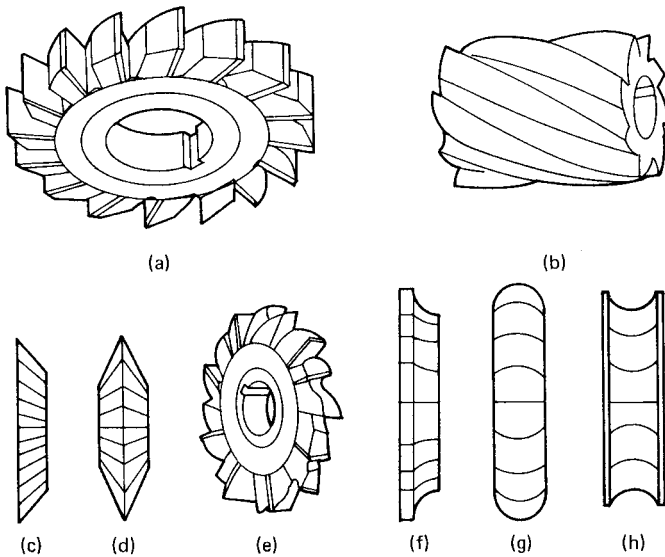


Figure 4.6. A variety of horizontal milling cutter: (a) side milling cutters; (b) plain milling cutter; (c) single-angle cutter; (d) double-angle cutter; (e) staggered-tooth cutter; (f) corner-rounding cutters; (g) convex cutter; (h) concave cutter.

Plain milling cutters feature cutting edges only on their outer periphery and generally are used for milling flat surfaces. They fall into several categories:

1. *Light-Duty.* Light-duty plain milling cutters have parallel flutes and are narrow in width—usually  $\frac{3}{4}$  in. or less. If a greater width is desired, then the flutes are helical to allow for better shearing of the metal when machining takes place.
2. *Heavy-Duty.* The heavy-duty plain milling cutter permits taking heavier cuts, accomplished by virtue of fewer teeth and helical flutes to promote efficient removal of chips. These cutters are wider than light-duty cutters, ranging anywhere from 3 in. to 6 in. in width. Helix angles range from 25 to 45 degrees, with a 30-degree helix being the typical choice.
3. *Narrow.* Very narrow plain cutters are used for slitting and for machining screwdriver and similar narrow slots. The plain cutter gains its “beam strength” from its cross-sectional width; its beam strength determines the cutter’s ability to resist lateral deflection. Consequently, the narrower the slot, the smaller the diameter of the cutter must be to ensure a straight cut.

When the sides or interior portions as well as the face of a workpiece must be milled, then milling cutters with cutting edges on their sides in addition to those on the periphery must be used. These come in two basic types: (1) those designed for specific yet fairly broad kinds of milling operations and (2) those designed to perform a specific but narrow or even single machining step.

Several basic types of milling cutters designed for the broader kinds of applications include plain side; half-side; and staggered-tooth milling cutters.

Their characteristics and general applications are as follows.

1. *Plain side cutters* feature parallel teeth on the side in addition to the parallel teeth on the periphery. The side teeth go in slightly toward the center of the cutter in order to provide relief to the cutting edges there. Normally, the teeth on the periphery do the rough-cutting whereas the side teeth are used to finish the slot to the required size.
2. *Half-side cutters* are used in combination with other cutters to machine slots of specific size. They can be obtained in various sizes to machine varying widths and depths, and, in addition, they can be used to size a part to an exact width using only one side of the cutter. This type of machining is called “straddle milling.”
3. *Staggered-tooth milling cutters* are used for milling operations in which typically only one side of the cutter is engaged at any given time. The teeth are longer and deeper on the side and are helical on the periphery, promoting and allowing greater chip clearance. These cutters permit heavy-duty milling.

## Milling Cutter Holders

The type of holder used for these milling cutters is the *arbor* (see Figure 4.7). Arbors come in various diameters, and the taper of the arbor that inserts into the spindle of the milling machine is the National Milling Machine Taper. This taper



*Figure 4.7. Several types of milling machine holders. (a) R-8 end mill holder. (b) ISO (National Taper) end mill holder. (c) Shell mill holder. (d) Milling machine arbor.*

is very steep, running at the rate of  $3\frac{1}{2}$  in./ft, and the arbor comes in sizes 30, 40, and 50. The size (designated ISO for international use) to be used is determined by the size of the milling machine.

The cutters are mounted to the arbor equipped with a key matched to a key slot in the cutter. Spacers are used to position the cutter at an exact point along the length of the arbor. When performing straddle milling or multicutter slotting operations, spaces are used to position the cutters to create the exact workpiece dimensions between slots or other milled features.

The arbor is mounted in the spindle and held firmly in place by means of a drawbar. There are two keys on the face of the spindle that match two slots in the tapered holder of the arbor; when the arbor is drawn tight by the drawbar and the keys seat in the slots, this design ensures a positive drive for the tool.

The arbor is supported at its outer end by an arbor support held in position by the overhead ram. For heavy-duty machining, additional support may be provided by extending a bar from the arbor support that is bolted to and held in place by the knee of the machine.

Where cutters are spaced far apart in straddle-milling or multiple slotting operations, an arbor support can be put in place between them to prevent deflection and ensure very precise milling.

## MILLING MACHINE ACCESSORIES

There are a considerable number of attachments and accessories for use with vertical and horizontal milling machines. Some of the most popular ones are discussed in the following sections.

### Indexing or Dividing Head

This device, available in both manually operated and power-driven versions, is probably the most popular attachment of all.

Work may be held in the dividing head chuck or between centers, with one mounted in the dividing head and the other mounted in a foot stock that supports the work and functions much like the tailstock on a lathe. The dividing head holds the workpiece and positions it very accurately, allowing virtually any angular form to be milled, including milling spiral gears, helical angles (used with a universal table), and splines.

Helping account for the popularity of the dividing head is its ability to permit machining evenly divided flats and holes on workpiece surfaces or gear-type splines for driving shafts with positive action. A dividing head is indispensable for machining gears on a milling machine, and, when used in conjunction with a universal milling machine (swiveling table), helical gears can be readily machined.

In addition, the dividing head makes it possible to easily yet accurately produce special cutting tools.

The attachment has a headstock, rootstock, and a steady rest for use when necessary or desirable for high-precision work.

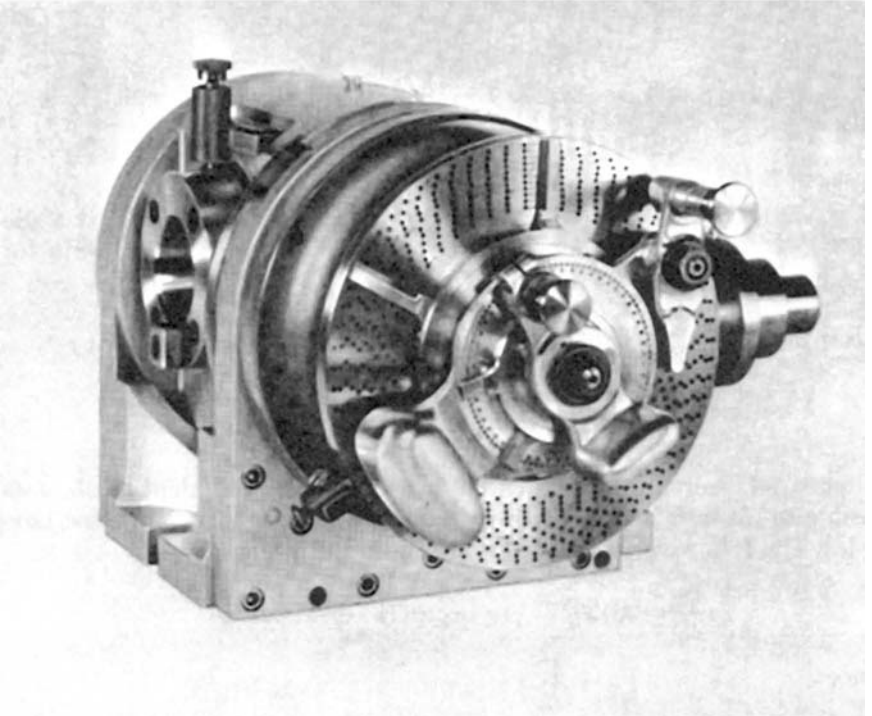
The workpiece can be mounted on the headstock in a chuck or between centers, with a dog and face plate used to guide the work. An index plate attached to the headstock is equipped with a series of precisely located holes that are used to divide the number of turns to position the work in the proper position.

When machining a helical angle, a series of gears can be attached to the dividing head and to the acme leadscrew of the table—which must be set to the correct angle—to permit obtaining the correct lead and helical angle.

### The Dividing Head

The dividing head is the headstock. The work is held in the headstock by a three-jaw chuck or it may be mounted between centers and driven by a drive plate with a dog.

An index plate with a plunger arm, an index crank, and a sector arm are on the side of the headstock (Figure 4.8). The index plate is a plate with a series of precisely located holes. The holes are usually arrayed in four rows, each row equally spaced and numbered so that they can be evenly divided by a denominator of a fraction.



*Figure 4.8. Rotary table (a) and index plate (b). The rotary table is used to mill arcs and circles. The index plate is used for equally dividing a hex shape or spacing holes in a bolt circle.*

To cover all possible machining requirements, several plates with combinations of holes are provided; these mount interchangeably on the headstock. When the plate is correctly positioned, an alignment pin is inserted to maintain that position.

Dividing heads have a 40:1 turn ratio; that is, it takes 40 turns of the index crank to make one complete revolution of the dividing head spindle. Consequently, any regular shape that divides evenly into 40 can be easily machined by dividing the number of sides into 40. (Example: If an octagon-shaped workpiece is to be machined, dividing 8 into 40 gives 5. Thus to machine each of the 8 sides, turn the crank five full times to machine each flat. When this has been done 8 times, the octagon will have been machined.)

If the shape to be machined does not fit evenly into 40, the process is a little more complicated. [Example: A hexagon is required. Dividing 6 into 40 gives  $6\frac{4}{6}$ . To machine the piece, it will be necessary to turn the handcrank  $6\frac{4}{6}$  times or 6 full turns plus  $\frac{4}{6}$  of a turn. To do this, a plate with a row of holes divisible by the denominator of the fraction (6) is required. If a plate with 18 holes is used, for instance, dividing by 6 gives a quotient of 3. That number is now multiplied by the numerator (4); the answer is 12. Consequently, to machine the hexagon, after the crank is turned 6 full times, it must be turned to span an additional 12 holes to complete the setting for each side.]

Another way to make the calculation is to multiply the number of holes in the index plate (18) by the fraction ( $\frac{4}{6}$ ). The answer is  $\frac{72}{6}$  or 12—the same answer as obtained above. Note that 12 is  $\frac{2}{3}$  or  $\frac{4}{6}$  of 18, the number of holes in the plate.

When setting up the dividing head, the sector arm, which is adjustable, is set up for 12 holes. The first hole will be used for the reference of 6 turns, then the plunger arm is used for the additional 12 holes.

After finding the position, move the sector arms for the next 6 turns and 12 holes after machining the first flat on the part. Repeat until the part (hexagon) is finished.

When the denominator is a number larger than the number of holes in the index plate to be used, reduce the fraction to its lowest denominator. For example, if 36 holes are to be drilled in a plate, the number of turns would be 40 divided by 36 or  $1\frac{1}{36}$ . This calls for the crank to be turned 1 full turn and  $\frac{4}{36}$  of a turn. If a 36-hole plate is available, the setting would be made by making 1 full turn and then moving another 4 holes. If a 36-hole plate is not available, reduce the fraction; this gives  $\frac{1}{9}$ . Use an 18-hole plate. Multiplying 18 times  $\frac{1}{9}$  gives an answer of 2. To machine the part using an 18-hole plate would require 1 full turn plus setting the sector arm to 2 additional holes.

If a sector is to be divided by degrees (angles), the procedure is:

Divide 360 degrees (the number of degrees in a full circle) by 40 (the number of times the handcrank must be turned for one full revolution of the dividing head). The result is 9; thus each revolution of the handcrank represents 9 degrees. If the index plate being used has 18 holes in it, then each hole amounts to  $\frac{9}{18}$  degrees or  $\frac{1}{2}$  degree—that is, 30 minutes.

If 36 holes are to be placed in a plate to degrees apart and an 18-hole index plate is being used, each hole setting would require 1 full revolution of the crank



plus setting the sector arm for two additional holes (the full revolution provides 9 of the 10 degrees required; each hole accounts for 30 minutes of  $\frac{1}{2}$  degree, so 2 holes rounds out the setting to 10 degrees).

## Cutting a Helix Angle with the Dividing Head

Machining such helical shapes as flutes in drills, taps, end mills, or gears is helical milling. A universal milling machine must be used, and the machining is accomplished by gearing the dividing head to the lead screw in the machine table.

Gearing the dividing head to the table makes it possible to generate a path with a cutting tool lengthwise at a uniform rate while turning the workpiece cylinder at a uniform rate.

The procedure is as follows.

1. Gear the dividing head to the table leadscrew.
2. Determine the helix angle; this is the intersection of the helix with the axis of the part being machined.
3. Set the table to the proper direction of the angle.
4. Mount the correct gears linking the dividing head to the table to ensure that the work will rotate exactly one full turn while the table travels the distance of the lead. The lead is the distance the helix angle moves in one full revolution. For selection of gears, see pages 1470–1479, *Machinery's Handbook*, 22nd edition.
5. Mount the dividing head at the end of the table.
6. Mount the work between centers.
7. Set the index crank and sector arms to the number of turns and holes to be used for the number of grooves to be machined.
8. Mount the cutter on the arbor and center it over the work. Raise the table so that the cutter just touches the work. Clear the cutter so the work can be raised to make the correct depth of cut.
9. Turn the machine on and machine all of the flutes for the part to be made. Use plenty of cutting fluid while taking chips.

Helical milling on a milling machine is usually performed when a special or a one-of-a-kind repair must be made to prevent the downtime that would be necessary to obtain the part from an outside source. But while such work is normally only performed as an emergency step, machining just one part or a special tool when required may more than justify the time and effort required to make it.

Cams also can be machined on a milling machine using a dividing head, but virtually always can be made faster with comparable accuracy by laying it out and cutting it on contour band machine and then finishing it by filing and polishing.

## The Rotary Table

A rotary table is similar to a dividing head, but its purpose is different. The rotary table is normally used for much larger work, and allows for the machining

of shapes and holes, whereas the dividing head is used primarily for precision positioning. Use of the rotary table for helical shapes is not convenient or practical.

The rotary head comes in two basic variations—one lies flat on the table, the other can be mounted either flat or standing upright—and various diameters, usually from 9 in. to 24 in. The table is usually divided into degrees of motion and can be purchased with index plates that can be attached to its handle. Indexing of the table with index plates is very convenient when equal divisions of a circle in minutes is desired.

The rotary table is very useful when machining arcs on parts. It can be provided with motors to drive the table for machining, making the unit very useful and handy for toolroom work. In addition, the rotary table can be used and interfaced with an NC machine to provide an additional axis.

The rotary table can be mounted or removed from the machine very easily, making it a very useful and versatile accessory.

## **The Slotting Attachment**

This device converts the rotary motion of the spindle into a reciprocating motion, up and down on the vertical mill, suitable for machining slots. A single-point tool similar in appearance to a lathe necking tool is used for that purpose. The attachment can be swiveled, permitting the machining of slots at various angles.

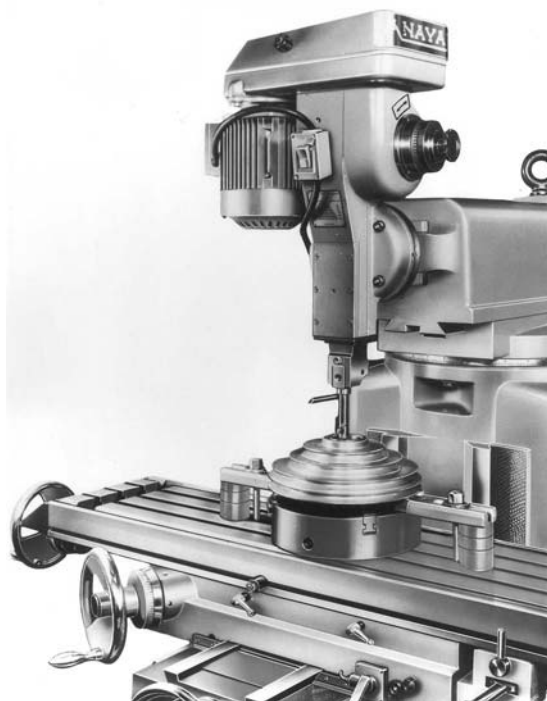
In addition to the spindle-mounted version, there is another type of slotting attachment that mounts on the back of the ram through a hole designed for it and is not driven by the spindle. This type of attachment has its own power source and thus is self-driven; the ram is simply rotated 180 degrees to position the attachment directly over the machine table (see Figure 4.9).

## **The High-speed Attachment**

This unit mounts to the spindle and has internal gearing that boosts RPMs up to four times the normal spindle speed range. This device is used to hold and turn very small end mills and drills that must rotate at very high speeds, because of their tiny diameter. This attachment is “universal,” which simply means that it can be rotated to any angle to accommodate special shapes and forms.

## **Horizontal Milling Attachment**

This attachment for vertical mills permits them to machine in the manner of horizontal mills. It permits vertical milling machines to handle jobs in which arbor mounted cutters are indicated or in which horizontal milling is the preferred method. The attachment is powered by the spindle and, since power is lost in the process (and vertical mills usually are not as heavy and rigid as comparably sized horizontal model), the attachment is only suitable for light-duty horizontal work.



*Figure 4.9. A slotting attachment for a vertical mill. This attachment fits on the reverse side of the ram and has its own motor for driving the slotting tool. (Courtesy of DoAll.)*

## **Vertical Milling Attachment**

This device permits a horizontal mill to perform vertical milling steps. By swinging the head up to 45 degrees to either side of the vertical position, the attachment can also be used for machining angular surfaces.

The Vertical Milling Attachment is also available in “compound” versions with spindles that can be set in two planes, simplifying the setup for such angular work as cutting long bevels.

## **Universal Milling Attachment**

This is similar to the horizontal and vertical milling attachments, but, since its spindle may be set to any angle in both the horizontal and vertical planes, it has the effect of making the machine, as the attachment name implies, truly “universal.” This permits it to perform the same work as such other accessories as the vertical milling attachment, making them unnecessary.

## **Universal Spiral Attachment**

This unit permits a plain milling machine (the table does not swivel) to cut spirals, rack teeth, helical gears, screws, threads, and worms that otherwise can only be cut by a universal milling machine (one with a swiveling table).

The spiral attachment can also be used on a universal machine—it's a little simpler than swiveling the table—but in most cases the attachment is too costly for the amount of use it gets in the usual mix of machine shop work.

Other attachments and accessories designed to make setup and operation easier and more productive are as follows.

## **Milling Machine Arbors**

These are used mainly on horizontal mills to carry angle cutters and cutters either for slabbing (facing) or for slotting.

## **Collet Holders**

These are used to reduce the size of the spindle taper, permitting it to hold smaller end mills with straight or tapered shanks. They also are handy when various size tools with the same-size shanks are required for a variety of work, since they can be quickly changed.

## **Quick-Change Holders**

These are adaptors that fit into the spindle of the head and feature a special clamping arrangement that permits a cutting tool mounted in a tapered holder to be inserted directly into the spindle and tightened. This holder performs the same function as collet holders.

Quick-change holders allow more rapid tool change than can be accomplished with a drawbar. However, they are not yet standardized, so one maker's tools and holders are not interchangeable with those of another manufacturer.

These are other types of adaptors, namely, those that permit holding facing cutters, such as shell end mills.

## **Workholding Fixtures**

These include vises, parallels, holddowns of various kinds, etc.

## **Digital Readouts**

These supplement the conventional handwheel dials, improving the accuracy of positioning by eliminating parallax problems and making it unnecessary to keep track of handwheel turns, etc.

Readouts are very useful accessories for both vertical and horizontal machines because they permit maintenance of a level of accuracy otherwise not attainable. The reasons for this are: (1) They are very easy to read, eliminating such possible sources of error as losing track of the number of turns in setting a dimension. (2) They eliminate backlash; they operate on glass scales and do not rely on the leadscrews for the accuracy of their displays. (3) The scales on digital readouts can read to "tenths"—0.0001 in.

Digital readouts are available in versions that monitor one, two, or three dimensions, and can be programmable. The programmable feature constitutes an

excellent “learning tool” in that it involves a simple programming step that leads into numerical control.

The three axes are the  $X$  axis (longitudinal), the  $Y$  axis (cross-feed), and the  $Z$  axis (vertical feed). Axis direction is indicated by a plus (+) or a minus (−) sign. The appropriate sign is determined by the direction the cutting tool moves in removing material, and not the direction in which the table is moving. For example: (1) If the cutting tool is removing material from left to right in the  $X$  axis, it is considered to be moving in the positive (+) direction. If it is removing material from right to left, it is moving in the negative (−) direction. (2) If material is being removed in the direction of the column in cross-feed operation, that is considered positive (+) movement. If material is being removed away from the column, the direction is negative (−). (3) Removing material upward on the  $Z$  axis is positive (+) motion; when the cutting tool is removing metal downward—such as when drilling or boring—that is negative (−) movement.

The digital readout basically is a point-to-point operation—that is, measurements are relative from one point to another. If a series of holes is to be drilled, then the first hole is located from a reference point and each successive hole is located from the preceding one. If “frame milling” is to be done—that is, a series of slots is to be milled to a specific width—then each distance is located from a point on the preceding slot. This is called *incremental milling*.

If one reference point is used and all other locations are found from that original reference point, then this is called *absolute milling*. This can also be done using the digital readout.

When more than one part is to be made, a digital readout with memory is handy and can be used for as many parts as necessary. The memory holds all required figures, and these can be recalled as needed.

Digital readouts with memories are typically programmable types, and these permit making computations for machining requirements in advance and entering them into the memory of the readout before setting up the job for machining.

When making more than one part, location of the piece is vital; a stop or simple fixture mounted to the table or a vise locates each successive part quickly and accurately. The cutting tool also must be properly located relative to the work, and speeds and feeds should be computed in advance. Finally, a reference point indicating precisely where the tool is to be located at the start of the machining operation should be established.

With these steps taken, machining each part becomes a simple matter. All programming should be kept on record so that it can be recalled for future reruns.

These are the most popular accessories; there are other devices such as the rack milling attachment for milling the teeth of the rack portion of a rack-and-pinion, but they tend to be narrowly special-purpose or even single-purpose.

## THE BASIC MILLING MACHINE

The milling machine comes in two basic versions that perhaps can be best understood when compared and contrasted with each other.

The most obvious difference between a vertical and a horizontal milling machine is the way in which the cutting tool is held and wielded:

1. *Vertical.* Not only is the cutting tool held vertically on the vertical machine, but the head can be tilted up to 45 degrees to the left or to the right and 45 degrees front-to-rear. This gives the vertical version great flexibility and versatility in maneuvering the cutting tool to reshape a workpiece. The trade-off for that flexibility and versatility is a tool that is normally supported at only one end; exceptions include machining with tools that have pilots, such for as spotfacing. Single-end support limits the amount of power that can be applied without creating unacceptable tool deflection and, consequently, limits the volume of chips that can be removed in a given time period. There is no point in giving a mill more power than it can use, so vertical mills typically are lighter and less powerful than horizontal models that handle comparably sized work.
2. *Horizontal.* Although the spindle is horizontal and cannot be angled without a special attachment of some kind, that horizontal position permits using arbors and outboard supports. That, in turn, means the cutting tool can be given much greater support than is possible on the vertical machine, and therefore tool rigidity will be substantially greater. As a result, more horsepower can be utilized, permitting greater chip removal rates for any given machine size or “workpiece envelope” (the maximum dimensions a workpiece can possess and be machined without re-positioning the work).

Together, the vertical and horizontal milling machines make a potent pair: The horizontal machine is ideal for high-horsepower, heavy-duty, high-chip removal tasks, whereas the vertical mill is by far the more versatile and more commonly used.

## The Vertical Milling Machine

This is convenient to set up and use, and it can readily accomplish many functions that either cannot be done, or are more difficult to perform, on a horizontal mill. A vertical milling attachment can be mounted to a horizontal mill, thereby, in effect, converting it to a vertical mill. This, however, is akin to using a big over-the-road tractor-trailer to make deliveries. If one is available, it is much simpler, easier, and more economical to use the vertical mill and reserve the horizontal model for more suitable application.

## Setting Up the Vertical Milling Machine

Although the vertical milling machine is relatively simple to use, the alignment of the head to the table and the alignment of the workpiece or of the vise or other workholding device to the head is critical for accurate machining.

### Tramming the Head

Because the head of the vertical milling machine can be tilted side-to-side and front-to-back for machining versatility, it is essential to check the head-to-table

alignment from time to time to be sure it does not inadvertently “drift” out of square as well as to realign to the table after all angled operations. Trimming the head ensures that the cutting tool will be held at precisely the correct angle; on a properly trimmed vertical milling machine, an end mill, for example, will be held at exactly a 90-degree angle to all points on the table.

The step-by-step procedure for trimming is:

1. Mount a dial indicator, held on a rod clamped at exactly a 90-degree angle to another one mounted in the spindle (Figure 4.10). Naturally, the longer the horizontally held rod, the more accurate the reading will be, provided the arrangement is sufficiently rigid. It should extend a minimum of 6 in. from the spindle; this permits indicating a full 12 in. of the table. The dial indicator should read to 0.001 in. or less.
2. Position the indicator over the front of the table.
3. Lower the spindle until the probe on the dial touches the table and then *gently* continue until the dial registers about one-half of a revolution. Set the bezel to zero.
4. Rotate the spindle 180 degrees by hand and, with the dial probe now resting on the table surface at the rear of the table, check the dial.
5. If the readings match, the head is aligned from front to rear. If they do not match, divide the difference between the front and the back reading by two. Now loosen the locking nuts on the swivel mounting for front-to-back angling and adjust the head until the dial registers one-half of



*Figure 4.10. A right-angle milling attachment for a vertical mill. It fits directly in the spindle and is driven by the milling machine head motor. (Courtesy of DoAll.)*

the difference between the two readings. Retighten the locking nuts and repeat step 5; continue until the front and rear readings match.

6. When the machine is aligned front-to-rear, rotate the indicator dial 90 degrees to one side or the other, and repeat steps 1 through 4, but from side to side rather than front to back.
7. Now rotate the dial indicator 180 degrees and read the indicator. If the readings match, the machine is now “trammed in.” If not, repeat the alignment process described in steps 5 and 6, but from side to side. (Loosen the locking nuts for side to side swiveling, and tilt the head to take up one-half of the difference between readings. Tighten lock nuts and check readings. When they match, the head is trammed in.)

*Note:* When rotating the dial indicator to take readings, it is important not to let the probe catch in the “T-slots” of the table, as the setup is delicate and this can misalign the dial and rod arrangement. The best way to minimize the chance of the probe catching in the slots is, when the initial readings do not match, to work from the highest to the lowest reading.

## Aligning the Work to the Head

To ensure that the cutting tool will create the desired shape when it transits through the work, the work must be properly aligned to the head. This can be accomplished by mounting it directly to the table and aligning it to the head or by mounting it in a vise or other workholding device that aligns the work to itself and then aligning the workholding device to the head.

**Mounting the Work Directly.** Unless the part is only to be milled on its upper segments, it should be mounted on parallels to preclude the possibility of machining the table. To align the part to the head, the workpiece must have a finished side or slot that is known to be square. An indicator is then mounted in the spindle and run along the side or slot with the part loosely clamped.

Tap the part on the “high side” with a soft-faced hammer until the indicator reads the same all along the side or slot, then tighten the holddowns. The indicator is mounted and applied as described in aligning the vise (the following subsection).

**Aligning Work Held in a Vise.** When the work to be milled is held in a vise clamped to the table, the work must be properly aligned to the vise and the vise must be aligned to the table. With the head properly trammed to the table, that ensures the squareness of the vise to the spindle.

The procedure for aligning the vise is as follows.

1. Mount the vise on the milling machine table and secure it with bolts located in the “T-slots” of the table. The bolts should be snug but not yet tightly clamped, permitting the vise to be adjusted for proper alignment.
2. Mount an indicator with 0.001-in. graduations in the spindle and secure it using a suitable holder.



3. Touch the fixed jaw of the vise with the indicator probe so that the pointer of the indicator moves about one-half of one full revolution of the dial face.
4. Run the table parallel to the direction of the jaw of the vise and read the indicator from one end to the other. Using a soft-faced hammer, adjust the vise until the reading does not deviate from zero all the way across the jaw.
5. When the reading is zero all the way across the jaw, tighten the holddown bolts securely and take another reading to make sure the tightening process did not disturb alignment. If the reading does deviate, repeat the above process until the reading runs true.

*Note:* Square and rectangular work is automatically aligned in the vise, but care must be taken to ensure that the piece is not improperly clamped. If the part is mounted on parallels in the vise, tap it gently with a soft-faced hammer, then clamp it and check by hand to make sure the piece will not rock, indicating that it is slightly cocked.

With the workpiece properly held and aligned to the vise, the vise to the table, and the table to the head (“trammed”), the work will be properly aligned to the spindle.

## Angling the Head

The vertical mill has angular scales on the appropriate side of the mill head that permit fast adjustment to the desired angle setting. These are, however, merely approximations; so, whenever angles are to be machined to very precise settings, it is also necessary to make the angle setting precise.

The procedure is as follows.

1. Bolt an angle plate securely to the table, set an indicator in the spindle, and indicate the angle plate square to the spindle.
2. Set up a sine bar, using the combination of gage blocks required to set the correct angle. This setup should be made alongside the angle plate so that it can be clamped lightly to the plate when it is completed.
3. Using the angular scales as a guide, swivel the head of the mill to the approximate angle desired.
4. Now touch the indicator to the sine bar with sufficient pressure on the probe tip to move the dial indicator to make about one-half of a full revolution.
5. Using the hand crank on the vertical head, feed the spindle along the sine bar; travel will be from 5 to 6 in., depending on the size of the machine. If the reading does not deviate, the correct angle has been set; if it does, adjust the head until the reading remains constant.

*Note:* The handcrank for engaging the spindle is activated by pulling out a knob mounted at the center of the handle; this knob is normally engaged for automatic feed when drilling or boring. When performing angular machining, disengage this knob once the head has been aligned.

## Compound Angles

Compound angles can be milled very easily by tilting the head in one direction and then, if the part is being held in a swivel-based vise, swiveling the vise for the second angle.

If the part is not being held in a vise, then the part must be located on the table for the second angle required.

## Machining Slots

Use of the vertical milling machine to machine slots is routine (see “Milling Cutters,” page 99), with keyway slotting being the most common slotting operation. Slot milling is typically done using two-flute milling cutters; tolerance is plus 0.000 in., minus 0.002 in. The two-flute end mill has cutting edges on its tip (face) that allow it to make “plunge cuts” (that is, function somewhat like a drill), permitting it to cut internal slots.

Three-flute milling cutters may also be used; they also permit plunge cutting. Deflection of the milling cutter is perhaps the most tricky problem encountered in slotting operations. Consequently, feeds and speeds must be very carefully selected and set, and, when deep slots are being machined, a number of light cuts normally will provide better results than fewer heavy cuts.

The end mill has a straight shank that can be inserted either into holders made expressly for that size shank or into collets made for that shank size. Collets are convenient to use because removal from the holder is somewhat simpler than is true of conventional holders. End mills mounted in collets are held by the friction created when the collet is tightened against the tool by the action of the drawbar, and, consequently, can slip under heavy loads. For that reason, regular holders in which a set screw is secured against a flat in the end mill shank (Figure 4.7) are preferable, since they cannot slip under heavier loads.

Speeds are the same as for any other cutter of the same diameter cutting the same material, but feeds may be adjusted to the lighter side if the end mill is very small. Lighter feeds also are necessary when using small cutters to avoid tool breakage.

When a protrusion—a slot in reverse—is to be milled to a specific width, then a four-flute end mill is used. The four flutes will ensure a satisfactory finish while permitting efficient cutting rates, on small- to medium-sized work, whereas larger workpieces may call for a large diameter, six-flute end mill.

The procedure for machining slots is as follows.

1. Lay out reference lines showing exactly where the slot will be. Although this step can be skipped if accuracy is not vital, it ensures detection of any deviation, allowing correction of centering in that event. Layout can be accomplished after mounting the part in a vise clamped to the table. Find the center with a combination square, apply layout ink, and then scribe the centerline.
2. Select a two-flute end mill with a diameter equal to the width of the slot and center it. This can be done by first touching the cutter very lightly to one side of the shaft, then raising the cutter and moving it one-half the

width of the shaft and one-half the width of the cutter. A more accurate centering procedure is to use an edge finder. Touch the swiveling end of the tool to the side of the part. When the tool stops swiveling and moves exactly in the plane of its own body, it is precisely in line with the part being measured. After aligning the edge finder, move it one-half the width of the part and then a distance equal to one-half of the edge finder's diameter. The spindle is now located over the exact centerline of the part. Remove the edge finder and insert the end mill. Lock the cross-feed table.

3. Turn on the machine and then move the cutter into the work until the cutter is removing its full diameter; this can be done by elevating the table or lowering the tool using the quill. Lock the head and vertical movement of the table before machining.
4. Machine the slot to the proper depth. For a keyway, the correct depth is one-half the thickness of the key.

## Machining a Woodruff Key

A Woodruff key is a semicircular key frequently used for various types of fittings such as shaft-mounted keys on operating machinery. A major advantage is their positive locking action combined with relative ease of insertion and removal.

The keyway for a Woodruff key is known as a "keyseat," and the tool used to create it is circular and made to create a curved slot that the key fits precisely. The machining procedure is similar to that for making a regular keyway.

The procedure is as follows.

1. Lay out the center of the shaft, using layout ink and scribing the slot so that deviations can be caught and corrected. When mounting the shaft in the vise, set the layout line parallel to the cutter face.
2. Touch the top of the shaft with the face of the cutter. The machining will be done on the side of the shaft because of the way the tool is held in the spindle.
3. Raise the table (or lower the quill) one-half the diameter of the shaft to be machined, and then one-half the width of the Woodruff cutter. Lock the head and the vertical movement of the table.
4. Using the closest end of the shaft as reference, locate the center of the slot where it is to be positioned on the shaft. Lock the horizontal movement of the table.
5. Turn on the machine and touch the part with the cutter.
6. Machine the keyseat to the proper depth; keyseat proportions and depth of cut are listed in *Machinery's Handbook*, 22nd ed., pages 871–894.

Whether the part is to be clamped in a vise or mounted on the table is determined by the length and width of the shaft. If the part is too long to be held properly in a vise, the portions not supported will vibrate, causing poor finish and inaccuracies in measurements. In that case, the part should be mounted on the

table. It can be mounted directly to the table, but the usual practice is to mount it on parallels placed so as to support all main bearing points.

## Machining Circular Slots on a Rotary Table

Circular slots and arcs can readily be milled using a rotary table.

The procedure is as follows.

1. Center the rotary table to the spindle to ensure accuracy. This is accomplished using the hole for pilot guides that all rotary tables have at their exact center. Taking a plug of the exact hole size (one can readily be made, or a drill blank of the proper size can be used), insert it into the center hole to use as a pilot. Make sure the plug fits snugly, with no play, with enough protruding for a dial probe to take a reading all around its diameter.
2. Insert an indicator into the spindle, holding it in a drill chuck or a holder made for the purpose. Indicate around the plug, adjusting the rotary table position until the indicator reads zero in all positions. The table can be rotated to accomplish this using its crank handle or, with the crank disengaged, turning it by hand. The table must be manually adjusted on the *X* and *Y* axes for the indicator reading; this is done by tapping the table lightly with a soft-faced hammer. The table should be lightly clamped during the adjustment stage so these taps can move it. Once the indicator rotates without any deviation in readings, the setting is correct. Now securely tighten the clamping bolts. **Important: Do not move the machine table cross-feed or longitudinal feed handles.**

The rotary table is now ready to accept a part and machine a radius in the form of a circle or of an arc.

The procedure is as follows.

1. Mount the workpiece on the rotary table, aligning the center of the radius to be machined with the center of the table. This can be done in two ways:
  - a. If the part had a center hole or one can be drilled to the same size as the rotary-table center hole without affecting the part's function, then a rod of the exact diameter of the center holes will precisely align the rotary table and the part. The part can then be clamped to the rotary table "T-slots."
  - b. If that procedure cannot be followed, then centerpunch the workpiece at the center of the radius. Mount the part on the table and locate with a center finder. Move only the workpiece; it should be lightly clamped so it can be adjusted by tapping it lightly. Once it is properly aligned, the part can be secured by clamping to the rotary table.
2. Once the part has been located, use the cross-feed (*Y* axis) or the longitudinal (*X* axis) of the milling machine to move the table an amount equal to the radius required. Once this is done, lock both the *X* axis and the *Y* axis on the machine to preclude accidental movement.
3. Adjust for the diameter of the end mill that will create the radius.

4. Mount the end mill and rotate the rotary table to its starting point. Set the depth of cut, start the machine, and cut the slot. Depending on the diameter of the cutter and the depth of the slot, several passes may be necessary.

The ratio of turns of the indexing attachment on the rotary table is normally 40:1, as it is on the dividing head, but other ratios are used—for example, 72:1 or 80:1. The method of calculating the number of times the hand crank must be turned to achieve a desired rotation of the rotary table is the same as for the dividing head, except, of course, for the number to be used. The formula is:

$$\text{Indexing} = \frac{\text{Number of teeth}}{\text{Number of division required}}$$

If, for example, the ratio of the rotary table is 120:1 and 10 holes to be drilled, the number of turns of the hand crank required to set each of the 10 divisions or holes or positions would be:

$$\text{Indexing} = \frac{120}{10} = 12 \text{ turns (per division)}$$

## Milling Dovetails on a Rotary Table

A dovetail is a single-angle end mill ground to a specific angle. The dovetail is used to achieve tightly (precisely) controlled reciprocative motion between two parts, such as the table and column of many milling machines. To ensure accurate fit and motion, gibs are used to make any adjustments necessary between the positioning of the two dovetails.

There are two types of dovetails, internal and external, the one mating to the other.

Milling dovetails on a rotary table eliminates possible errors that can occur in the event the set-up must be moved. The procedure is as follows.

1. With the rotary table located to the spindle, center the workpiece in the center of the rotary table and indicate the part so that it is parallel to the line of table travel.
2. Locate the center of the dovetail with an edge finder, moving the *Y* axis the correct distance to compensate for the diameter of the edge finder.
3. Mount an end mill with a diameter *narrower* than the width of the center section of the dovetail in the spindle.
4. Touch the tip of the end mill to the top of the work, then move it clear of the piece. Set the depth of the cut about 0.032 in. deeper than the bottom of the finished dovetail; this allows clearance for the cutter.
5. Mill the slot the width of the upper portion of the dovetail.
6. Remove the end mill and insert the dovetail cutter. Set a depth for a roughing cut, leaving about 0.020 in. for finishing. Make the roughing cut.
7. Clear the cutter from the workpiece and rotate the rotary table 180 degrees.

8. Without moving the cross-feed of the machine, make the roughing cut on the new side.
9. Clear the dovetail cutter from the workpiece and adjust the setting 0.020 in. deeper for the finish cut.
10. Make the finishing cut, clear the work, rotate the rotary table 180 degrees, and make the finish cut on the other side.

Use cutting fluid liberally while machining. If dovetails are machined without the use of a rotary table, then the cross-feed table must be used to position the cutter for each side. All alignment procedures must be followed to ensure proper location of the dovetail.

## Machining T-slots

T-slots are used with T-bolts to hold parts securely to machine tables, fixtures, and similar devices. Specifications for T-slots are listed in *Machinery's Handbook* (page 1017, 22nd edition). T-slot cutters are made to conform to these specifications.

To mill T-slots, an end mill is first used to mill the clearance for the bolt body. This size should be slightly larger than the bolt body to allow proper clearance so that the bolt body will fit almost snugly while permitting it to be freely moved in and out of the slot.

After milling the body clearance, remove the end mill and mount the appropriate T-slot cutter.

Locate the correct depth and, using plenty of cutting fluid, mill out the slot.

## MILLING WITH THE HORIZONTAL MILLING MACHINE

The versatility of the milling machine makes it one of the most important machines in the shop—not only can it mill flat (facing) and irregularly shaped workpieces, but it can also cut gears, threads, bevels, splines, shafts, and racks as well as slot, drill, ream, and bore.

The horizontal mill can, with attachments, perform every operation in the milling sphere. Because of its power and rigidity it is, however, particularly well-suited for facing operations.

## Slab Mill

The plain horizontal milling machine used primarily for machining flat surfaces is known as a slab mill. Because the basic idea is to accurately create parallel top and bottom surfaces, it is usually not necessary to square the workpiece or workholding device to the table travel.

For slabbing operations, the cutter should be wider than the part being machined; this permits the operation to be completed with one pass. The cutting

speed must be correct for the material being machined, and cutting fluid should be liberally applied at the point of cut.

Rigidity of the setup is important. Since the cutter will be mounted on an arbor, the idea is to keep the cutter as close to the column as possible to ensure maximum rigidity while allowing for tool clearance and for the cutter width to slightly exceed the workpiece width.

## Feeds and Speeds

The feed rate is independent of the spindle RPM and must be calculated for the material being machined. There are charts available for initial set-up; the calculations are, however, based on the stock removal capability of each cutting edge.

### Cutting Speed

The formula for determining cutting speed is

$$\text{RMM} = \frac{\text{Surface feet per minute} \times 12}{\pi (=3.1416) \times \text{Diameter (cutting tool)}}$$

### Feed Rate

Refer to the chart to determine the feed rate per tooth to be set for the workpiece material. Multiply that rate times the number of teeth. Finally, multiply that times the RPM, as determined using the above formula. The formula for determining the feed rate is

$$\text{Feed rate} = \text{Feed per tooth} \times \text{Number of teeth} \times \text{RPM}$$

## Setting Up the Slab Mill

The procedure for setting up the slab mill is as follows.

1. Mount the slab mill cutter on a suitable arbor. Remember to keep the cutter as close to the column as possible to ensure rigidity, while also making certain there is enough room to mill the part completely in one pass without anything interfering.
2. Make sure the cutter keyway matches the key on the shaft of the arbor. Use an appropriate set of the spacers that come with the arbor to properly position the cutter.
3. Clamp the arbor to the spindle and attach the outboard support. Note that there are two basic types of arbors:
  - a. *Light-duty*—features a pin on its end that fits into the arbor support with a small bearing. This type is found on small, light-duty mills.
  - b. *Heavy-duty*—has a large support on the end that fits into a heavy-duty bearing on the arbor support. The bearing may be either a bronze bearing or, to allow for rotation of the arbor, a ball bearing.

4. Mount the work to be milled in a vise or on the table of the milling machine. It is a good idea to use parallels under the work to allow for clearance between the table and the work. If the work is to be mounted directly to the table rather than held in a vise and the entire top surface is to be faced, then clamp holes should be machined in the sides of the work to permit clamping it to the table. If the workpiece will only be faced on a portion of the top surface, then it may be possible to clamp over the top and avoid the need for machining clamp holes. When all surfaces of the workpiece are to be faced, then a vise must be used.
5. Select a cutting fluid, fill the coolant reservoir, and position the coolant nozzle so it will apply the fluid at the point of cut. If the machine does not have an internal coolant tank, then use an external one.
6. Using milling recommendation charts and the formulas listed above, find the correct cutting speed for the material (in SFPM) and calculate the feed rate to be used.
7. Start the spindle and dial in the correct speed. Make sure the cutter is clear of the work, then set the depth of cut to be made. Set the feed rate and start cutting. It is sometimes necessary to make minor adjustments to achieve optimum cutting results. These adjustments can be made while machining is taking place; it is usually best to make adjustments in small incremental steps.

## Plain Side Milling Cutters

Plain side milling cutters come in various distinct types, each designed to perform specific types of operations:

- \* *Straight-tooth* cutters can be mounted with spacers separating to precise distances so that they will mill widths to a specific size.
- \* *Side* cutters, matched for same-sized diameters, can be mounted side by side to mill a slot of specific size when a slotting cutter of the desired size is not available (Figure 4.6).
- \* *Staggered-tooth* cutters (Figure 4.6) are used to machine deep slots or grooves. The design features generous chip clearance allowance and helical teeth to shear the metal for fast, efficient stock removal.

## Conventional and Climb Milling

The two basic methods of milling are conventional and climb milling (Figure 4.11).

In conventional milling, the cutting edges move upward as the part is fed into the cutter. The cutter teeth and the workpiece move in opposite directions.

In climb milling, the cutting edges move downward as the work is being fed into it. The cutter teeth and the workpiece are moving in the same direction.

As the name implies, conventional milling was the method originally used. Table backlash was not compensated for in older conventional milling machines,



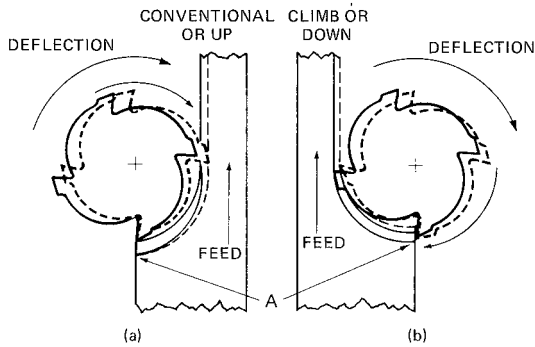


Figure 4.11. (a) Conventional and (b) climb milling, and the deflection of each when used. When possible, climb milling is preferred for the better finish it produces.

and therefore the force exerted in climb milling would pull the table along with the work, damaging the work and the cutter. Newer machines control backlash, permitting climb milling to be safely used.

Although climb milling puts more stress on the machine and can cause damage if there is any play in the table or workholding device, it provides a much better finish than conventional milling. The reason for this lies in the fact that in climb milling, the cutting edge takes its heaviest “bite” at the beginning of the cutting action and the lighter cut taken at the exit point promotes a finer finish.

In contrast, in conventional milling, the light cut is taken at the beginning of the cut, whereas the cutting edges are taking the heaviest cut near the exit point. The result is a slight tearing and breaking at the point the cutting edges exit, leaving a rough surface.

## Facing on the Vertical Mill

Facing of materials on vertical milling machines is very common. Although this is normally done with end mills, if the vertical mill has the size and power to effectively use them, some very large cutters can be used.

Shell end mills normally are used when facing materials more than 2 in. in width. They are mounted on holders that have a pilot with a bolt hole in the center and two keys opposite each other (Figure 4.7).

The cutter has a pilot hole and a shoulder smaller than the pilot so that a bolt can be inserted and tightened for rigid mounting. Two keyways are located on the cutter; these match those on the holder and are lined up before the bolt is tightened. The keys provide positive drive of the cutter during machining.

The taper of the shell end mill holder is usually a National Standard, size 40 or 50. Facing of the material is accomplished by the cutting edges on the face of the cutter. Good stock removal rates are possible provided the machine has sufficient horsepower; cutting fluid must be used for optimal results.

The procedure for facing with a shell end mill is as follows.

1. Select a cutter with the appropriate size for facing the top of the work-piece. Mount the cutter on a shell end mill holder and secure it tightly.
2. Insert the holder with the cutter into the spindle of the mill and tighten the drawbar until the holder is firmly seated in the spindle.
3. Mount the work in a vise or on the table, bolting it down using clamp holes.
4. Select the proper recommended speed and feed. Calculate the feed for the machine setting.
5. Fill the coolant reservoir with the proper cutting fluid for the material and job; if the machine does not have an internal reservoir, use an external tank. Start the machine spindle, then, with the tool clear of the work, raise the table for the depth of cut. When using a carbide cutter, do not use cutting fluid: it may cause thermal shock and crack the carbide.
6. Set the feed rate and, applying coolant to the point of cut and power to the table, start machining. Adjustments in feed and speed can be made if necessary.

## **BORING ON A MILLING MACHINE**

Milling machines can do a highly satisfactory job of boring holes to reasonably accurate nominal dimensions, and this operation is a common, accepted, and perhaps an even routine step.

### **Milling Machine Versus Jig Bore Machine**

If, however, extreme precision is a requirement, then a jig bore machine must be used. For instance, hole locations for interchangeable parts must be held to extremely precision tolerances to ensure that the parts will truly be “interchangeable.”

In high-production drilling of holes, the guide used to ensure precision location of each hole is known as a “jig,” and the jig can be the guide for a single hole or for hundreds. The drill is guided by hardened bushings located in these holes. Each bushing is pressed in the hole. Location must be accurate to within 0.0001 in. to ensure accuracy, and so for such applications a jig bore and its sister machine, the jig grinder, must be used to locate and bore the locating hole that a bushing fits in and that guides the drill.

The jig bore looks like a milling machine, but the difference in their respective price underscores the difference in the levels of accuracy they can maintain. The jig bore is a very expensive machine that, in addition to making interchangeable parts, produces high-precision punch pads and die plates; it also bores pilot holes to locate parts and bushing holes that align the stripper plates with dies.

For these kinds of operations, only a jig bore will do. There are, however, a great many boring tasks that are not as demanding, and for these, the less expensive “look-alike” milling machine can do the job.

## Boring Tools

Boring is done with drills, reamers, counterbores, and single-point tools held in a boring tool chuck.

Single-point boring is the most accurate way of generating a hole, and for that reason it is commonly used for boring on the milling machine as well as on the high-precision jig bore.

Whenever a hole must be located with reasonable accuracy but not high precision, as long as a boring tool can be inserted into the hole, it can be drilled somewhat undersized and brought to final dimensions by single-point boring. Nevertheless, the initial hole cannot deviate too much from the nominal size.

Under the best of circumstances, a drill will “drift” a little, especially in deeper holes, since the flutes of the drill can become partially clogged, causing the drill to deflect a little to one side or another. In this case, the hole can be bored to be sure of ending up with a round, straight hole.

## The Starter Hole

The first step in drilling a starter hole to permit boring a hole to an acceptable level of accuracy is to start with a drill that is very accurately ground.

The center point of the drill, for example, must be exactly at the center of the drill, and the slope of the leading cutting edges must be at exactly the same angle to create an acceptable initial hole. Since drills are typically made in high-production lots that provide acceptable accuracy for most uses, they may leave something to be desired when used for this operation—check each drill to be used for this purpose, “cleaning them up” if necessary.

## The Boring Bar and Dovetail Offset Boring Chuck

The boring tools most frequently used are a solid boring bar and a dovetail offset boring chuck.

Solid **boring bars** are the most rigid and are particularly useful when boring deep holes. The solid boring bar has an adjusting screw for advancing the bar short distances. This tool demands a high level of operator skill when checking measurements; a “go, no go” positive reading gage greatly simplifies the process and helps ensure satisfactory results.

The **dovetail offset boring chuck** is more commonly used. It can be moved outward 90 degrees from the axis of the spindle because of the dovetail slide. Adjustment of tool movement can be held to within 0.001 in. Some dovetail slide boring tools are built so that if the collar is held while the tool is turning, the slide will automatically move outward to a set dimension; this permits an operator to counterbore or to face the part being machined, and it also allows undercutting to be done, if necessary.

Some dovetail boring chucks permit tapers to be milled in a hole, a very useful capability when boring die holes that are to permit slugs to drop out after punching. For this type of boring, however, the boring tool taper and the machine

# MILLING FEEDS AND SPEEDS

## End Milling

Materials	Depth of Cut in.	Speed fpm	High Speed Steel in./tooth				Carbide Tool in./tooth				
			Cutter Diameter				Cutter Diameter				
			$\frac{3}{8}$ in.	$\frac{1}{2}$ in.	$\frac{3}{4}$ in.	1-2 in.	$\frac{3}{8}$ in.	$\frac{1}{2}$ in.	$\frac{3}{4}$ in.	1-2 in.	
Carbon Steels 1005-1029	.020	210	.001	.002	.004	.005	520	.002	.004	.006	.008
Medium Carbon 1030-1055	.020	160	.001	.002	.003	.004	485	.002	.003	.005	.006
High Carbon 1060-1095	.020	105	.001	.002	.003	.004	424	.001	.002	.004	.005
Alloy Steels 1340-1345 4140-4340	.020	165	.001	.002	.003	.004	390	.001	.002	.004	.005
Alloy Steels Hi Carbon 50100-51100 52100-m-50	.020	100	.001	.002	.003	.004	390	.001	.002	.004	.005
Tool Steels Hi-Speed M1 M2 M6 M10 T1 T2 T6	.020	65	.0007	.002	.003	.004	320	.001	.002	.003	.004
Tool Steels Hi-Speed T-15	.020	60	.0005	.001	.0015	.002	225	.001	.0015	.002	.003
Stainless Steels 201 202 301 302 303 304 405 409	.020	110	.001	.002	.004	.005	360	.001	.0015	.002	.004
Aluminum Alloys 1060-7178	.020	800	.003	.004	.005	.007	1300	.003	.004	.005	.007

downfeed have to be calculated to get the correct taper desired. That may be difficult or impossible on many of the milling machines in use today.

## Boring Procedure

The procedure for boring on a milling machine is as follows.

1. Drill the pilot hole. Then, without changing the location of the spindle relative to the part, take the drill out and insert the boring chuck.
2. Mount the cutting tool in the chuck, and adjust it for the first cut, removing about 0.010–0.015 in. to “clean up” the hole.
3. Check the size of the hole with a telescopic gage. Determine how much is left for boring the hole to finish size.
4. Remove only about 0.002 in. for stock removal for each bore until the final cut.
5. Finish the hole, taking, on this last cut, just 0.001 in. of stock. The hole will then be perfectly round and straight.

Use of measuring rods and digital readouts can greatly enhance the accuracy of the milling machine, permitting a very high order of locating accuracy—for a milling machine. But while this brings the milling machine closer to the range of high-precision accuracy provided by the jig bore machine, it only approaches it at the lower end.

Among other considerations, the leadscrew on a milling machine lacks the accuracy of the one used on a jig bore, so while the milling machine can handle a lot of routine loose to moderately accurate tolerances, do not use it to do jig bore work.

The milling machine is an accurate, reliable machine tool when utilized within its design and operating limits; it is not—and cannot be made into—a jig bore machine.

## MILLING MACHINE CHATTER

The following are suggestions when encountering chatter while milling:

- \* Reduce the stock removal.
- \* Reduce the width of cut
- \* Try heavier feeds is presently below 0.010.
- \* With high-speed steel cutters, try reducing the RPM.
- \* With high-speed steel cutters, try dulling the cutting edges.
- \* In end mill cutting, avoid high wraparound, generate the interior radius rather than plunging.
- \* Try serrated-edge end mills.
- \* Use a cutter with half as many teeth.
- \* Remove half of the inserts when using a face mill.

## SFMP/RPM/MILL DIAMETER TABLE

Surface Feet Per Minute	20	30	40	50	60	70	80	90	100	125	150	200	300	400	500	600	800	1000
Diameter																		
Inches	REVOLUTIONS PER MINUTE																	
$\frac{1}{16}$	1222	1833	2445	3056	3667	4278	4889	5500	6112	7639	9167							
$\frac{1}{8}$	611	917	1222	1528	1833	2139	2445	2750	3056	3820	4584	6112	9167					
$\frac{3}{16}$	407	611	815	1019	1222	1426	1630	1833	2037	2546	3056	4074	6112	8149				
$\frac{1}{4}$	306	458	611	764	917	1070	1222	1375	1528	1910	2292	3056	4584	6112	7639	9167		
$\frac{5}{16}$	244	367	489	611	733	856	978	1100	1222	1528	1833	2445	3667	4889	6112	7334	9778	
$\frac{3}{8}$	204	306	407	509	611	713	815	917	1019	1273	1528	2037	3056	4074	5093	6112	8149	
$\frac{7}{16}$	175	202	349	437	524	611	698	786	873	1091	1310	1746	2619	3492	4365	5238	6985	8731
$\frac{1}{2}$	153	229	306	382	458	535	611	688	764	955	1146	1528	2292	3056	3820	4584	6112	7639
$\frac{5}{8}$	122	183	244	306	367	428	489	550	611	764	917	1222	1833	2445	3056	3667	4889	6112
$\frac{3}{4}$	102	153	204	255	306	357	407	458	509	637	764	1019	1528	2037	2546	3056	4074	5093
$\frac{7}{8}$	87	131	175	218	262	306	349	393	437	546	655	873	1310	1746	2183	2619	3492	4365
1	76	115	153	191	229	267	306	344	382	477	573	764	1146	1528	1910	2292	3056	3820
$1\frac{1}{4}$	61	92	122	153	183	214	244	275	306	382	458	611	917	1222	1528	1833	2445	3056
$1\frac{3}{8}$	56	83	111	139	167	194	222	250	278	347	417	556	833	1111	1389	1667	2222	2778
$1\frac{1}{2}$	51	76	102	127	153	178	204	229	255	318	382	509	764	1019	1273	1528	2037	2546
$1\frac{5}{8}$	47	71	94	118	141	165	188	212	235	294	353	470	705	940	1175	1410	1880	2351
$1\frac{3}{4}$	44	66	87	109	131	153	175	196	218	273	327	437	655	873	1091	1310	1746	2183
$1\frac{7}{8}$	41	61	82	102	122	143	163	183	204	255	306	407	611	815	1019	1222	1630	2037
2	38	57	76	96	115	134	153	172	191	239	286	382	573	764	955	1146	1528	1910

## MILLING MACHINES

### Horizontal and Vertical Maching Centers Computer Controlled

The greatest contribution computers have made to machining occurred in application to milling machines. As is true of other machine tools, we must again emphasize the point that the basic machine is still the most important one for it permits making the first piece. What the computer permits is to allow the milling machine to become a unequalled production machine. We will show you how it can machine slots, forms, trapezoids, turn, manipulate parts to be made, move gigantic pieces to form, drill, tap, bore and perform just about any machining operation without the operator manually change tooling and workpiece position. Moreover, advancements continue to be made, dramatically improving its machining capabilities and cost effectiveness.

Starting with the basic machine shown with all of its attachments, the computer has been added to allow the machinist to make the first part manually and simultaneously digitize the part on the computer after the machinist has determined the part is ready to go. This allows several extra parts to be made for the prototype.

As is the case for horizontal turning/machining centers depicted in the preceding chapter, the horizontal milling machine-type machining center has the tool facing horizontally to the workpiece. This translates into less restrictions on workpiece height. Mounting costs may be higher because the workpiece must be fastened to a vertical fixture. The vertical milling-machine-type machining center affords greater flexibility with wide, plate-type work.

The need for faster production has constantly led to the manufacture of more efficient and higher speed machine tools, with engineers designing equipment to handle special and specific as well as production needs of industry. These more versatile and powerful machines in turn required higher speeds and better cutting tools to meet these needs. Increases in metal removal, reduction in secondary



*Horizontal Turning Center, left, courtesy of Yang Machine Tool Co.  
Vertical Machining Center, right, courtesy of Bridgeport Machines, Inc.*

“**MACHINING CENTERS**” can take a wide variety of forms, including designs that are marked departures from conventional ones, such as this five-axis Tricept 805 machining center by Neos Robotics AB, Taby, Sweden.

**More sophisticated** versions of machining centers, complete with automated materials handling and QC on-line control and perhaps additional machining stations are known as Flexible Machining Systems [FMS].



operations, and better finishes were among the things that had to be accomplished. The machine tools built to do all this were given higher rigidity, while new terms were developed to describe the more varied motions the machines provide. Instead of the old description of the movement (axis) of the table on the milling machine, *longitudinal*, *transverse*, and *vertical*, they became  $x$ ,  $y$ , and  $z$ . Machines with a fourth and fifth axis are now common. The number of axes a machine possesses determines how readily the cutting tool can reach various sections of the workpiece in one setup. The  $x$ ,  $y$ , and  $z$  axis points are located by its distances from two intersecting points which are often perpendicular, lines in the same plane: each distance is measured along a parallel to the other line.

The other two axes,  $b$  and  $c$ , are a rotation of the  $x$  and  $y$  axes.

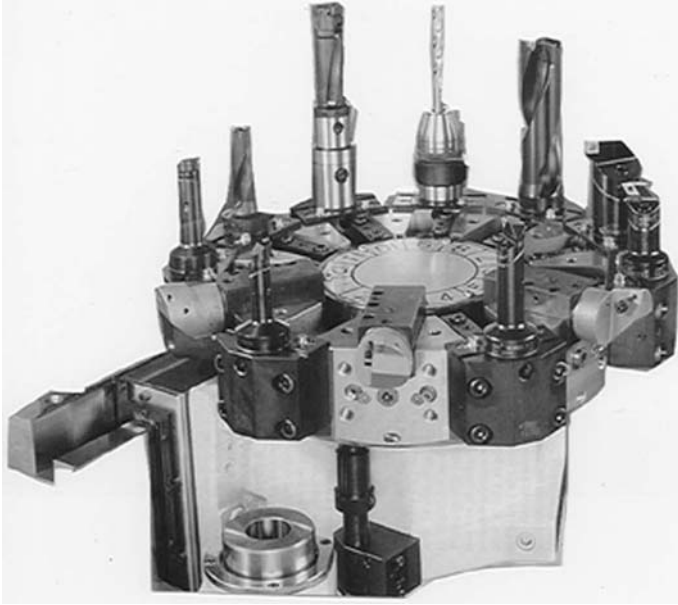
When a sixth and additional axis is needed that is applied by moving the workpiece pallet or table.

To allow the machine to accurately respond to these complex demands, the computer with its fast, efficient electronic control comes into play. This readily accommodates all of the functions mentioned above—milling, boring, turning, drilling, and grinding—without being required to move the part from one machine to another.

In addition to imposing less restriction on vertical workpiece height, a horizontal milling machine permits efficient chip removal while also making it easier to work with large, complex parts than is true of vertical machines.

On vertical machines the workpiece is held to the horizontal bed, which is more stable than the horizontal machine. For roughly comparable sized machines, this permits achieving greater accuracy because the part is usually more rigidly mounted. Workpieces can, however, be mounted as rigidly on the horizontal mill as on the vertical model, although it costs more to do so. Because of its design vertical machines are best for machining plate-type workpieces.





*MULTI-TOOL TOOL-CHANGER speeds the machining operation, permitting swift change over of the cutting tool for fast, efficient machining of complex shapes.*

For both types of machines the tool changers allow for automatic production without requiring handling by the operator.

The tools are first mounted in their holders and then set in the holders by precise gages so the machining of various slots, angles, contours, etc, will be accomplished accurately and cost-effectively.

## *Chapter 5*

# **SENSITIVE, GEAR-HEAD, AND RADIAL DRILL PRESSES**

## **INTRODUCTION**

Humans started making holes in things to bind them together and make them more useful as far back as the Stone Age, and we're still at it, with some three-fourths of all modern machining being concerned with making holes.

Although most metal-cutting machines can make holes, the drill press remains the basic machine for that purpose—the drill press can make just about any hole required, from drilling one 0.001 in. in diameter to boring one as large as 20 in. in diameter.

The types of drill presses most commonly used in the shop are sensitive, gear-head, and radial (Figure 5.1).

## **SENSITIVE DRILL PRESSES**

The sensitive drill press is manually operated and, as the name implies, ostensibly permits the operator to feel and thereby control the cutting action of the tool, sensing, for example, when the cutting tool is being overfed. It is particularly helpful when using very small diameter drills, since overfeeding can easily shatter them.

Because they are made for smaller diameters these machines are much smaller and provide speeds in higher ranges than other types of drill presses.

The cutting tools are driven by a belt and, depending on the machine, speeds are changed either by manually shifting the belt from one pulley vee to another or by a variable-drive control lever.



*Figure 5.1. Sensitive, gear-head, and radial drill presses. These are the three drill presses most commonly used in the shop: (a) the sensitive, (b) the gear-head, and (c) the radial. (Courtesy of DoAll.)*

Sensitive drill presses generally do not have power feed; they are hand-fed into the work by the operator.

Their drill size range is from micro-drills (under 0.013 in.) up to  $\frac{1}{2}$  in. They are usually bench models but can be floor-standing models for those in the upper ranges.

The cutting tools used on sensitive drill presses are usually straight shanks and are held by drill chucks (Figure 5.2). The miniature sizes require very accurate drill chucks for the drills to run true. Spindle runout for machines on which micro-drills are to be used must be held to extremely tight tolerances to ensure drill entry accuracy.



*Figure 5.2. Drill chuck and key. (Courtesy of DoAll.)*

A tapered shank can be used for many sizes.

The larger-size sensitive drill press typically only needs to hold “commercial” tolerances. The  $\frac{1}{2}$ -in.-capacity drill press is modest in price and is very handy for light tapping and for operations calling for smaller-size drills and reamers.

## **GEAR-HEAD DRILL PRESSES**

These are very popular types of drill presses. They are powerful, capable of heavy holemaking, and can be used for drilling, boring, hand and production tapping, and reaming.

Although some gear-head drill presses are provided with hand feed only, most are power fed and have three basic feeds for drilling and boring holes. Feeds are similar to those found on the spindle of a milling machine.

Do not confuse the drill press with the milling machine. The drill press is built to take thrust vertically—force exerted in an upward direction—permitting power exerted on the head to be utilized to the fullest extent. It is not constructed to take the same thrust horizontally that a milling machine can; therefore drilling is its main function. Conversely, the milling machine is designed to withstand horizontal thrust, so only light drilling on it is recommended; if heavy stock removal is necessary, use a drill press.

## RADIAL DRILL PRESSES

Radial drill presses in smaller sizes have become very popular in shops because they are less costly, take up less room, and are more manageable than typical earlier models.

The advantage of a radial press over the sensitive and gear-head machines is that it permits clamping the work to the table and moving the tool to the required location whereas with sensitive and gear-head machines it is necessary to move the work to the tool and then hold it to the table in some manner.

Radial drill presses come in all sizes, from ones matching the smallest gear-head size to ones comparable to very large boring mills.

With a radial arm that swings around a vertical column, rotation can be through a full 360 degrees, whereas on larger machines, reach can extend anywhere from 2 to 12 ft. The vertical column is proportionate to the length of the radial arm and the size of the drill head; the drill head slides along the arm for positioning. The arm also travels vertically on the column and is positioned by means of a power system.

Larger radial drill presses are grouted and bolted into the floor for stability and rigidity. Smaller models are mounted on very rigid cast iron bases and are only bolted to the floor when that is necessary for rigidity. Although it is similar in operation to the larger models, the radial arm reach is only about 18 in.

Drill heads come in a variety of sizes and horsepower, with some in the larger sizes capable of drilling holes up to 3 in. in diameter in solid steel.

Both large-diameter thread tapping and the boring of large holes is readily accomplished on these machines.

## DRILLS

Drill bit cutting tools are used to originate or enlarge a hole in solid material. On a drill press, the drill is rotated into stationary work; on a lathe, the work is generally rotated while the drill is held stationary. Drills are end cutting tools with flutes leading up to the cutting edges to help remove the chips.

Drills are classified in a number of ways—by material, method of manufacture, length, shape of the point, flute configuration, and size.

Tool selection starts with the size of the hole or holes to be machined. The most popular type of drill for holes up to  $\frac{1}{2}$  in. has a straight shank, comes in sets, and is mainly used on sensitive drill presses. Typical commercially available sets are:

- \* *Number sets*, 1–60 and 61–80. The 1–60 set ranges from size from 0.040 in. to 0.228 in. The 61–80 set, fewer in number, range in size from 0.0135 in. to 0.039 in. These two number sets cover the decimal-equivalent hole sizes most often required in a machine shop.
- \* *Letter set*, A to Z. This drill set covers the size range from 0.234 in. to 0.413 in. Note that the size range starts just above the largest numbered drill and ends below  $\frac{1}{2}$  in.
- \* *Fractional set*. The size range for this set is  $\frac{1}{16}$ – $1\frac{1}{2}$  in. increments of 0.064 in. These three groups of standard drill sets cover most drilling requirements.

For jobs that call for hole sizes not matched by drills in one of these sets, special drills must be used.

Although the drills with Morse tapered shanks in sizes up to  $\frac{1}{2}$  in. are available, they are not widely used because they cost more and are more difficult to store than straight-shank tools. Tapered-shank drills are, however, the usual choice for sizes over  $\frac{1}{2}$  in. in diameter.

The tapered shank used on a specific drill is proportional to the diameter of the drill. For example:

- $\frac{1}{8}$ – $\frac{15}{32}$  in.: use #1 Morse taper.
- $\frac{31}{64}$ – $\frac{25}{32}$  in.: use #2 Morse taper.
- $\frac{51}{64}$ – $1\frac{1}{16}$  in.: use #3 Morse taper.
- $1\frac{3}{32}$ – $1\frac{1}{2}$  in.: use #4 Morse taper.
- $1\frac{9}{16}$ –3 in.: use #5 Morse taper.
- $3\frac{1}{16}$ – $3\frac{1}{2}$  in.: use #6 Morse taper.

Tapered-shank drills are only fractional and come in  $\frac{1}{64}$ -in. increments.

Although there are a few straight shank drills, by convention, all drills are considered to be “twist drills,” a carryover from the days when all drills were made that way.

Today, drills up to  $\frac{1}{2}$  in. in diameter normally are ground from solid hardened high-speed material, whereas drills in sizes of  $\frac{1}{2}$  in. and above usually are rolled under heat to the helix angle required.

Drills larger than  $\frac{1}{2}$  in. are milled and then twisted to the helix angle required; finally the shank is butt-welded to the drill. After the shank is attached, the Morse taper is machined and the remainder of the flute to the shank is milled.

All drill points are ground to their individual specifications after the drill has been made.

The helix angles of the flutes that are formed to each of the drills have a specific purpose. General-purpose drills have a helix angle of 30 degrees with a lip angle of 12 degrees.

When machining aluminum, copper, hard brass, and soft steel, a higher helix angle to help clear chips from the hole is desirable. Drills with high helix flutes—up to 40 degrees—are particularly useful for deep hole drilling. These are fast spiral drills.

For drilling plastics and soft brass, drills with a low helix angle are very successful. The gentle helix angle prevents the drill from grabbing in the work and pulling itself in.

Drill point angles are carefully selected for the types of material to be drilled. Standard drills used for general drilling are manufactured with a 118-degree included angle and a 12-degree lip angle (Figure 5.3). For most drilling tasks, this drill is satisfactory. For hard materials, however, a lip angle of about 6 degrees is used to give strength to the cutting edge. For softer materials, the lip angle should be increased—the softer the material, the greater the lip angle.

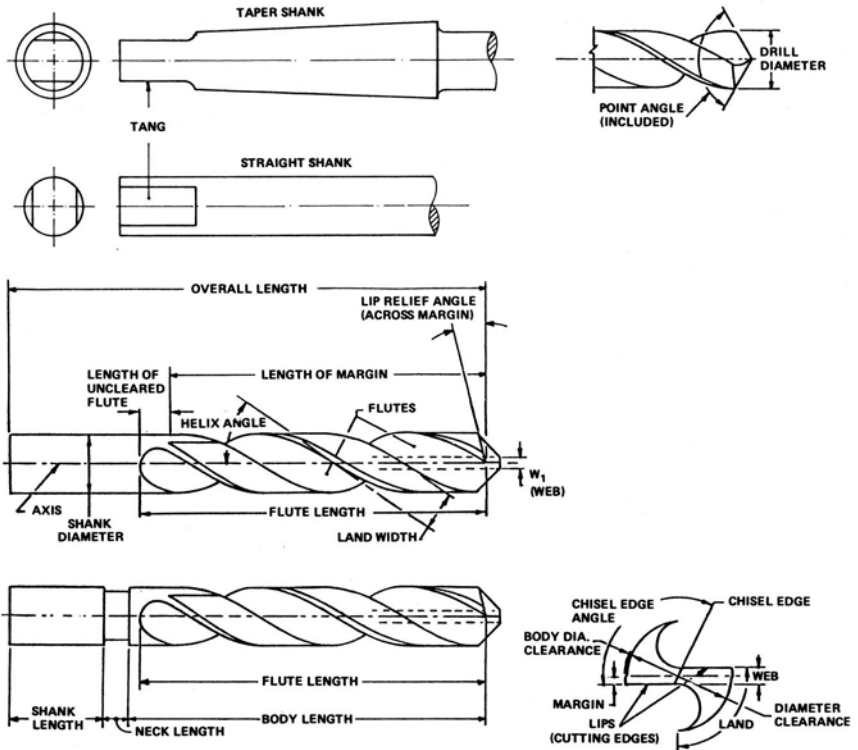


Figure 5.3. Drill angles. (Courtesy of DoAll.)

When great pressure is applied to a drill, as when drilling stainless steel, the recommended included angle is 135 degrees, with a slight lip angle. For wood or plastic, the included angle is long, about 90 degrees, with a lip angle of about 25 degrees.

The web on a drill, the portion of the drill between the flutes, normally becomes thicker toward the bottom of the drill (the shank) to give added strength. When a drill becomes worn, it must be resharpened, accomplished by removing all of the worn section and grinding a new point. In many cases the web must be thinned.

The procedure for grinding by hand is as follows.

1. **Use safety glasses.** Turn on the pedestal grinder and, using the rough wheel, grind away the worn part of the drill. Do not use too much pressure on the wheel. This can generate too much heat and cause the drill to lose its hardness.
2. If necessary, thin the web. The edge of the grinding wheel can be used to grind in between the cutting edges. Do not remove too much.
3. Have a drill gage handy. The correct angle will be a 118-degree included angle. Each cutting edge angle will then be 59 degrees. The drill gage permits measuring the length of the flutes. They must be the correct

angle and exactly the same length. The gage will also allow measurement of the lip angle; in this case, it will be 12 degrees.

4. When grinding free-hand, hold the drill firmly with the right hand (if you are left-handed, do the reverse) and put one flute up to the wheel for grinding. As you are grinding, sweep the drill up in a radial motion using your left hand to guide it for grinding the lip clearance. Now do the other cutting edge the same way.
5. Using the drill gage, check for the 59-degree angle on each cutting edge and check each for correct length to ensure even, balanced sharpening of the cutting edges. If everything checks out, make sure the lip relief angle is within 12 degrees as recommended.
6. Use light pressure when grinding and keep the tool cool during the grinding procedure by dipping the drill in water frequently.

If a drill-sharpening machine is available, use it; that is the recommended way of sharpening a drill. Angles will be accurate, flutes will be equal, and, if necessary, there usually is some provision of web thinning.

The roster of drills is virtually endless, but among the more commonly used types of drills available for various applications are:

- \* Plastic drills
- \* Brass drills
- \* Center drills
- \* Starting drills
- \* Straight flute drills
- \* Core drills
- \* Oil hole drills
- \* Drill and countersink drills
- \* Half-round drills
- \* Step drills
- \* Subland drills
- \* Cobalt drills
- \* Carbide drills
- \* Gun drills

The following are characteristics of the various drills:

**Twist Drills** These are the most common. Most are made with two helical flutes that allow the removal of chips and admission of cutting fluid.

**Application** Used for the majority of clearance holes for fasteners and pilot holes for reaming or Tapping.

**Step Drills** have two or more diameters. The steps or different diameters are ground on the same Land.

**Application** Used to produce two or more diameters in one pass. Can be used as combined drills and countersinks, combined drills and counterbores, and combined drills and reamers.



**Subland Drills** They are combination tools having separate lands or margins that extend the full length of the flutes for each of the two or more diameters.

**Application** Two or more diameters can be drilled simultaneously. They can be ground to drill and countersink simultaneously, drilling and countersinking for flathead screw, drilling and countersinking for socket head screws. Two or more diameters can be maintained throughout the life of the drill.

**Flat Drills** These are flat rather than the normal round configuration. The flats are usually not parallel this creates a web that is thicker at the shank.

**Application** These are occasionally used for drilling hard forging and castings.

**Spade Drills** have a removable tip in a special holder attached on the drill shank. Many have a hole through the shank to allow cutting fluid to be applied to the drill point under pressure.

**Application** Drills under  $1\frac{3}{4}$ " should be only be used for shallow holes, one diameter deep [vertical], two diameters deep (horizontal). Do not use a center drill for starting holes. Use a short spade drill with the same point angle as the drill to be started.

**Oil-Hole Drills** take the form of either a twist drill or a straight fluted drill that has one or more continuous holes through its body and shank to permit passage of cutting fluid under pressure. These holes can be through either the lands or the web of the drill.

**Application** for deep holes where flood applications of cutting fluids would be ineffective. Increased metal removal rates are possible in both normal and deep holes by using oil-hole drills instead of flood cooling. (When larger deep holes such as 2 up to  $\frac{4}{7}$  inches are required, trepanning becomes more practical. This head consists of a heat-treated alloy-steel body with  $\frac{3}{16}$ -in-wide carbide tipped replaceable blade and two carbide-tipped replaceable wear pads. Both the blades and the pads are held in place mechanically. Cutting fluid is entered through a guide bushing and around the trepanning head to flush out the chips.)

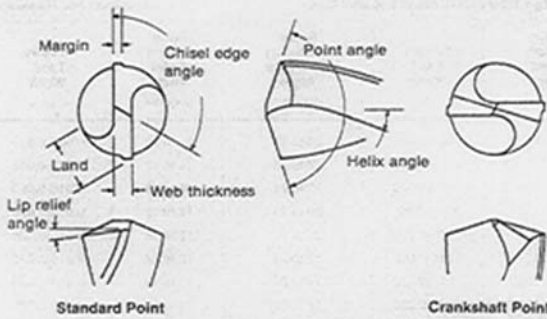
**Gun Drills** are used for producing deep holes. They have a straight fluted tip, cutting to center mounted in a tubing shank. The tips are often brazed to the shanks, but in larger sizes the tops are keyed or pinned to the shanks for convenience in removal for regrinding. Carbide pads are used for guiding the tips as shown in the sketch.

**Application** Used for making very deep holes that require extremely close tolerance in straightness and diameter.

## DEEP HOLE DRILLING: GUN DRILLING, SPADE DRILLING, TREPANNING

Gun drilling, along with spade drilling and trepanning, falls into the category of deep hole drilling. A twist drill is considered to be deep hole drilling when the depth of the hole is three times the diameter of the drill. When that is the case,

**Drills, High Speed Steel Twist**  
**TOOL GEOMETRY**



MATERIAL	HARDNESS Bhn	DRILL TYPE	POINT ANGLE* degrees	LIP RELIEF ANGLE* degrees	HELIX ANGLE	POINT GRIND
Free Machining Carbon Steels-Wrought Carbon Steels-Wrought, Cast and P/M Free Machining Alloy Steels-Wrought Alloy Steels-Wrought, Cast and P/M Machining Steels-Wrought Tool Steels-Wrought, Cast and P/M Nitriding Steels-Wrought Armor Plate-Wrought Structural Steels-Wrought	85-225	General Purpose	118	A	Standard	Standard
	225-325	General Purpose	118	A	Standard	Standard
	325-425	General Purpose	118 to 135	B	Standard	Crankshaft
	45R <sub>c</sub> -52R <sub>c</sub>	Heavy Web	118 to 135	B	Standard	Crankshaft
High Strength Steels-Wrought	175-325 325-52R <sub>c</sub>	Heavy Web	118 118 to 135	A A	Standard Standard	Crankshaft
Austenitic Manganese Steels-Cast	150-220	Rail Drill	135	B	Low	Split
Free Machining Stainless Steels-Wrought	135-425	General Purpose	118	A	Standard	Crankshaft
Stainless Steels, Ferritic-Wrought and Cast Stainless Steels, Austenitic-Wrought, Cast and P/M Stainless Steels, Martensitic-Wrought, Cast and P/M Precipitation Hardening Stainless Steels-Wrought and Cast	135-200	General Purpose	118 to 135	A	Standard	Standard
	200-325	General Purpose	118 to 135	A	Standard	Crankshaft
	325-425 48R <sub>c</sub> -52R <sub>c</sub>	Heavy Web	118 to 135	B	Standard	Crankshaft
Gray Cast Irons Ductile Cast Irons Malleable Cast Irons Compacted Graphite Cast Iron White Cast Iron	110-220	General Purpose	118	A	Standard	Standard
	220-400	Heavy Web	118	A	Standard	Standard

NOTE: Use stub-length drills whenever possible on high strength materials.

\*Chisel edge angle: 115° to 135°

†See chart at end of table on next page.

*Drill Speeds and Feeds.*

the drilling is usually conducted by periodically retracting the drill from the hole to clear the chips. Oil hole drills are used to apply coolant while drilling to help clear the chips. The amount of coolant applied normally affects and limits the depth of the hole that can be safely drilled.

A conventional gun drill consists of a single-lip straight-flute drill attached to a tube through which coolant is applied with sufficient force to flush the chips

**Drills, High Speed Steel Twist**  
**TOOL GEOMETRY**

MATERIAL	HARDNESS Bhn	DRILL TYPE	POINT ANGLE* degrees	LIP RELIEF ANGLE* degrees	HELIX ANGLE	POINT GRIND
Aluminum Alloys-Wrought and Cast	30-150 500 kg	Polished Flutes	90 to 118	C	High	Standard
Magnesium Alloys-Wrought and Cast	40-90 500 kg	Polished Flutes	70 to 118	C	High	Standard
Titanium Alloys-Wrought and Cast	110-275	General Purpose	118 to 135	B	Standard	Crankshaft
	275-440	Heavy Web	118 to 135	B	Standard	Crankshaft
Copper Alloys-Wrought, Cast and P/M	40-200 500 kg	Polished Flutes	118	C	Low	Standard
Nickel Alloys-Wrought, Cast and P/M Chromium-Nickel Alloys-Cast	80-360	General Purpose	118	B	Standard	Crankshaft
NiInco Alloys-Wrought	210-330 48R, 52R <sub>c</sub>	General Purpose	118	B	Standard	Crankshaft
High Temperature Alloys-Wrought and Cast	140-475	Heavy Web	118 to 135	B	Standard	Crankshaft
Columbium, Molybdenum and Tantalum Alloys-Wrought, Cast, P/M	170-290	General Purpose	118	B	Standard	Standard
Tungsten Alloys (Anviloy) <sup>†</sup>	290-320	General Purpose	118	B	Standard	Standard
Zinc Alloys-Cast	80-100	General Purpose	118	C	Standard	Standard
Uranium Alloys-Wrought <sup>‡</sup>	180-210	Special Carbide	118	5-8	20°	-5° Land on Drill Lip
Zirconium Alloys-Wrought	140-280	General Purpose	118	A	Standard	Crankshaft
Thermoplastics and Thermosetting Plastics <sup>§</sup>	All	Special Polished	60 to 90	C	Low	Standard
Magnetic Core Iron	165-240	General Purpose	100 to 118	A	High	Standard
Controlled Expansion Alloys	125-250	General Purpose	118	A	Standard	Crankshaft
Magnetic Alloys (Hi Perm 49, HyMu 80)	165-240	General Purpose	118	A	Standard	Crankshaft
Carbon and Graphite	8-100 Shore	General Purpose	90 to 118	C	Standard	Crankshaft
Mechinable Carbide (Ferro-Tic)	40R <sub>c</sub> -51R <sub>c</sub>	General Purpose	118	B	Standard	Crankshaft

NOTE: Use stub-length drills whenever possible on high strength materials.

\*Chisel edge angle: 116° to 135°

†For both hss and carbide drills.

‡For carbide drills.

§See following chart.

Lip Relief Angles at Periphery	Drill Size							
	#80 to #61	#60 to #41	#40 to #31	¼ to ⅜	½ to ¾	1 to 1½	2 to 3	1 Inch up
A	24°	21°	18°	16°	14°	12°	10°	8°
B	20°	18°	16°	14°	12°	10°	8°	7°
C	26°	24°	22°	20°	18°	16°	14°	12°

*Drill Speeds and Feeds.*

away from the cutting edges at the point of machining. The tube can be quite long for deep hole drilling; if needed, a steady rest can be applied for support. For larger holes, carbide wear pads can be added at 90 and 180 degrees from the single flute to guide the cutting edge. Coolant is applied at very high pressure to evacuate the chips. Gun drills range in size up to 2-inches and more as specials. The

**Oil-Hole or Pressurized-Coolant Drills, High Speed Steel**

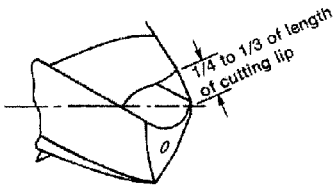
**TOOL GEOMETRY**

Drill geometry as shown in section 15.7 (HIGH SPEED STEEL TWIST DRILLS) may be used for Oil-Hole or Pressurized-Coolant Drilling.

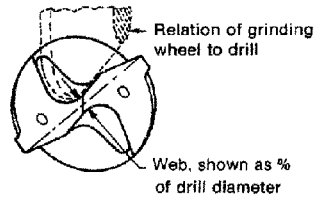
Drills used for Oil-Hole or Pressurized-Coolant Drilling operations, however, have heavy or semi-heavy webs which require thinning of the point. Instructions for thinning are shown below.

DRILL DIAMETER inches	WEB
Up to 1/4	12%
Over 1/4 to 1/2	10%
Over 1/2 to 3/4	8%
Over 3/4 to 1	8%
Over 1 to 1-1/4	7%
Over 1-1/4 to 1-1/2	7%

**NOTCH THINNING OF HEAVY WEB OIL-HOLE DRILLS**

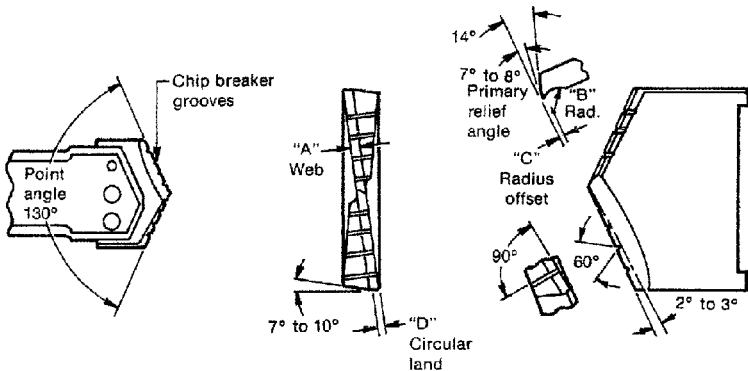


Side View



Front View

**Spade Drills, High Speed Steel**



*“Oil-Hole or Pressurized-Coolant Drills, High Speed Steel”; and “Spade Drills, High Speed Steel”.*

cutting edge is carbide; high strength materials can readily be machined when the correct feeds and speeds are used.

When larger deep holes such as 2 up to  $\frac{4}{Z}$  inches are required, trepanning becomes more practical. This head consists of a heat-treated alloy-steel body with a  $\frac{3}{4}$ -in-wide carbide tipped replaceable blade and two carbide-tipped replaceable wear pads. Both the blades and the pads are held in place mechanically. Cutting

### Spade Drills, High Speed Steel TOOL GEOMETRY

DRILL SIZE RANGE Inches	D I M E N S I O N S	MATERIAL*						
		STEELS, Wrought and Cast and CAST IRONS 100 to 175 Bhn Inch	STEELS, Wrought and Cast and CAST IRONS 175 to 330 Bhn Inch	STEELS, Wrought and Cast 330 Bhn to 52R <sub>c</sub> Inch	STAINLESS STEELS, Wrought and Cast Inch	ALUMINUM and MAGNESIUM ALLOYS, Wrought and Cast Inch	TITANIUM ALLOYS, Wrought and Cast Inch	COPPER ALLOYS, Wrought and Cast Inch
1-1/32 to 1-1/4	A	1/16	5/64	5/64	1/16	5/64	1/16	5/64
	B	3/16	1/4	1/4	3/16	3/16	1/64 and 10° Rake	5/32
	C	0 to 1/32	0 to 1/32	0 to 1/32	3/64	3/64	1/32	5/64
	D	1/16	1/16	1/16	3/64	1/16	1/16	1/16
1-9/32 to 1-1/2	A	5/64	3/32	5/64	5/64	3/32	5/64	3/32
	B	7/32	9/32	1/4	7/32	1/4	1/64 and 10° Rake	5/32
	C	0 to 1/32	0 to 1/32	0 to 1/32	3/64	1/16	1/32	5/64
	D	5/64	5/64	5/64	3/64	1/16	5/64	5/64
1-5/16 to 2	A	3/32	7/64	3/32	5/64	3/32	3/32	7/64
	B	1/4	5/16	5/16	1/4	5/16	1/64 and 10° Rake	3/16
	C	0 to 1/32	0 to 1/32	0 to 1/32	1/16	5/64	1/32	1/16
	D	5/64	5/64	5/64	1/16	1/16	1/16	5/64
2-1/16 to 2-1/2	A	7/64	1/8	7/64	7/64	7/64	7/64	9/64
	B	9/32	11/32	3/8	9/32	3/8	1/64 and 10° Rake	7/32
	C	0 to 1/32	0 to 1/32	0 to 1/32	1/16	5/32	1/16	7/64
	D	5/64	3/32	5/64	1/16	5/64	5/64	5/64
2-9/16 to 3	A	1/8	9/64	1/8	1/8	7/32	5/32	11/64
	B	9/32	3/8	3/8	5/16	1/2	1/64 and 10° Rake	1/4
	C	0 to 1/32	0 to 1/32	0 to 1/32	5/64	9/32	1/8	7/64
	D	3/32	3/32	3/32	5/64	3/32	3/32	3/32
3-1/16 to 3-1/2	A	5/32	11/64	5/32	5/32	-	-	13/64
	B	5/16	3/8	3/8	11/32	-	-	9/32
	C	0 to 1/32	0 to 1/32	0 to 1/32	3/32	-	-	1/16
	D	7/64	7/64	7/64	5/64	-	-	7/64
3-9/16 to 4	A	11/64	3/16	11/64	11/64	-	-	15/64
	B	5/16	7/16	3/8	3/8	-	-	5/16
	C	0 to 1/32	0 to 1/32	0 to 1/32	7/64	-	-	1/16
	D	1/8	1/8	1/8	3/32	-	-	1/8
4-1/16 to 5	A	3/16	7/32	3/16	3/16	-	-	-
	B	3/8	1/2	1/2	7/16	-	-	-
	C	0 to 1/32	0 to 1/32	0 to 1/32	1/8	-	-	-
	D	5/32	5/32	5/32	3/32	-	-	-

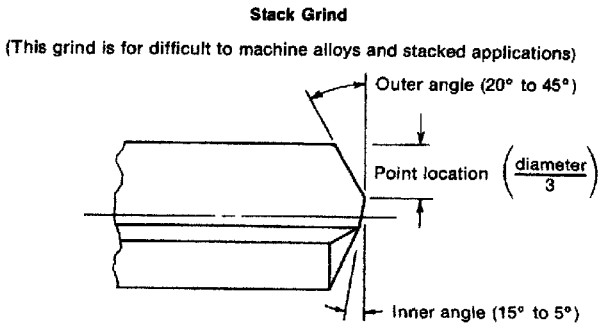
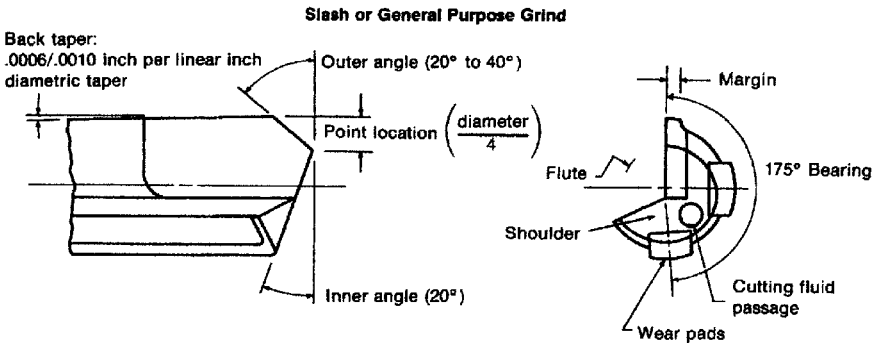
\*For other alloys use recommendations for steel as starting geometry.

### Spade Drills, High Speed Steel.

fluid is entered through a guide bushing and around the trepanning head to flush out the chips.

A spade drill is a high-speed flat drill that cuts on both side of its center point. It is ground with the precision of a gun drill and is mounted on high alloy holders. Large holes are possible to drill with coolant applied at the point of machining. The coolant is applied externally so feed and speeds can be critical when drilling deep. Pilot holes should be drilled first.

**Gun Drills, Carbide**  
**TOOL GEOMETRY**



*Gun Drills, Carbide.*

The following illustrations depict sketches of twist drill geometry, oil hole drills, spade drills and speeds, and gun drills.

Cutting fluids are indispensable for optimum performance and are essential for life of the tool; they should be used when drilling as well as for all other machining.

**REAMING**

Reaming is a sizing operation where the tool cuts slightly larger than its own diameter. The material is essentially scraped off and the amount that the reamer cuts oversize is in direct proportion to the amount of stock to be removed.

The rule of thumb for stock to be removed is:

Diameters under 0.500 in.	0.015 in.
Larger diameters	0.030 in.

For finishing reaming operations,

Diameters under 0.500 in.	0.005 in.
Larger diameters	0.015 in.

Too much stock for removal leaves oversized and rough holes.

## Rose Reamer

A rose reamer is a finishing reamer and has cutting edges only at the tip of the chamfer. It should only be used for finishing operations. The correct cutting speed for reamers is slower than for drilling, about 65–70% of drilling speed.

## Reamer Feeds

Reamer feeds should, however, be two to three times the feed of drilling. The feeds depend on the material being machined and the strength of the reamer. If chatter occurs, then adjust the cutting speed or increase the feed to eliminate it.

## Cutting Edges

Reamers are end-cutting tools, that is, they do not cut along the flute but instead cut on the chamfer, the angular cutting portion at the entering end of the reamer (Figure 5.4). The chamfer allows the reamer to enter into the work properly.



*Figure 5.4. Reamer, showing chamfered cutting edges. The reamers cut on the chamfer; the angled cutting edges cut at the front of the tool.*

## Chamfer Angle

The chamfer angle is 45 degrees. If a finer finish is required, then a lead or starting taper is employed. This lead chamfer is in back of the first one and has an

angle ranging from 1 to 10 degrees. The purpose of this edge is to help size and smooth out the walls of the hole.

## Reamer Flutes

There are several types of flutes provided with reamers. The first of these are straight flutes, most commonly provided with reamers because of cost. The others are right-hand spiral and left-hand spiral.

All flutes are for providing chip carrying and removal. Without sufficient clearance, chips get packed in the reamer and can cause deflection, oversized holes, and heat buildup.

Reamers usually have an even number of flutes, but there are some exceptions.

## Machine Condition

Reamers should be used with machines that are in good condition; machine spindles should be tight and in good running condition.

If reaming on a machine with excessive spindle wear, it is good practice to use a reamer with a left-hand spiral (helix angle of the flute). This requires a much heavier feed pressure when reaming, which helps stabilize the spindle and gives a much better finish to the hole being reamed.

## Types of Reamers

There are many types of reamers. They are discussed in the following subsections.

### Hand Reamers

These are used to finish or size previously reamed holes. They are also used for removing small amounts of metal when finishing such things as fitting bushings or bearings.

They are either the solid type or expansion. The expansion reamer is, however, made out of a carbon material because carbon expands easily.

There are limits to expansion, and the manufacturer's recommendations should be followed. A guide to follow is:

$\frac{1}{4}$ – $\frac{1}{2}$ in.	0.006 in.
$\frac{1}{2}$ –1 in.	0.010 in.
1 in. and above	0.013 in.

### Chucking Reamers

These are used in machines and are common in a machine shop. They are available in sizes corresponding to drill sizes in the numbered, letter, and fractional set sizes. They also come in undersize and oversize sets both 0.001 in. under and over the nominal size.

Frequently used by tool and diemakers, these are available in straight-shank, or left- or right-hand spiral flutes.



## Tapered Reamers

There are several types of tapered reamers, namely, Morse taper, Brown & Sharpe taper, and taper pin reamers.

The Morse and B & S taper reamers are both finishing reamers and heavy stock removal tools. By the end of their cuts, the flutes are fully engaged. As a result the reamer must be structurally sound, with flutes fully supporting the cutting edges.

The taper reamer has fewer flutes, and these are usually straight. If a spiral flute is used, a left-hand spiral is found to be most efficient. Most suppliers, however, do not carry spiral-fluted taper reamers.

The taper pin reamer is different because they ream holes, usually deep, of small diameters. Because of the nature of the holes to be reamed, helical (spiral) taper pin reamers do the best job and are the only ones available through suppliers. The helix angle is about 45 degrees and is of the opposite direction of rotation for cutting. Therefore the torque is less, even though more end pressure is necessary.

Hand reaming of left-hand spirals is not satisfactory because of the heavy pressure needed to penetrate.

## Dowel Pin Sizes

It is worth mentioning here that dowel pin-size reamers are very popular. Although they are the same as chucking reamers, their sizes are under the nominal size of regular chucking reamers. They differ from under- and oversized reamers in that they are all undersized and range from 0.123 in. to 0.4995 in.

When dowels are pressed into fixture and die locating holes, they are securely retained.

## Adjustable Reamers

Do not confuse these with expansion hand reamers. They are chucking reamers and cut on the end of the tool.

When dull, they are expanded to the nominal size and resharpened.

They come in straight or tapered shank and their flute length is relatively short.

If used to enlarge a hole, this reamer has limited range and should only be used to remove light amounts of material; otherwise, it can produce a poorly finished hole.

## Other Types of Reamers

These include stub screw machine reamers, core reamers, combination drill and reamer, deburring reamers, and structural reamers. Structural reamers are seldom found in a machine shop.

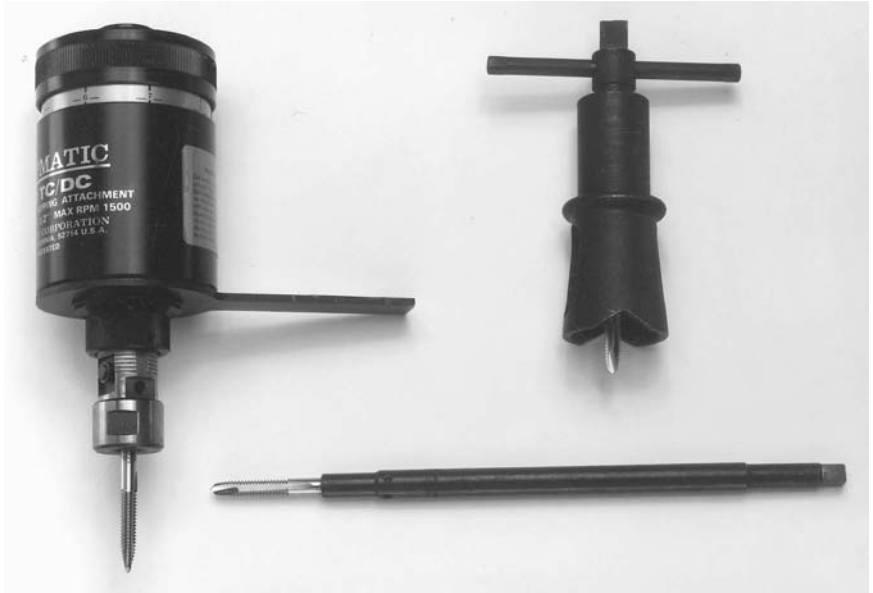
## Sharpening

Reamers should *never* be sharpened by hand—accuracy and finish will suffer if attempted. Cutting fluids foster optimum results and should be used.

## TAPPING

*Tapping*, defined as “the machining of threads in various parts,” can be accomplished on most machines in a machine shop.

Lathes can cut a thread, and milling machines can tap or thread mill, but drill presses that are equipped with a special tapping head that allows for the lead and devoted solely to tapping are very popular for this operation (Figure 5.5).



*Figure 5.5. Tapping. Most machines in the shop can make threads, but drill presses with special tapping heads are popular for this purpose.*

## TAPS

*Taps* are tools designed to cut screw threads in a part. There are many methods for making screw threads, but, in a machine shop where such one-of-a-kind parts as a jig, a fixture, or a die is made or when a part is to be replaced or repaired, the machinist normally uses hand taps and dies.

It is important to understand the various types of taps and their uses.

### Two Basic Tap Types

There are two basic types of taps made, the ground thread and the cut thread. The ground thread is initially more expensive but is much more accurate and it will last as much as six times longer than a cut thread. Therefore it results in lower overall cost and is the one most often used.

## Tolerances

Because of the nature of their respective manufacturing methods, the tolerances of the two types of threads are quite different.

A cut thread, which is turned to size before hardening, has a lead tolerance of  $\pm 0.003$  in., whereas a ground thread that is finished after hardening has a lead tolerance of  $\pm 0.0005$  in.

Pitch diameter tolerances of the cut thread are from 0.001 in. to 0.0055 in. whereas the ground thread has a tolerance of only 0.0005 in. to 0.0025 in.

A tap must be made to exacting specifications. For example, the lead must be accurate or else the thread of a part will not bear evenly with screw going into it, thus weakening the structure of the fitting. *Lead* is defined as “the distance the screw thread advances axially in one full revolution or turn.”

Even though the machine may run true if the tap isn't accurately made, it can run out and cause a poor thread.

There are many styles of taps (Figure 5.6). They are discussed in the following subsections.

## Hand Taps

Hand taps are taps in which the thread and shank have the same length. The shank has a square on the bottom to permit applying a wrench to it.

Although they were originally used for hand tapping, these tools are now used as much for machine tapping as for hand tapping.

The dimensions of the three styles of taps are identical; the only difference is the chamfer at the beginning of the tap.

The taper tap has a chamfer the length of 8–10 threads, the plug tap has a chamfer of 3–5 threads, and the bottom tap has a chamfer of  $1-1\frac{1}{2}$  threads. Hand taps come in two-flute variety, up to  $\frac{5}{16}$ -in. diameter; three-flute variety, up to  $\frac{1}{2}$ -in. diameter; and four-flute from  $\frac{1}{4}$ -in. to  $\frac{1}{2}$ -in. diameter.

## Spiral-Pointed Taps

These taps are normally used for production in machines and are not for hand tapping.

The spiral point shears the material and forces the chip ahead. Because of this chip action, this tap is recommended for through holes.

If a blind hole is to be tapped, then spiral pointed bottoming taps are available. When a bottoming tap is used, the blind hole must be deep enough to hold the chips being driven forward.

Spiral taps can be run at higher speeds and with less power; because of their design, they have fewer flutes, thereby making them stronger tools.

Spiral-pointed taps are available only in two- and three-flute designs (Figure 5.7).

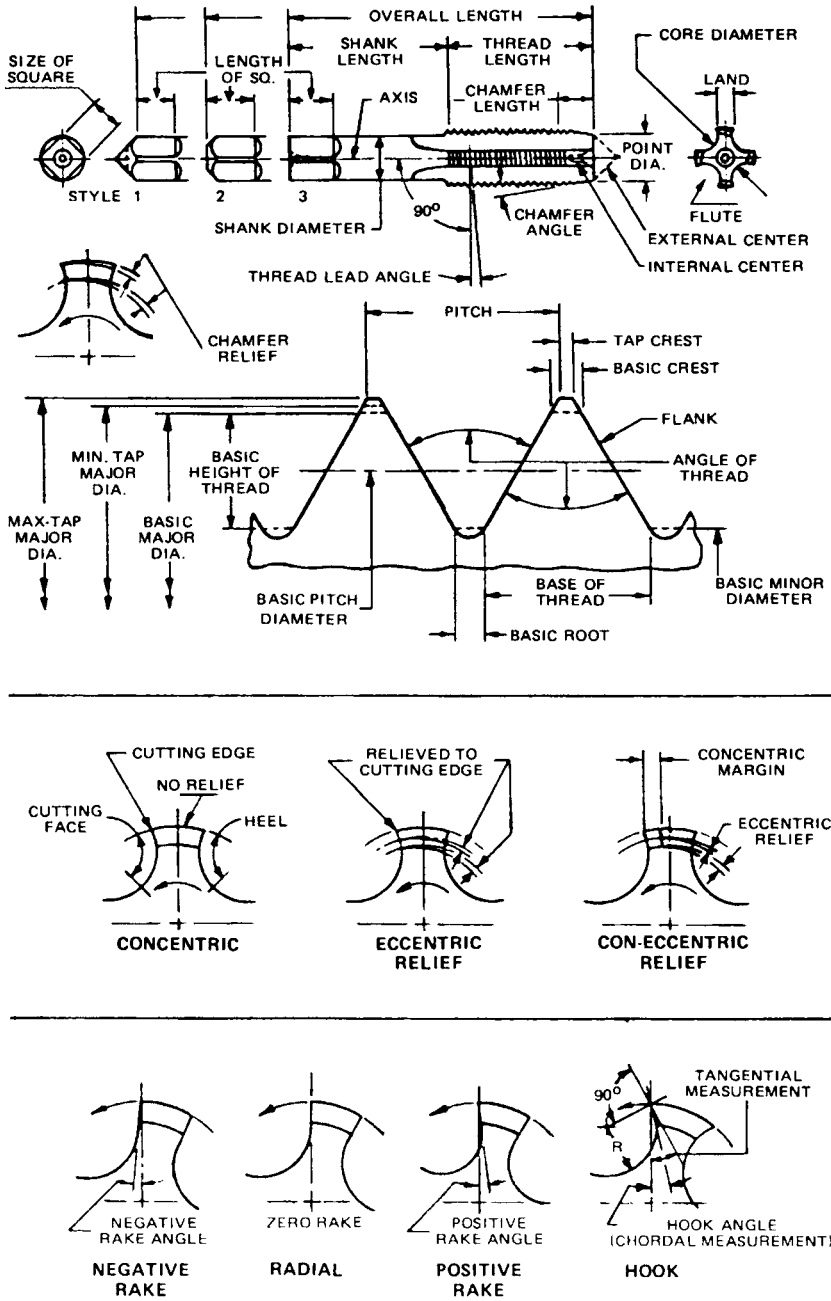
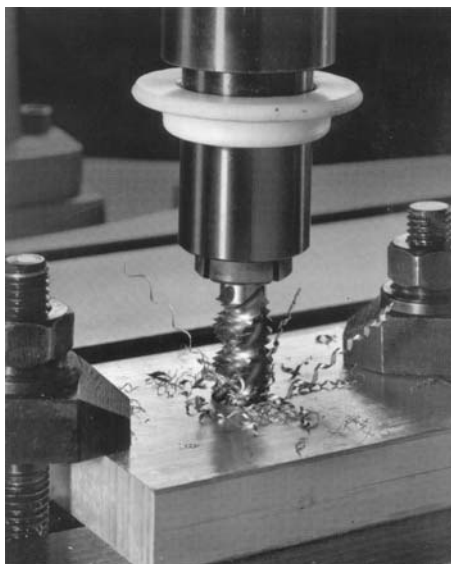


Figure 5.6. Nomenclature of a tap. Pitch, lead angle, and pitch diameter are among the items described.



*Figure 5.7. A high spiral tap showing removal of chips.*

## Machine Screw Taps

Machine screw taps are small-sized hand taps and are designated by numbers 0–12. They are available in two-, three-, and four-flute configurations.

Machine screw sizes are more readily available than fractional sizes under  $\frac{1}{4}$  in.

## Hy-Spiral Taps

These taps are for blind holes in stringy alloys and ductile steels, which clog taps of more conventional design. They are made with spiral instead of straight flutes. The stringy chips are drawn up the flutes in a manner similar to the action found in drills.

Because of their design, the metal is sheared; therefore, less torque is necessary to machine a thread.

The spiral flutes are supplied with a 45- to 52-degree helix angle depending on the manufacturer.

These taps are available in machine screw sizes and fractional sizes up to  $\frac{3}{4}$  in.

## Tapper Taps

These are used by nut manufacturers and are furnished with long shanks and short threads.

As succeeding nuts are tapped, they run up on the shank until full. When the tap is removed from the holder the nuts slide off.

The threaded portion has a chamfer of 11–12 threads for coarse-pitch threads and 15–17 threads for fine-pitch threads.

In bent taper taps, the shanks are bent for this taper tap so that nuts running up the shank can be ejected without stopping the machine to unload them.

## Nut Taps

These are used where the nature of the work requires a tap with a long shank or where the hole is of depth greater than can be handled by a hand tap.

The threaded portion has a long chamfer to distribute the cutting over many threads.

These are also used in bolt cutters and drill presses.

## Pulley Taps

Pulley taps are used for tapping set screws and oil cup holes in hubs of pulleys.

The shank is made to the basic major diameter of the tap to act as a guide through a hole in the pulley rim.

The long shank permits tapping in places inaccessible to regular hand taps.

## Taper Pipe Taps

Taper pipe taps are used for tapping pipe fittings on steam, fluid, and compressed gas systems. They are tapered  $\frac{3}{4}$  in./ft to ensure a tight joint. They come in American Standard pipe form or Dryseal.

The Standard requires application of a compound to the threads cut to ensure sealing, whereas, as the name indicates, the Dryseal does not.

These are also available with an interrupted taper cut for tough stringy material. The interrupted cut prevents tearing of the material because the tap cuts each land in alternate grooves, leaving the whole thread section to support cutting only on one side at a time.

Full-form threads cut on both sides simultaneously.

For aluminum, soft bronze, magnesium, and stainless steel, Hy-Hook taper pipe taps that have threads with a high hook that shears the material are available.

## Straight Pipe Taps

Straight pipe taps are used for tapping lock nuts and couplings for taper thread assemblies. They have the same cutting action as any hand tap; the sizes, however, are governed by tapered pipe thread sizes.

## Acme Taps

Acme threads are used for transferring power and have an included angle of 29 degrees.

Standard Acme taps are recommended for either blind or through holes in brass, cast iron, zinc die cast material, and free machining steel.

The length of the thread to be tapped should not exceed  $1\frac{1}{2}$  times the diameter of the tap.

For through-hole Acme tapping of most materials, a tandem style tap is recommended. This is actually two taps in tandem, with the first tap doing the

roughing. It has a thread with a sharper angle that permits the threads to be relieved and eliminates the wide, flat Acme crest.

The last few threads on the roughing section act as a guide for the short finishing section, which corrects the angle and brings the thread to size.

The follow tap then finishes the thread to its correct size.

## Forming Taps

These taps form threads rather than cut them. They are useful only in very ductile materials such as brass, copper, die castings, aluminium, lead and leaded steels.

This is a cold forming or swaging process. The thread is formed under pressure and the grain structure follows the contour of the thread.

Tap drill sizes are different from the tap drill sizes of cut threads. Slightly more power is required for tapping, but the thread formed is quite strong.

## Ground Thread Taps

Ground thread taps are made to sizes above or below basic pitch diameters. If they are higher, they are designated by a letter H; if they are below basic, they are designated by a letter L.

The pitch diameter limit numbers are:

L1—Basic to basic minus 0.0005 in.

H1—Basic to basic plus 0.0005 in.

H2—Basic plus 0.0005 to 0.001 in.

H3—Basic plus 0.001 to 0.0015 in.

H4—Basic plus 0.0015 to 0.002 in.

H5—Basic plus 0.002 to 0.0025 in.

H6—Basic plus 0.0025 to 0.003 in.

The markings on tap shanks represent various parameters. For example, a  $\frac{1}{4}$ -in. 20 NC or UNC ground thread tap marked GH3 is identified as follows:

G—Symbol for ground thread

H—Symbol for above basic

3—Symbol for pitch diameter size; in this case basic plus 0.001 in. to basic plus 0.0015 in.

Specifications for all taps are listed in the *Machinery's Handbook*.

## COUNTERBORES, SPOTFACES, AND COUNTERSINKS

Counterboring and spotfacing use similar tools, with counterboring enlarging a hole concentric with a smaller hole, whereas spotfacing is facing a flat circular surface (spot) on the work, usually perpendicular to a hole in the work (Figure 5.8). Both tools are guided by a pilot to ensure location. A counterbore can be used for

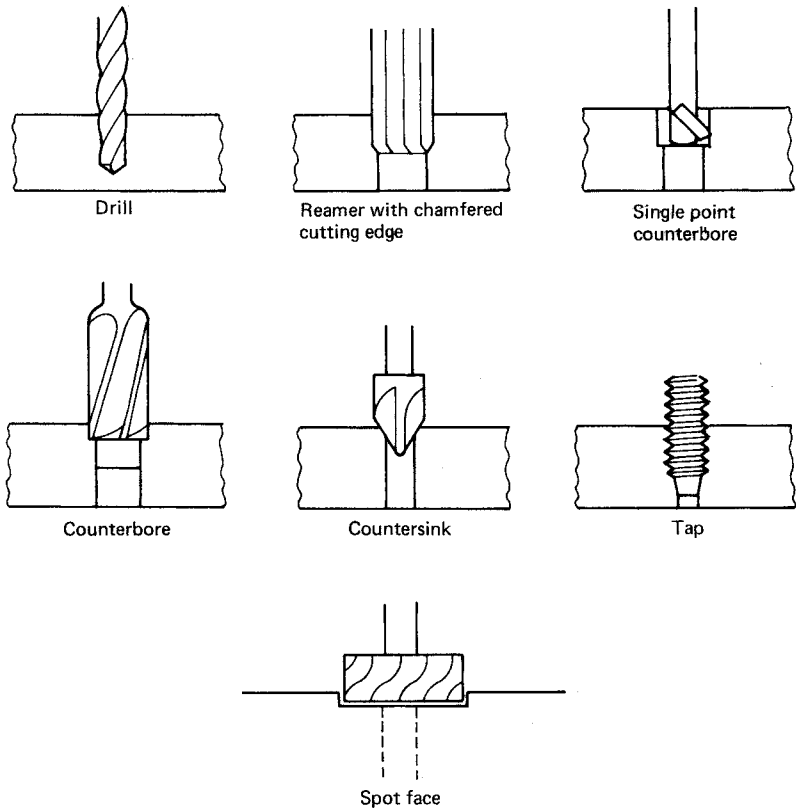


Figure 5.8. Counterbore, spotface, and countersink.

spotfacing smaller spots, although spotfacing usually has a larger area to be flattened so a part can be located on the surface.

## Counterboring

Counterboring is typically used for locating a screw in a hole slightly below the surface of the part it fits into. This allows other parts to be located above the piece while also ensuring that the mounted screw will not interfere with motion or location of the part.

## Countersinking

Countersinking bevels or tapers the work material around the periphery of a hole. This allows a flat-head screw or other part to lie flush in that hole.

Flat-head screws may come in various angles.

Other uses for a bevel made by a countersink include fitting a punch to a part to keep that punch from pulling through when it is retracted from a hole it has just penetrated through.



## Counterbore Styles

Counterbores come in three basic styles:

1. Solid, where the shank, cutter, and pilot are in one piece. These are used for machine and cap screws.
2. Inserted-blade, where the shank can accept various-size blades and various-size pilots. These are economical and can be used for large-sized bolts.
3. Three-piece counterbore; this is like the solid type but can accept a cutter of the same design as the solid and the pilot. Similar to an inserted-blade, it fits a series of sizes of cutters and pilots. This style costs more than inserted-blade.

## Spotfaces

Spotfacers are similar in style to counterbores, but a back spotfacer is inverted and consists of a one-piece shank and pilot with an interchangeable cutter. This type is inserted in the hole from the other side, then mounted in the drill press. The cutting action is upward.

## Countersinks

There are many types of countersinks:

1. Machine countersink, a three-flute tool used mostly to produce countersink holes for screws and rivet heads. It is also used for deburring holes and comes in various angles—60, 82, 90, and 120 degrees.
2. Chatterless, a countersink so designed as to eliminate chatter. They have three main flutes and three intermediate flutes. They also come in four-flute and four-intermediate-flute designs. Single-flute countersinks are also chatterless. Multiflute countersinks cause chatter because they cannot always be manufactured perfectly to cut at the same time. The single-flute type probably has the best style for eliminating chatter.
3. Countersinks with pilots ensure concentricity. Pilots are interchangeable and permit locating the position of the countersink.
4. Adjustable countersinks allow for the depth of the bevel to be accurate to within 0.001 in. They are used extensively in aircraft work.

# DRILL PRESS OPERATIONS

## Kinds of Operations

Drilling, reaming, boring, counterboring, and center drilling are all operations that can be done on a drill press. It is important that safety precautions be carefully observed when operating a drill press (Figure 5.9).

General operating considerations include the following.

1. Always clamp the work—*never* attempt to hold it by hand. If the material is thin and not clamped to the table, it can spin when the drill exits



*Figure 5.9. Drilling on a drill press, using cutting fluid, with part held in vise and clamped to drill press table. (Courtesy of DoAll.)*

and severely cut your hand. If a small hole is to be drilled in a large, thick piece that can be supported by butting an end against the column, then clamping may be avoided since very little torque will be generated. Otherwise the work must be clamped.

2. All gear-head machines must be stopped before changing speeds. The same is true for belt-driven machines that do not have variable speeds; on these, speeds must be changed while the machine is running. When setting speeds on a variable-speed machine, make sure the cutting tool is not in the machine—set the speed first. If changing the speed while the tool is mounted in the chuck, be sure to keep your hands and body well away from the tool.
3. Keep your head away from the machine and always wear a cap or other protective headgear to keep your hair from getting entangled in the tool.
4. When breaking through the hole at the bottom of the cut, ease up on the pressure to prevent grabbing.
5. Remove all burrs with a file.
6. Always check to see that the chuck key has been removed from the drill chuck before drilling.

## Preparing for Drilling

A typical sequence of steps to be taken in preparation for drilling is as follows.

1. Before mounting the part on the machine, prepare the part by laying out the location of the holes to be drilled. Use a center punch, a light hammer, and layout ink to lay out the hole. The part should be flat; either use



*Multiple Drilling Attachment. (Courtesy of DoAll.)*



*Multiple Tapping Attachment for Drill Press.*

ground flat stock or finish the part to be worked on so that it is flat and reflects the layout ink clearly. Coat the surface of the part with the layout ink and accurately locate the holes. Mark their location with a center punch. For extra accuracy, scribe a circle the size of the finished hole. When you start the hole, this will visually indicate whether the drill is actually in the center of the layout. If not, the center of the part must be relocated.

2. Secure a center drill of the correct size (that depends on the size of the hole to be finished) in the drill chuck. Clamp the part on the drill press with the hole to be drilled located directly under the center drill. Using the correct cutting speed and cutting fluid for the material to be machined, bring the center drill to the center punch mark and drill to half the depth of the taper on the center drill.
3. Stop the machine and remove the center drill. If the hole is  $\frac{1}{2}$ -in. or larger, a pilot hole (use a drill about one-fourth to one-half the size of the hole to be drilled) is recommended to ensure straightness of the hole.
4. Select the drill size to be used and set the speed of the machine to the correct surface feet per minute. Using cutting fluid bring the drill down to the work and machine the hole.

## Counterboring

To counterbore the hole, do the following.

1. After drilling the hole to the correct size, select a counterbore for the screw or pin that is going to be placed in the hole. The correct size for a screw to fit in the hole is  $\frac{1}{32}$  in. over the diameter of the screw; the counterbore pilot will accurately fit into the drill hole. The counterbore cutter will be  $\frac{1}{32}$  in. oversize to permit the cap of the screw to seat. If a spotface is going to be machined, then the pilot will fit accurately into the drilled hole but the spotface will be of the correct size for locating the flat surface.
2. The drilled hole is put in first and then the counterboring is done next. This ensures accurate location and saves time in setting up. After

drilling, remove the drill and mount the correct-size counterbore in the drill press.

3. Set the drill speed for the correct surface feet per minute for the diameter of the counterbore cutter.
4. Bring the counterbore close to the work and make sure that the pilot turns freely in the hole.
5. Start the machine and, using cutting fluid, bring the counterbore down into the work and drill to the proper depth, usually  $\frac{1}{32}$  in. deeper than the height of the screw head. If a pin is to be inserted and the motion of the pin is to be retarded by the head of the pin location, then the depth of the counterbore hole should be exactly the height of the pin head so that when the pin is installed, the head will be flush with the material.

## Reaming

To ream a hole, do the following.

1. To ream an accurate hole, the drill size must be selected correctly. The rule of thumb is 0.015 in. under the finished size. Select the correct drill size.
2. After drilling the hole using the above procedure, mount the reamer in the drill press. Always ream immediately after drilling to ensure correct location.
3. When accuracy and good surface finish are musts, reaming should be done in two operations, roughing and finishing. Use an undersized reamer anywhere from 0.002 in. to 0.005 in. for roughing, then finish with a Rose reamer.
4. For a good finish, the cutting speed of the reamer should be about 25–50% slower than the speed of the drill. Always use a good cutting fluid when reaming.
5. Start the machine and enter the work slowly. Continue reaming until through the hole—never turn the reamer backwards. While running, slowly raise the spindle of the drill press to remove the reamer and continue using cutting fluid while withdrawing the tool.

Sometimes it is necessary to transfer hole locations from one part to another to ensure accurate repair work or for other reasons. To do this:

1. Be sure both parts—the part to locate from and the part to locate to—are flat and clean with no burrs.
2. Clamp both parts together, making sure they are aligned perfectly.
3. Select a drill of the same diameter as the hole to be transferred and mount it in the drill press spindle. Alternatively, a transfer punch can be used; select one with the same diameter as the hole to be transferred (a transfer punch has a sharp center on its bottom so when it is struck, it leaves an impression on the new part similar to a center punch mark). When using a transfer punch, the parts do not have to be mounted on the drill press; simply align them carefully, clamp them together, and do the marking on a bench.

4. When drilling, enter into the old drilled hole of the guide part and spot a mark in the new part. The spot should not be deeper than the diameter of the drill. Do this to all the holes to be transferred. If a transfer punch is used, then mark all the holes with the punch.
5. Remove the original part and drill the spotted holes to the required diameter.

## Boring

When boring a hole on the drill press, either a boring bar that is adjustable by hand or a boring head that has a very accurate adjustment method can be used. Boring is the recommended method for achieving extreme accuracy, and both methods will accurately locate a hole that has previously been drilled. Boring can be used for locating a hole that can then be finish reamed. Reaming is not, however, practical for large holes.

### Considerations for Boring

When boring, use the following procedure.

1. After locating the position of the hole, drill to a size that makes it practical for the boring tool to enter. If the diameter of the work is large, drill the hole to a size that leaves only about  $\frac{1}{8}$  in. of material for removal by boring. For very large holes, it may be necessary to leave more material because of the capacity of the drill press. If boring is done on a radial drill press, a spade drill can be used for larger holes. The work should be mounted on parallels to ensure clearance for the boring tool when it exits the hole.
2. Mount the boring tool in the spindle and select the correct surface feet per minute for the boring tool. This is determined by the diameter of the hole to be finished.
3. Select a boring tool and mount it in the spindle or in the boring head.
4. Take a trial cut and using telescopic gages, measure the hole to see how much material is being removed. Make the cut.
5. Continue to bore the hole and reset the tool until the correct size is obtained. Use cutting fluid for each machining operation.

## Tapping

There are three methods of tapping with a drill press—hand tapping, tapping directly with the drill press, or tapping with a tapping attachment mounted in the drill press.

### Hand Tapping

When tapping small holes, hand tapping is the safest method to avoid breakage. Use the following procedure.

1. Drill the hole with the work clamped; after drilling the hole for tapping, do not move the workpiece.

2. Remove the drill and mount a center in the chuck. These are special centers to fit.
3. Place the tap in the drilled hole with a tap wrench mounted on the square portion of the tap. If the tap has a center hole on the end of the square, bring the center down into it to hold the tap securely. If the tap does not have a center in the top of the square, then the square will be pointed to allow a female center to locate on it. You must have this special female center to locate on the tap.
4. Turn the tap wrench to start the tap in the hole while applying light pressure with the center. Apply tapping compound to the tap before starting.
5. Continue to turn the tap with pressure applied on the handle until the depth required is obtained. Stop every two or three turns and reverse the tap to free the chips.

## Machine Tapping

For larger holes, machine tapping can be used. Be sure the part has not been moved after drilling. The part should be mounted on parallels. Use the following procedure.

1. Insert a tap sleeve into the spindle. This sleeve allows a loose fit of the tap and has a square machined in the center to accept the square of the tap and to drive it.
2. Mount the tap in the sleeve and bring the spindle down slowly to the hole. Coat the tap with tapping compound.
3. Set a spindle speed of 25% of the drill speed and, with light pressure, bring the tap in to the hole. When the tap enters the hole, it will feed itself forward and machine the thread. **Do not attempt to feed it by applying pressure to the spindle handle.**
4. When the tap has gone all the way through the hole, stop the machine. Reverse the spindle of the machine and allow the tap to exit by itself, as a result of the reversal action. The machine must have a reversal action to do this. Otherwise the tap must be removed by hand with a tap wrench.

## Tapping Attachment

This attachment is usually used for production tapping using two- or three-fluted spiral-pointed taps. It has a friction clutch or torque adjustment to adjust for the pressure applied on the tap. If excessive pressure on the tap causes it to stick, the clutch will slip and keep the tap from breaking.

After the holes have been drilled, the tapping attachment is located above the hole with the correct-size tap. Applying cutting fluid or tapping compound to the tap, insert it downward by hand into the hole. Once the tap grabs, it pulls itself downward until it is through the hole. The tapping attachment has a reversal mechanism that is actuated when slight upward pressure is applied, by hand, to the handle of the drill press. Once that motion is applied, the tap moves upward until it exits the hole.

For further information on drilling, refer to *Machinery's Handbook*.



*Threading for Automatic Tapping on a Drill Press. (Courtesy of DoAll.)*

## Project Number 2: Wheelpuller

This project is carried through the book. It is first used in this chapter to layout manually and machine. It is next used in Chapter 8 showing how it is programmed manually for machining on a numerical controlled machine. It is then shown in Chapter 8 how it is programmed on a user friendly controller on a numerical controlled machine. Finally, in Chapter 8 it is shown how it is programmed on a computer for a numerical controlled machine.

### WHEEL PULLER-PLATE

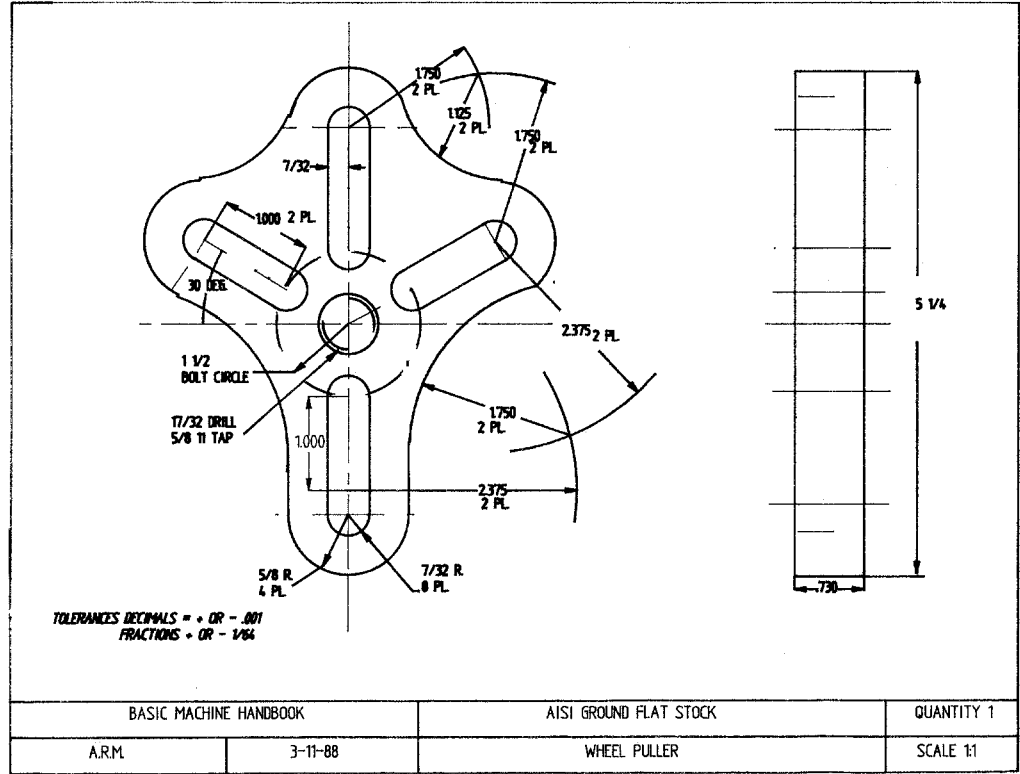
#### TOOLS:

Steel Ink Remover	White Chalk
Layout Ink	3/8 × 18 in. Oval File Band
Combination Square Set	1/4 in. 6 Pitch Imperial Saw Band
Scriber	7/16 in. 4-Flute End Mill
Hammer, Ballpenn	3/8 in. 2-Flute End Mill, Center Cutting
12 in. File, Flat	Center Finder
12 in. File, Half Round	Parallels, 3/16 × 1 × 6 in.
4 in. Solid Square	Coolant/H <sup>2</sup> O–Kleen Kool 25/1
6 in. Steel Rule	Cutting Oil #
4 in. Dividers	Tap Magic #
1 in. Micrometer	Coolant/H <sup>2</sup> O– #360 10/1
5/8-11 in. Tap	Contour Template
17/32 in. Drill	Countersink, 3/4 in. × 90°
Prick Punch, Auto	
Center Drill	

### PROCEDURE

*(Calculate speeds and fill in blanks before starting.)*

1. Secure 5 × 6 × 3/4 in. flat C.R.S.; cut off in C-80, use coolant. \_\_\_\_\_ SFM  
\_\_\_\_\_ Pressure. LEAVE GREEN COLOR CODE.
2. Deburr 12 edges.
3. Surface sand best side for layout.
4. Clean layout surface with steel ink remover.
5. Apply layout ink–3 light coats–protect bench with cloth.
6. Lay out per drawing.
7. Prick punch on 1/4 in. centers along peripheral layout lines and CL. intersections (9 points).



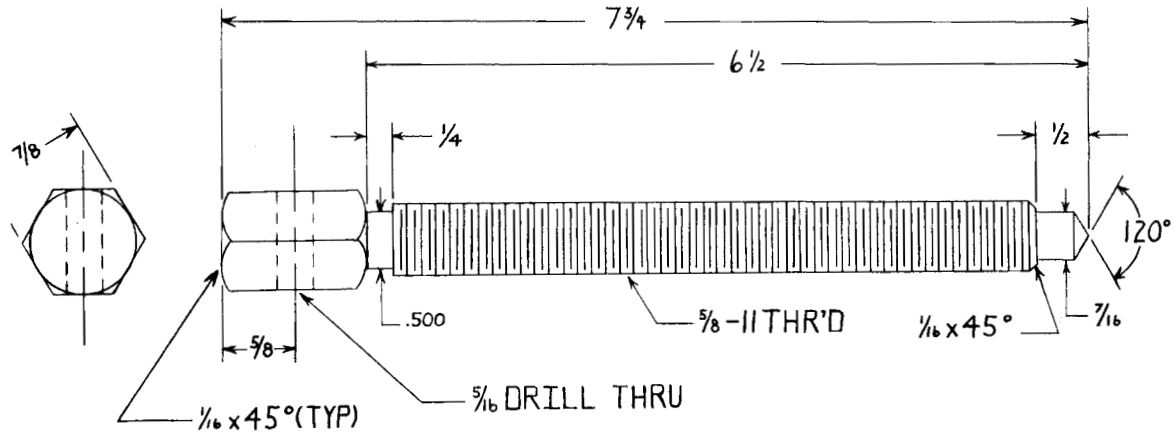


**WHEEL PULLER-PLATE (Cont.)**

8. Set up on milling machine and locate centerlines for slots. Using vise, parallels and center finder secured in *APPROPRIATE* holder (Erickson or drawbar). Repeat as necessary for all slots. Do *not* contact stock with center finder point.
9. Set table dials to zero at each slot end CL. Starting from tapped hole CL.
10. Install 3/8 in. end mill in appropriate holder and secure in spindle.  
NOTE: Make sure set screw is on a *FLAT*, and *TIGHT*.
11. With coolant on plunge, mill a through hole at each end of slot using *HAND FEED*; clear chips by retracting cutter every 1/32 in. (\_\_\_\_\_ RPM). Coolant/ H<sup>2</sup>O #360 10/1.
12. Secure spindle position; locate cutter to surface of stock and set dial to zero; raise table .200 in.
13. Using coolant and power feed, mill length of slot. \_\_\_\_\_ RPM \_\_\_\_\_ Feed. (CAUTION: Disengage feed before reaching final zero point; complete cut with hand feed.)
14. Raise table .200 in.; reverse feed; repeat Element #13.
15. Repeat Element # 14.
16. Repeat Element # 15.
17. Remove 3/8 in. dia. cutter and replace with 7/16 in. dia. 4-flute end mill.
18. With coolant on plunge, mill through full depth \_\_\_\_\_ RPM. Both ends.
19. With power feed, mill full length (full depth) \_\_\_\_\_ RPM \_\_\_\_\_ IPM. (See caution note in Element #13.)
20. Repeat Elements #9 through #19 for other slots.
21. Set up, center drill and drill 17/32 in. dia. hole \_\_\_\_\_ RPM \_\_\_\_\_ IPM. Countersink both sides.
22. Secure part and tap 5/8-11 in. hole; use tapping compound; check alignment with 4 in. solid square.
23. Clean 5 in. round, firmly apply template and practice contour sawing, using 1/4 in. 6 pitch blade \_\_\_\_\_ SFM (air feed). Mist coolant/H<sup>2</sup>O #360 10/1.
24. Band machine profile to layout lines.
25. File smooth with 3/8 × 18 in. oval band file; apply chalk as required.
26. Draw file edges to smooth finish.
27. Grind to .730 in. ±.001 in. DIM. Coolant/H<sup>2</sup>O Kleen-Kool 25/1.
28. Break all sharp edges with file or abrasive cloth.

**CLEANUP**

Clean machines, floors and tool racks. Put clean chips in drums. Return all tools to designated locations. Report any breakage or repairs needed.



NOTE: BREAK ALL CORNERS

TOLERANCES: DECIMALS  $\pm .001$   
 FRACTIONS  $\pm \frac{1}{64}$

DoALL TECHNICAL INSTITUTE	Material AISI 1018	Scale   :   Qty.
Part SCREW-WHEEL PULLER		
Dr. A.G.K.	Date 1-31-80	Part Number

## WHEEL PULLER-SCREW

### OBJECTIVE:

The trainee will use the proper saw blade, cutting speed and feed rate to shape the metal on the band machine.

The trainee will use the proper end mill, cutting speed and feed rate to form the metal on the milling machine.

The trainee will use the proper layout tools to correctly lay out the form to be machined on the above tools.

The trainee will use the proper tools and demonstrate ability to turn a thread on the lathe.

### REVIEW SAFETY RULES-DO NOT LEAVE "T" WRENCH IN CHUCK

#### STOCK:

1—3/4 × 5 × 6 in. Long Cold Drawn Flat Stock #1018. Coded Green.

1—7/8 × 8-1/4 in. Long Cold Drawn Hexagon #1018. Coded Green.

1—5 in. Dia. × 3/4 in. Long Cold Finished Round #1018. Coded Green.

#### TOOLS:

Facing/Turning Bit	3/8 in. Countersink
Threading Tool Bit	Live Center
5/8-11 in. Thread Gage	Chuck, Drill
Center Gage	3/16 in. Blade Grooving Tool
5/16 in. Drill	12 in. Mill File
12 in. Scale, Flexible	5/8-11 in. Threaded Split Sleeve
#4 Center Drill	

#### VERTICAL BAND MACHINE MILLING MACHINE LATHE

### PROCEDURE

$$\text{RPM} = \frac{\text{SFM} \times 12 \text{ in.}}{3 \times \text{D in.}}$$

$$\text{RPM} = \text{SFM for 4 in. D}$$

*(Calculate speeds and fill in blanks before starting.)*

1. Secure 7/8 × 8-1/4 in. C.R.S. Hex Stock. (Cut to length on C-80; use coolant.)  
SFM \_\_\_\_\_ Pressure. Leave Green Code.
2. Install 3-jaw chuck (use safety board) and drill chuck in tailstock.
3. Position stock in chuck and tighten; allow 1 in. projection maximum.
4. Center bit, face one end \_\_\_\_\_ RPM and center drill, 3/32 in. deep \_\_\_\_\_ RPM (use cutting oil).
5. Remove drill chuck and install live center in tailstock.
6. Reposition stock to center and in 3-jaw chuck allow 7-3/16 in. projection. Tighten chuck and tailstock.
7. Turn to .623 × 6-15/16 in. and 7/16 × 15/16 in. dia. per print. (Allow "7/16 in. ref only" length to remove center hole after threading.) \_\_\_\_\_ RPM \_\_\_\_\_ Feed.
8. Turn .500 in. dia. × 1/4 in. wide groove per print. \_\_\_\_\_ RPM. (Align and center bit; use cutting oil.)

**WHEEL PULLER-SCREW (Cont.)**

9. Center and align bit with center gage; turn 5/8-11 in. thread per print \_\_\_\_\_ RPM (retract tailstock to check threads with gage; *DO NOT LOOSEN CHUCK.*)
10. File burrs off threads and all sharp corners.
11. Reposition part in chuck; use split brass sleeve to protect threads; secure in chuck; allow maximum 1 in. projection.
12. Turn on 7/16 in. dia.  $\times$  (1/2 in. LG  $\times$  120° point. \_\_\_\_\_ RPM.
13. Reverse part in chuck and secure using split sleeve located against shoulder at 1/4 in. groove.
14. Face head to length. \_\_\_\_\_ RPM \_\_\_\_\_ Feed.
15. Adjust compound and tri-post and chamfer per print; reverse spindle rotation; cut from rear. \_\_\_\_\_ RPM.
16. Lay out 5/16 in. hole per print.
17. Drill 5/16 in. hole per print. \_\_\_\_\_ RPM \_\_\_\_\_ Feed.
18. Countersink both sides per print.
19. Remove all burrs and sharp edges.

**CLEANUP**

Clean machines, floors and tool racks. Put clean chips in drums. Return all tools to designated locations. Report any breakages or repairs needed.

## *Chapter 6*

# **GRINDING**

## **INTRODUCTION**

Grinding is the final operation in a machine shop. It allows for interchangeable parts, contributes extreme accuracy, and, in addition to creating an appealing finish, provides a surface finish that gives extra life to tools. It also is used for sharpening such tools as punches and dies, toolbits, end mills, and milling cutters. Finally, it produces flatness, cylindrical shapes, or forms for specific uses.

Project No. 3, located at the end of this chapter, is for machining a toolmaker's vise. The step-by-step procedure is given, along with drawings of each component.

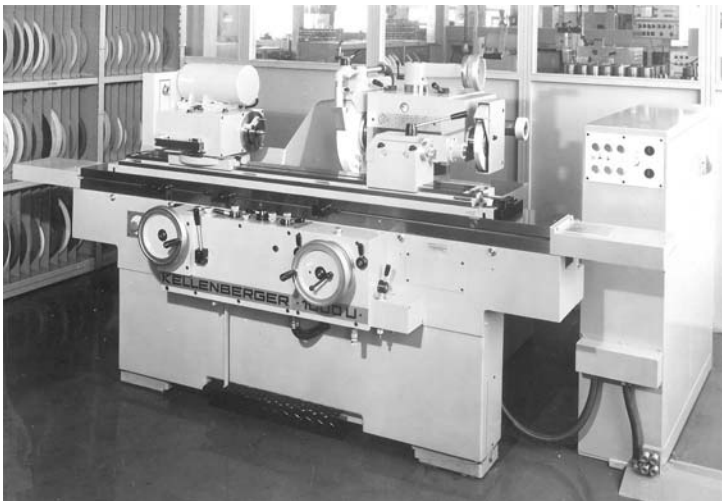
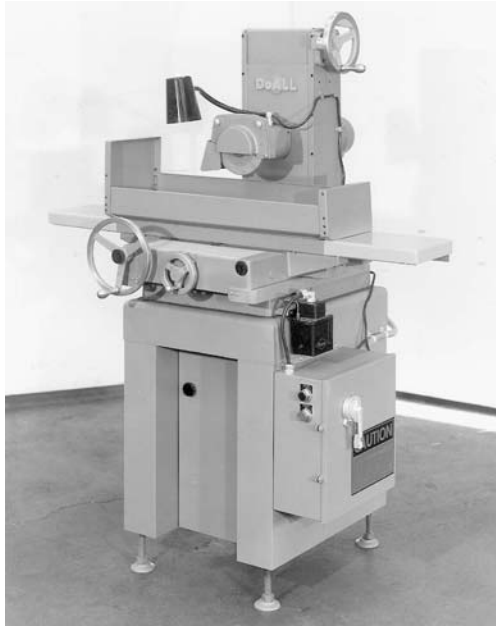
## **THE PROCESS**

Grinding is a metal-cutting process in which chips are removed from the work-piece, just as they are on a milling machine, lathe, or other chip-making machine. Chips are produced as the grain of the grinding wheel cuts into the material. This means that the principles of metalworking that hold true for other machines also apply to grinding. For example, as the point of the abrasive grain wears down, it gets dull just as do the cutting edges of any other tool. As dullness increases, so does friction, thereby generating even more friction and heat.

Therefore, to maintain accuracy and precision, it is critical to keep the abrasive cutting tool sharp. A description of the abrasive wheels and their maintenance will follow the grinding procedure.

## **BASIC GRINDERS**

The three basic types of grinding covered in this chapter are surface grinding, cylindrical grinding, and tool and cutter grinding (Figure 6. 1).



*Figure 6.1. The three basic types of grinding machines. Top to bottom: the surface grinder, cylindrical grinder, and, on p. 176, tool and cutter grinder. (Courtesy of DoAll and Kellenberger.)*

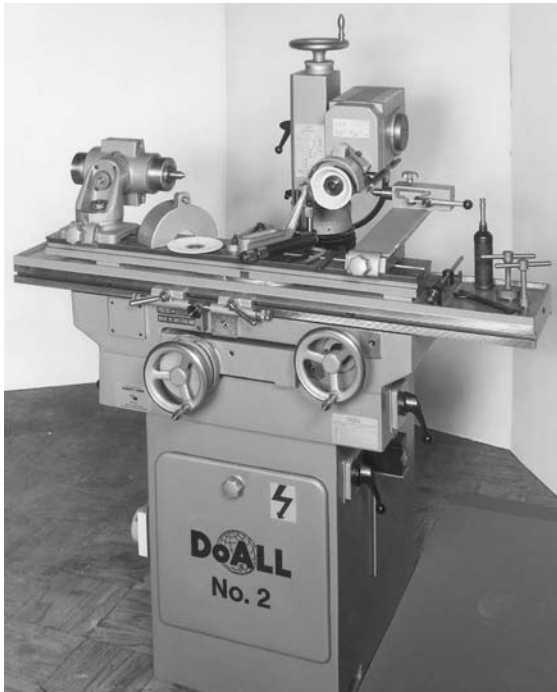


Figure 6.1. (cont'd)

## Surface Grinding

Although all grinding is finishing a surface, the term refers to the finishing of a flat surface. This is known as “precision surface grinding” and it comprises about 75% of all grinding done in a machine shop. There are four types of precision surface grinders:

1. Precision surface grinders with horizontal spindles and reciprocating tables. The wheel cuts on its periphery as the table carries the workpiece back and forth beneath it. This is the most common type of grinding machine found in toolrooms and machine shops today. Most are hand or manually manipulated grinders; larger, hydraulically operated surface grinders are found in smaller numbers, and the smaller grinders are also available with hydraulics. The tables are rectangular and usually 6 in. wide by either 12 in. or 18 in. long. The high-production types have tables as big as 6 ft by 16 ft. The grinding wheels cut on their peripheries and vary in sizes from 7 in. in diameter and  $\frac{1}{2}$  in. in width to 20 in. in diameter and 8 in. in width. This type of grinder is the one most often used because of its high accuracy and the fine surface finishes it imparts. It grinds a straight pattern, which accounts for its superior finishes and high precision—the greatest in its class.
2. Precision surface grinders with horizontal spindles and rotary tables. Its wheel also cuts on its periphery, but the spindle traverses horizontally

from the edge to the center of the table, feed is accomplished by moving the work mounted on the table up into the wheel, and the table motion is rotary rather than reciprocating. This machine is also found in tool-rooms, although not in the same numbers as the reciprocating table machines. Since the table and work rotate in a circle beneath the grinding wheel, the surface pattern is a series of intersecting arcs. It is an excellent machine for round, flat parts because the wheel is in contact with the work at all times.

3. Precision surface grinders with vertical spindles and reciprocating tables; The work rides back and forth on a reciprocating table, but the wheel is cylindrical and cuts on its side rather than on its periphery. These are generally high-production machine tools. Because of its design, these machines can remove large amounts of material at one time—as much as  $\frac{1}{2}$  in. of material can be removed in one pass. Two flat surfaces are in contact with each other during grinding. These are the flat side surface of the grinding wheel and the flat surface of the material being ground.
4. Precision surface grinders with vertical spindles and rotary tables. The work and the wheel rotate into each other. With vertical spindles and rotary tables, this type of precision surface grinder is primarily used for large work and volume grinding of large loads of small parts. This machine also removes large amounts of material in one pass, taking cuts  $\frac{1}{2}$  in. deep. The side or the face of the wheel does the grinding. The wheel is either solid or mounted in segments to provide cooler grinding action and to save wheel material. If parts are small and mounted off center around the table, the surface patterns created are a series of intersecting arcs. The rotary table tends to make it a very accurate grinder and is used in many ways; it grinds production parts, grinds very large parts, and is excellent for grinding large batches of small parts. Some of these machines utilize multiple spindles, as many as five, thus speeding up grinding processes. On some models, the wheel can be tilted slightly to increase the pressure applied to the work and achieve even higher metal removal rates, then can be returned to its normal position parallel to the workpiece surface for finishing.

A magnetic chuck holds the workpiece for all of the above machines. The chuck is clamped to the worktable mounted on a saddle on the machine; on reciprocating models, the table rides left and right on the saddle on ways machined into the bottom of the table and the top of the saddle; on rotary machines, the chuck rotates on the table. These precision ground and hardened ways mate, allowing the movement of the worktable, chuck, and the workpiece.

The chuck can be designed to use permanent magnets, with the holding force being actuated or released by a lever that positions the magnets, or can be actuated by an electromagnetic unit with an outside power source. Both types of magnets are very powerful and hold the parts securely.

Toward the rear of each type of machine is the column that supports the spindle to which the grinding wheel is attached. On the typical vertical machine with reciprocating table, the spindle is moved up or down by means of a very accurate screw located in the column; a vertical handwheel allows the operator to move the



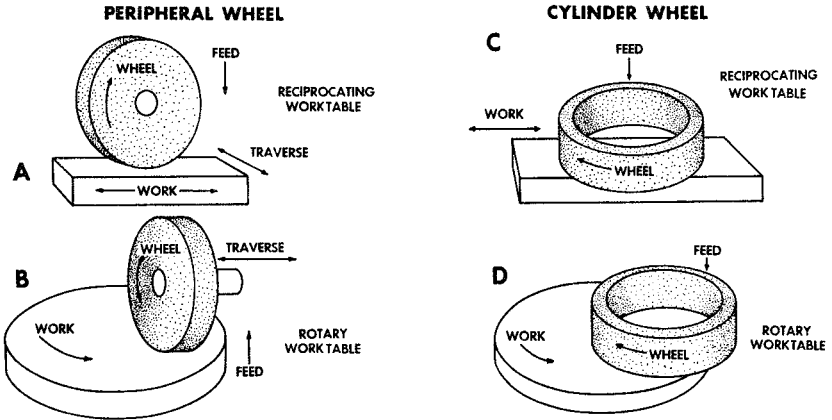


Figure 6.2. The four basic types of precision surface-grinding machines: (a) horizontal spindle and reciprocating table; (b) horizontal spindle and rotary table; (c) vertical spindle and reciprocating table; (d) vertical spindle and rotary table. (Courtesy of DoAll.)

spindle up or down with an accuracy of from 0.0001 in. to as much as 0.000050 in., depending on the machine.

The base of the surface grinder is constructed to ensure its rigidity and that it does not move up or down; the column must also be very rigid. This rigidity is necessary to maintain the extreme accuracies built into the machine.

Guards are appropriately located to protect the operator against accidents.

## Cylindrical Grinding

On this machine the work is mounted between centers or in a three- or four-jaw chuck and rotates while in contact with the grinding wheel (Figure 6.3). Both the work and the grinding wheel rotate counterclockwise. If the finished section to be ground is wider than the wheel, the wheel can be moved transversely.

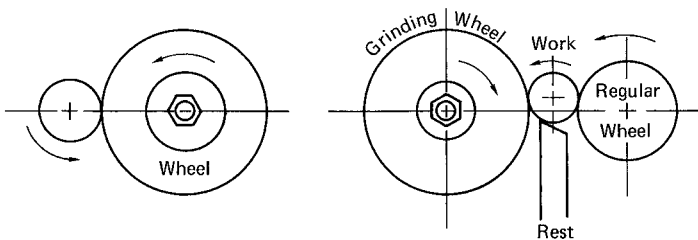


Figure 6.3. Cylindrical grinding. In cylindrical grinding, the workpiece and grinding wheel both rotate in a counterclockwise direction so at the point of contact the surface of each is moving in the opposite direction. This produces extremely fine finishes and accuracies.

If the part to be finished is the same size as or less than the width of the wheel, then plunge grinding is done; that is, the wheel can be fed in continuously to finish the part.

Very fine finishes are obtained with cylindrical grinding. Accuracies to within 0.000010 in. are possible; extreme care must, however, be taken to obtain such accuracies.

Parts are normally mounted on clean, accurately located centers and driven by a dog. The centers that the parts are located on must be ground in place and be free of nicks and dirt. If necessary, the work should be supported by work rests; these are used for long pieces, and they support the work from the opposite side of the wheel to prevent deflection. Cutting fluid is extremely important and must be clean and fresh.

When grinding transversely, the wheel should overrun the part by at least about one-third of the wheel to ensure accuracy. Otherwise an oversize condition can occur. The wheel should be balanced and trued (Figure 6.4).

When plunge grinding, several diameters or shapes can be accomplished in one operation (Figure 6.5).



*Figure 6.4. Wheel balancer. To ensure accurate, vibration-free operation, the grinding wheel must be properly balanced and trued.*

## Internal Grinding

Internal grinding, as in all grinding, is performed to finish a part to size. For this operation, the part is located on a face plate or a four-jaw chuck. The part must be located very carefully to ensure accuracy. If many parts are to be done, then locating a fixture on a face plate is most practical. That allows the next part to be located very accurately after the previous one is finished. It is fast and economical.

## Centerless Grinding

In centerless grinding, it is not necessary to mount the work between centers—the part is simply positioned on a work support between the grinding wheel and a feed wheel that causes the part to rotate and feeds it into the grinding wheel for



*Figure 6.5. Several shapes and diameters being ground during one cylindrical grinding operation.*

cutting. There are three methods of centerless grinding and all are production types of grinding.

One is through feeding, where the part is ground to the same size all the way through.

The second is called infeed grinding; a stop is set and the part is ejected after grinding to the stop position. This allows shoulder and step grinding.

The third is end-feed grinding, in which short tapers and spherical shapes are ground. After grinding one of these shapes, the part is ejected. Both the infeed wheel and the grinding wheel are dressed to the shape required and the part is fed in to the side of the wheel to an end stop.

## Abrasive Belt Grinding

Abrasive belt grinding is a fast method for removing metal and replaces some conventional machining operations because of the speed of metal removal and accuracy. This method does not provide the fine finishes that surface grinding creates, but it is a viable replacement for many regular machining operations.

## Cutter Grinding

Cutter grinding provides the means of resharpening cutters after they become dull. Although milling cutters and end mills are the types of tool most frequently sharpened, taps, drills, reamers, and broaches are also sharpened. Broaches, however, require special machines or attachments for this.

Tolerances must be maintained when sharpening cutting tools; all sorts of attachments are available for the cutter grinder to attain these tolerances.

When grinding with the standard tool and cutter grinding machines, a tooth rest is used to position each cutting edge quickly and accurately.

The selection of grinding wheels is important to accommodate the need for clearances, the size of the cutter, and the shape of the tooth.

The cutter grinder, in the main, is a hand operation that requires great skill on the part of the operator.

## GRINDING WHEELS

### The Tool

Grinding is a chip-making process and the grinding wheel is a metal-cutting tool that uses small abrasive grains of stone to cut metal. As the point of an abrasive grain cuts into the material, a chip is produced.

### Wheel Characteristics

A grinding wheel has five basic characteristics that are important to its cutting action. These are described in the following subsections.

**1. Type.** Two types of abrasive, aluminium oxide and silicon carbide, are used to manufacture standard wheels. These are classified by their hardness and friability.

Hardness is the property that gives the grain the ability to cut through a variety of material easily, whereas friability identifies its tendency to fracture and expose new sharp cutting points as the exposed ones dull.

**2. Grain Size.** Grain size represents the size of the abrasive particles used to manufacture the wheel. The size used in a given wheel is determined by the amount of stock the wheel is designed to remove and the finish required for the job.

If, for example, a sharp corner is required, the grain size used will be small (fine). On the other hand, if large amounts of material are to be removed—as on a rotary grinder—then a large (coarse) grain wheel is used. Other requirements would fall in between.

**3. Bond.** The primary purpose of a bond is to hold the abrasive grains together. There are several types of bonds available:

*Vitrified.* These are the ones most commonly used in grinding wheels. They are strong and rigid and produce smooth and accurate surface finishes, and they perform nearly every type of grinding operation. They are limited, however, to speeds of about 6500 surface feet per minute (SFM).

*Resinoid.* These are generally used for cut-off, snagging, and portable grinding wheels. They are strong, withstand speeds of up to 16,000 SFM, and, with special grain coatings, can be used with cutting fluids.

*Rubber bonded.* These are used, in the main, as regulating wheels for centerless grinding. In some instances, they are also used for cut-off applications.

*Shellac bond.* These are used primarily for very high finishes such as those found on cutlery, but otherwise are rarely found in the machine shop.

The choice of bond is determined by:

- \* The amount of metal to be removed.
- \* The surface finish required.
- \* The speed of the wheel.

**4. Structure.** The structure of the wheel is determined by the relationship among the abrasive grains, the bond, and the porous area. If the grains are widely spaced, the structure is considered to be open whereas, conversely, if they are close together, it is termed dense.

Open structure signifies a cooler cutting wheel suitable for heavy stock removal, whereas dense structure will give finer finishes, although much smaller amounts of metal can be removed at one time.

**5. Grade.** The grade is the measure of the holding ability of the bond. A hard wheel is one in which the abrasive grains are held very tightly in place, while in a soft wheel, the grains are more loosely held and fracture more readily.

The rule of thumb is: Use a “soft” wheel to grind hard material and a “hard” wheel to grind soft material.

Grade depends on the following conditions:

- \* Hardness of material.
- \* Area of contact between wheel and part.
- \* Whether grinding dry or using cutting fluid.

## Aluminum Oxide Wheels

The majority of grinding wheels utilize aluminum oxide as the abrasive, and in the shop they are used to grind ferrous metals—iron and steel and their alloys.

## Silicon Carbide Wheels

Silicon carbide wheels fracture easily and are used mainly for grinding carbides, but they wear very rapidly thereby requiring frequent replacement. As a grinding wheel wears down, its diameter decreases and, as a result, the periphery wheel speed slows down. As the peripheral speed slows with wear, the wheel breaks down even more rapidly and ultimately loses its ability to remove material faster than the grains fracture and are removed from the wheel.

There are two types of silicon carbide wheels, green and black. The green, which fractures more easily, is used for carbide grinding whereas the black is used

for grinding cast iron; such soft, nonferrous metals as copper, brass and aluminum; and ceramics.

## **SUPERABRASIVES**

The superabrasives include natural and man-made (polycrystalline) diamonds and cubic boron nitride (CBN).

### **Diamond Grinding Wheels**

Diamond wheels hold their shape. The breakdown of the bond is what allows the cutting edges to be exposed and maintain their diameters. Diamond is especially good for grinding carbides and will provide some 30–40 times the useful life of a silicon carbide wheel. This amounts to a considerable savings in tool costs.

Diamonds are not good for removing steel because they have a tendency to bond to it; consequently they are not used for grinding steel alloys and most other metals.

Diamond wheel bonds take one of three forms—metal, vitrified, and resinoid. The resinoid is the most popular because the bond breaks down evenly, exposing the diamonds for fast material removal. It can be supplied for either wet or dry grinding.

The metal bond must always be used with coolant and requires the slowest rates of grinding of the three forms. It produces fine finishes and is frequently used for hand grinding operations on a pedestal grinder. The bond breaks down very slowly, resulting in long wheel life but permitting only slow rates of stock removal.

The vitrified bond acts like a combination of the resinoid and metal. It breaks down faster than the metal bond but slower than the resinoid, providing good wheel life and faster stock removal rates than the metal. It also can be supplied for either wet or dry grinding.

### **CBN Grinding Wheels**

Where diamond cannot be used, CBN provides a practical answer, finding application for grinding tool steel, hard iron, gray cast iron, hard facing alloys, surface-hardened steels, and nickel- and cobalt-based alloys. Best suited for light-metal-removal requirements, CBN was originally used for tool and cutter grinding, but its use has expanded into part-production applications, particularly where pregrinding steps have put the part into a “near net shape.”

### **Grinding Wheel Standards**

There are two basic systems for identifying grinding wheels, one for aluminum oxide and silicon carbide grinding wheels, the other for diamond wheels (Figure 6.6).

**Standard Marking System Chart (for aluminum oxide and silicon carbide grinding wheels)**

Sequence	1	2	3	4	5	6
Prefix	Abrasive Type	Grain Size	Grade	Structure	Bond Type	Manufacturer's Record
51	A	36	L	5	V	23
<b>Manufacturer's symbol indicating exact kind of abrasive</b> <small>(use optional)</small>  Aluminum oxide—A Silicon carbide—C	Coarse	Medium	Dense to Open	V—vitrified S—silicate R—rubber RF—rubber reinforced B—resinoid BF—resinoid reinforced E—shellac O—oxychloride	<b>Manufacturer's private marking to identify wheel.</b> <small>(use optional)</small>	
	10 30 70 220	12 36 80 240				1 9
	14 46 90 280	16 54 100 320	2 10			
	20 60 120 400	24 150 500	3 11			
		180 600	4 12			
			5 13			
		6 14	Medium			
		7 15				
		8 Etc.	Hard			
		<small>(use optional)</small>				
Soft			GRADE SCALE			
A B C D E F G H I J K			L M N O P Q R S T U V W X Y Z			

**Identification Chart for Diamond Wheel Marking**

Sequence	1st	2nd	3rd	4th	5th	6th	7th	8th
	D	100	N	100	B	1/8		
	Abrasive Type	Grit Size	Grade	Concentration	Bond	Bond Modification	Depth of Diamond Section (Inches)	Mfrs. Identification
<b>D-Diamond</b>	24 120 600	A (Soft)	25 (Low)	B-Resinoid	Mfrs. Specific Bond Record	1/32	Optional	
	30 150 800							
	36 180 1000					1/16		
	46 220 1200				M-Metal			
	54 240 1500	To	To			(Optional)	1/6	
<b>(MD) Man Made (SD) Diamond</b>	60 280							
	80 320	Z (Hard)	75 (High)	V-Vitrified		1/8		
	90 400							
	100 500			100				

Figure 6.6. Systems for identifying grinding wheels made of aluminum oxide and silicon carbide (top) and for identifying grinding wheels made of diamond (bottom).

Although there are some 27 standard grinding wheel shapes, the machinist basically uses two types: the straight wheel, where grinding is done on the periphery; and the wall grinding wheel, where the grinding is done on the face or side of the wheel.

When grinding on a tool and cutter grinder, such other shaped wheels as the flaring cup and the dish are used (Figure 6.7).

Wheels used for operations other than surface grinding are discussed elsewhere in this chapter.

The wheel most commonly used in the machine shop is the type 1 straight wheel, which grinds on its periphery. The most complicated aspect of the wheel is

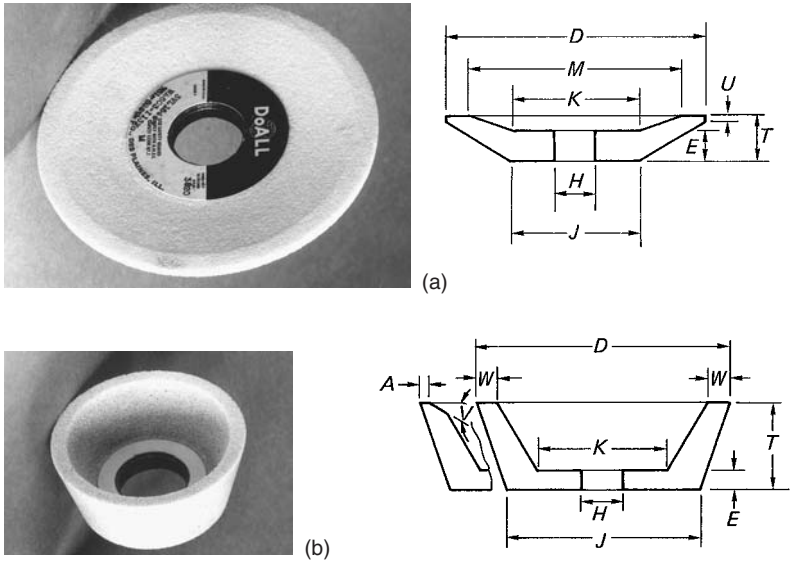


Figure 6.7. Dish (a) and flaring cup (b) wheels. The shapes of dish- and flare-type grinding wheels permit them to grind features inaccessible to peripheral wheels, making them useful for tool and cutter grinding.

the marking system used to identify it. All manufacturers use the standard marking system to identify their wheels, typically adding some of their own codes.

The standard codes require the following:

- \* **Abrasive Type**  
Aluminum oxide (A)  
Silicon carbide (C)
  - \* **Grain Size**  
Anywhere from 10 to 600
  - \* **Grade**  
Lettered A to Z
  - \* **Structure**  
From dense to open (1–16)  
This is an option. Not all companies use this number.
  - \* **Bond Type**  
Lettered, for vitrified (V) to shellac (E).
  - \* **Sample Lettering**  
A36-L5V
- |    |                    |
|----|--------------------|
| A  | Aluminum oxide     |
| 36 | Grit size (medium) |
| L  | Grade (medium)     |



- 5 Structure (dense)
- V Bond (vitrified)
- 9A-46J12-V22
- 9 Manufacturers symbol used to designate friability; this is not standard and will need to refer to manufacturer for meaning
- A Aluminum oxide
- 46 Grit size (medium)
- J Grade (medium)
- 12 Structure (open)
- V Bond (vitrified)
- 22 Manufacturing designation of composition; must check with manufacturer for meaning

## Grinding Wheel Use and Care

Before any grinding wheel is used, it should be tested for any flaws in its manufacture or damage incurred during shipping. The following precautions must be taken

1. Before mounting the wheel on a spindle, *always* use blotters. These are disks of compressible material supplied by the manufacturer; they protect against any irregularities between the flanges used to hold the wheel when it is mounted on the spindle and the wheel itself. All new wheels come with them and they have all the wheel markings on them. In the course of use, however, blotters may come off or be removed. If they are not on a wheel selected for use, put new blotters on regardless of the size of the wheel. **Do not mount a wheel on a spindle without a blotter on each side of the wheel.**
2. Hold the wheel with a pencil or other thin object thrust through the mounting hole of the wheel and tap it very lightly with a metal object. A clear ring should be heard; if the sound is dull it indicates there is a crack (not always discernible) in the wheel. **If a dull ring is heard, do not use the wheel. It is defective.**
3. After taking these first two precautions, mount the wheel on the spindle. It should fit snugly, but not so snug that the wheel must be forced on. **If wheel fit is too tight, do not use.** On the other hand, the wheel should not fit loosely—if it does, the hole is too large. **If too loose, do not use the wheel.**
4. There are two flanges supplied with the surface grinder. They must both be the same size. **Do not use flanges of unequal diameter.** The flanges must be equal to at least one-third the size of the wheel diameter. They are recessed so that only their rims bear against the wheel blotter.
5. The adapter flange (the one that is used to tighten against the wheel) must be tightened only enough to hold the wheel firmly. **Excessive tightening can break or damage the wheel.**

## Trueing and Balancing

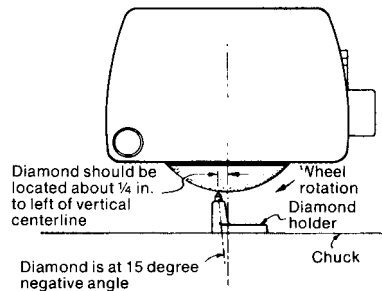
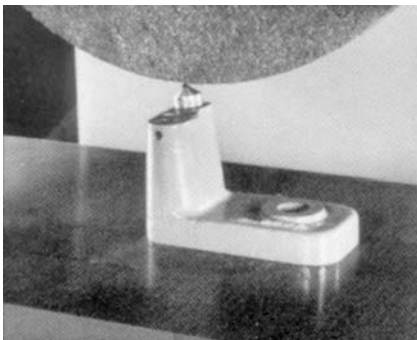
Trueing and balancing the wheel is the accepted method of preparing a grinding wheel for operation. Trueing is often thought to fully prepare a wheel, even taking care of the balancing. Trueing is necessary because it compensates for any lack of concentricity that may occur because of off-center mounting of the wheel on the adaptor and the spindle of the machine. It will not, however, correct an out-of-balance condition.

A wheel that is out of balance acts somewhat like a hammer. A 7-in. wheel that is 1 oz. out of balance hammers the work on its heavy side with the impact of a 7-lb hammer; i.e., each time the heavy side comes around, it is equivalent to a 7-lb hammer hitting the work. Therefore, to get a good finish on the part, the wheel must be balanced.

This is usually accomplished by providing the wheel with an adaptor, flanges, and two adjustable weights. An arbor is mounted in the wheel hole and located on a balancing machine that has two free-spinning disks that allow the wheel and arbor to rotate freely.

If the wheel is out of balance, it will come to rest with its heaviest side down. The adjustable weights are moved until the wheel, when rotated to any position, will remain at rest. These two weights counterbalance the heavy spot in the wheel, which is then in static balance.

Following balancing, the wheel must be trued by dressing it with either a single-point diamond dressing tool or a cluster dressing tool. This is probably the most important step affecting the ability of the wheel to provide a good finish. It is important to note that although trueing and dressing are both accomplished the same way, they are not the same thing: trueing is making the wheel concentric and dressing the wheel is exposing new cutting edges of the wheel (Figure 6.8).



*Figure 6.8. Trueing and dressing. These are both performed the same way, but the purpose of trueing is to make the wheel concentric whereas that of dressing is to clean out material picked up during grinding and expose new abrasive particle cutting edges.*

The periphery of a trued wheel is concentric with the center of the spindle. In addition, trueing not only centers the wheel but it also keeps the wheel clean; that is, it removes all of the material that has loaded the wheel during grinding.

Trueing ensures that all points on the wheel contact the workpiece at the same height above the table of the surface grinder. If this isn't accomplished, chatter occurs, resulting in a poor finish and poor accuracy.

## Dressing

Dressing removes materials picked up during grinding; it also removes high spots and other irregularities created on the surface of the wheel periphery during grinding and restores that surface to its proper condition.

Trueing and dressing the wheel is accomplished by the following procedure.

1. A diamond dressing tool is used for both; the diamond is brazed into a shank mounted in a holder secured in place by a magnetic chuck.
2. The diamond should be placed under the wheel about  $\frac{1}{4}$  in. to the left of the vertical centerline. This allows the diamond to wear evenly. The diamond can be rotated in the holder to keep a sharp point consistently exposed.
3. When beginning, position the wheel and grinder table so that the highest point on the periphery of the wheel can contact the diamond to prevent the diamond from taking too deep of a cut.
4. Lower the wheel until it touches the diamond, then move the table across the wheel so the diamond cleans up its periphery. Repeat this, lowering the wheel about 0.002 in. after each cross-feed. On the final cross-feed, lower the wheel about 0.0005 in. to complete the trueing. When dressing to expose the cutting edges, the wheel should be clean of all materials between the bond before making the final dress of 0.0005 in.
5. Cross-feed determines how the wheel will react in grinding. If a fast speed is used, a coarse dress results; this makes the wheel act as a soft wheel and it will remove greater amounts of material. A very slow cross-feed will give a very fine finish to the wheel, thereby making it act as a hard wheel; consequently, lesser amounts of material will be removed.
6. New cluster dressers are available that impart a very fine dressing to the wheels, which in turn allows them to create good workpiece finishes. Cluster dressers are carefully located under the center of the wheel; they can remove more material from the wheel per pass than single-point dressers. Again the rate of cross-feed determines the coarseness of the dress. The type of dressing tool used is a matter of preference for the person using the grinder.

Diamond dressing tools usually come in six-point or twelve-point forms. The six-point tools are better because the points are sharper. After a diamond point has become dull, the tool can be sent back to the manufacturer and reset to expose another sharp point for very little cost. This is inexpensive and gives six edges to work with before a new tool is needed. The cost of a good diamond dressing tool is minimal compared to the results obtained.

## MAGNETIC CHUCKS AND ACCESSORIES

There are a great many accessories and attachments designed to perform specific tasks that simplify the setup and operation and add versatility to precision surface grinders (Figure 6.9).

### Magnetic Chucks

The most common workholding devices used to secure material to the surface of the grinder table are permanent-magnet and electromagnetic chucks. Since the work is held to the chuck by magnetic force, the workpiece must be ferrous and possess magnetic properties. Permanent-magnet chucks are usually small and their use is limited to small workpieces.

Holding power is achieved by means of permanent magnets made of special alloys and separated by nonmagnetic materials. An on-off lever shifts the magnets to concentrate or disperse their combined magnetic field that constitutes the chuck's holding power. Sizes generally range from 6 in. × 12 in. to 10 in. × 30 in.

Electromagnetic chucks come in all sizes ranging from 4 in. to as large as 6 ft × 16 ft. They operate on direct current of 24 V and normally connect to either a 110- or 220-V power source.

The holding power of the chucks is affected by such factors as size, shape, and magnetic characteristics of the workpiece—larger and thicker pieces with greater areas in contact with the chuck are held more powerfully than smaller and thinner parts. When the holding power is great, it creates additional friction between the workpiece and the surface of the chuck, thereby keeping the work from sliding sideways when grinding pressure is exerted on the part.

### Magna-Vise Clamps

To aid in holding nonmagnetic and other difficult-to-hold parts, magna-vise clamps are a very handy accessory. These clamps have comb-like teeth that are unevenly spread throughout and pulled down in a forward motion when the magnet is turned on.

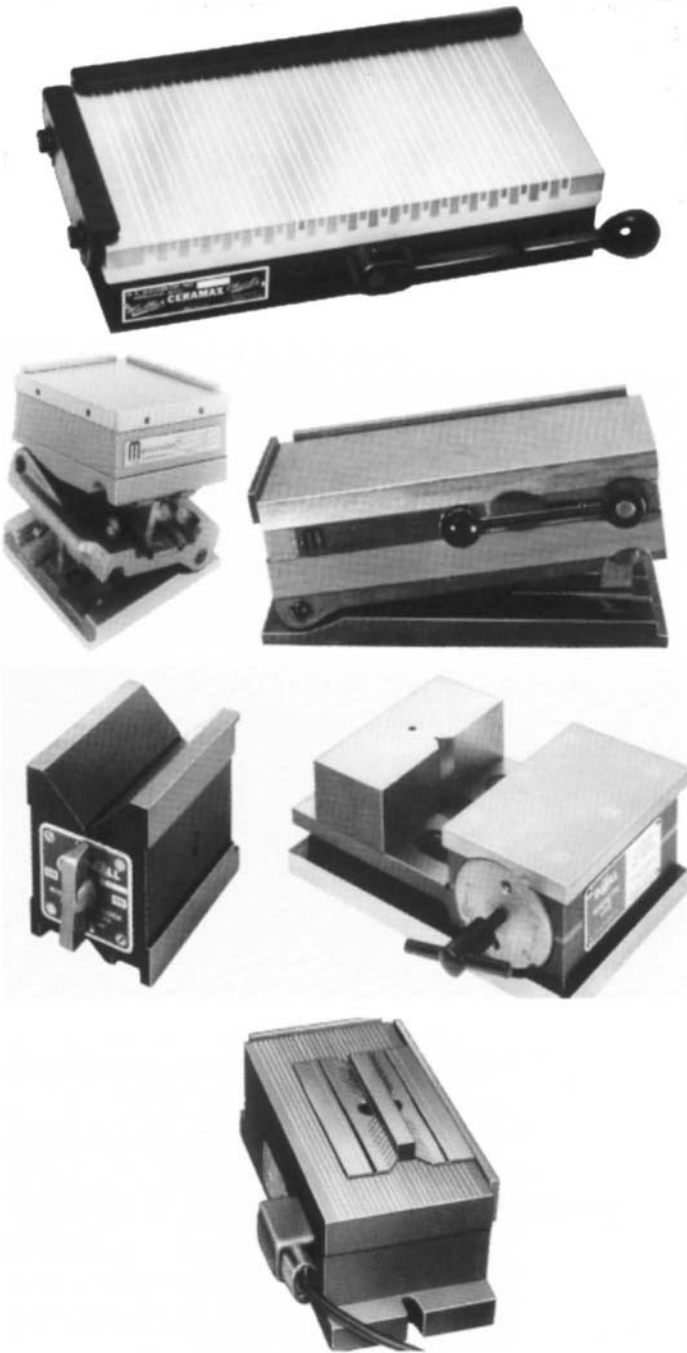
When butted against the part to be held, they grab sideways and pull downward on the part to firmly grip it and hold it for grinding.

### Magnetic Parallels

For parts that have uneven shapes, steps, or other features that prevent them from lying flat on the chuck, magnetic parallels are used.

They have alternate layers of brass and soft iron. The soft iron has high permeability, allowing free flow of the magnetic holding power from the chuck to a part placed on the parallel.

Magnetic parallels are precision-made to accuracies of 0.0001 in. flatness and squareness and should be handled with the respect and care due any precision tool.



*Figure 6.9. Walker Ceramax chuck, magnetic sine chucks, magnetic V-block and vise and magna-vise. Attachments and accessories simplify operation and add versatility to precision surface grinding.*

## Magnetic V-Blocks

Similar to the magnetic parallel is the magnetic V-blocks, constructed of the same materials as magnetic parallels but with a 45-degree V to hold round parts for grinding.

## Sine Chucks

Permanent and electromagnetic chucks are also available in a single or double sine-angle form.

Single angles or compound angles can be ground with these chucks using sine formulas to determine both the angle and the gage blocks to set it.

They come in various sizes, the most popular being the 5-in. sine. The smaller sine chucks normally are mounted directly on the chuck permanently installed on the machine table.

If, however, the primary use of the grinder is machining angles, then sine chucks as large as 10 in. can be permanently mounted on the table of the grinder. They are permanently installed because their weight makes it impractical to constantly mount and dismount them.

## Rotary Chucks

Rotary chucks are mounted on reciprocating tables to hold round parts for grinding. When mounted and in use, the table must be locked to prevent it from moving, and the rotary chuck in effect converts the grinder to a rotary table grinder.

It is a particularly good attachment for a grinder with a horizontal wheel.

## Cylindrical Grinding Attachment

The cylindrical grinding attachment allows cylindrical grinding to be performed on a surface grinder, and it is exceptionally good for grinding punches, pins, and other small parts. It is mounted perpendicular to the wheel and fed crosswise for cylindrical grinding (Figure 6.10).

It also can be tilted and put on an angle for cylindrical angle requirements. The angle is a sine angle and is computed with the sine tables, set with gage blocks, and then locked in position.

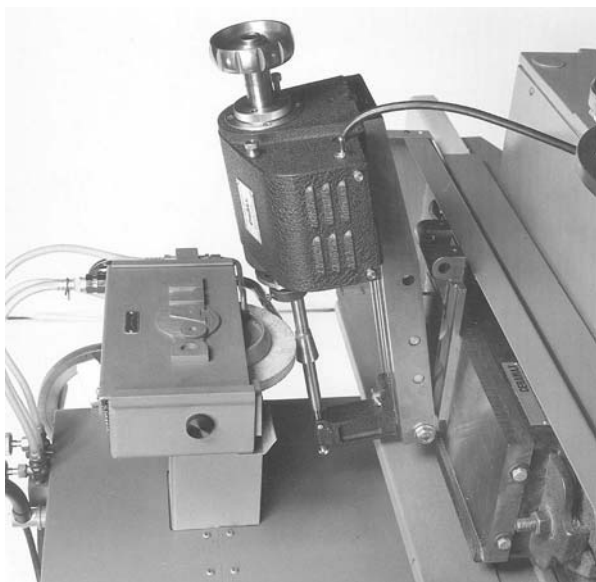
If mounted parallel to the grinding wheel it would not be used as a cylindrical grinder but can be indexed to grind slots in the workpiece.

An additional attachment is available with the cylindrical grinding attachment that allows centerless grinding of small parts.

## Radius and Angle Wheel Dresser

This attachment allows for the dressing of the wheel to virtually any form required for a part.

The diamond is mounted on a shaft that is located in the dressing tool. It pivots in an arc or radius to form the shape desired on the face of the grinding wheel. The



*Figure 6.10. Cylindrical grinding attachment with sine-bar capabilities. This allows us to do small cylindrical grind of punches and pins on a surface grinder. It can also be indexed to do hexagons and other equally spaced sections. (Courtesy of DoAll.)*

pivoting and cross-slide capabilities permit grinding convex and concave shapes as well as angles.

Another type of form dresser is one that uses a template and a stylus. The point of the stylus follows the shape of the template while actuating the diamond dressing tool located on the face of the wheel. Linkage causes the dresser to follow the form of the template and dress the wheel to that form.

There are other types of form dressers, such as an optical form dresser where the outline of the wheel face is projected onto a graduated reticule and viewed through a microscope. The grinding of the shape is then controlled visually through the lens of a microscope.

## Crush Grinding

Finally, for production form grinding, rolls of very hard material are shaped on a lathe and then placed in an attachment on the surface grinder where the wheel can then be fed over these forms. This is called *crush grinding* and is very handy when production parts are required.

## Coolant and Dust Collection

Coolant is very important in grinding, and the method of application of the coolant is critical.

This accessory typically consists of a reservoir, pump, and piping that applies fluid to the grinding wheel from the rear, with the coolant flooding the work. This is effective but only does a fair job of controlling the build-up of heat generated during grinding.

A more efficient method is the application of the cutting fluid through the wheel by means of special adaptors and tubes running to the outer surface of the wheel.

The fluid enters the wheel and is transferred through the wheel by centrifugal force and applied directly to the cutting edge of the wheel.

This controls about 70% of the heat generated, permitting high metal-removal rates without the danger of heat damage to the work.

If a large part is being ground, then additional flood can be used to cool down the workpiece and remove any residual heat from it.

In addition to considerably reducing the heat level and thereby improving the finish on a part, the coolant comes out in mist form that allows the operator to clearly see the part as it is being ground.

Flood cooling obscures the part and that is one reason many machine shops grind their parts dry, although, in that case, dust can be a problem.

Coolant normally holds dust to a minimum. If, however, the fluid does not adequately control the dust, a dust collector must be used to prevent an operator from breathing in the very tiny dust particles, largely debris from the wheel.

## **Dust Collectors**

A dust collector is basically a powerful vacuum cleaner designed to suck in dust as it is created during grinding.

This accessory mounts on the surface grinder, and it is equipped with special hoses and connections and a filtering system to screen out the dust.

But don't be misled by "vacuum cleaner;" the latter cannot be used for this purpose—the dust collector's filtering system is much more powerful, efficient, and sophisticated and is the only recommended solution to a grinding dust problem.

Dust collectors are available in various sizes to accommodate machine and workpiece size and configurations and the type of grinding performed. If coolant is being used, then the dust collector must be capable of taking in moisture without damage to itself.

## **SURFACE GRINDING AND SURFACE FINISH**

All grinding operations share basic operating principles and have a great many finish and accuracy considerations as well as strengths, limitations, and problems in common, so there is considerable carry-over from one type to another.

But there also are important differences; acceptable practices on one may cause poor results or even be dangerous on another, so human memory being what it is, before undertaking a grinding operation that you haven't done for a while, it's sound practice to carefully review the major operating factors controlling or influencing accuracy, finish, metal removal rates, and—most important—safety.



## Flatness and Finish

When grinding a surface there are two factors to consider:

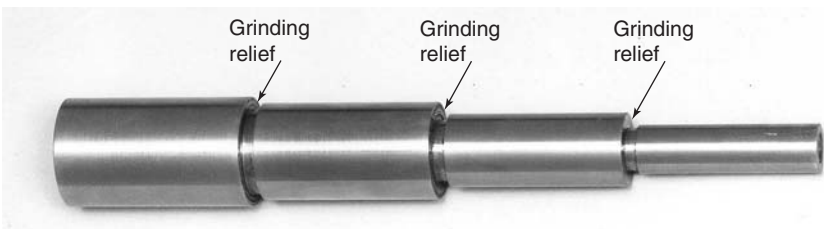
- \* Flatness of the surface.
- \* Surface finish.

To achieve flatness the following must be done.

1. The magnetic chuck must be mounted and ground according to the manufacturer's specifications and instructions (see "Mounting and Dressing in a Magnetic Chuck," later in this chapter).
2. Selection of the grinding wheel is critical. It must be one that can achieve the finish desired and of the correct bond and hardness for the material to be ground.
3. Mount and balance the wheel, then true it and dress it (see sections on mounting, balancing, truing, and dressing, earlier in this chapter).
4. Determine the amount of material to be removed by grinding.
5. If using coolant, check for quantity and cleanliness. Check filters. Always use coolant with a hydraulic machine for satisfactory results. On many hand-operated machines, coolant is not provided; it is recommended, but not necessarily available.
6. Mount the part on the magnetic chuck and energize the chuck. *Always* try moving the part manually before turning on the grinding wheel. If part cannot be moved, then it is OK. If it is possible to move the part, it must be blocked or supported to immobilize it.
7. If a hydraulic machine is being used, set the work stops on the machine so that the table traverse carries the part about 1 in. past the wheel contact point on each side. If you are a beginner or when an unfamiliar hand machine is to be used, work stops can be set to signal excessive travel to either side of the wheel contact point until you get the feel of the machine and maintain efficient traverse without them.
8. Turn on the spindle and the coolant.
9. Downfeed the wheel until it touches the high spot on the workpart. On a hydraulic machine, traverse the table back and forth while easing the grinding wheel down until it gently touches the high spot. Set the automatic traverse on the hydraulic table at about midpoint of the lever, or, if numbered, start with number 4. On a hand grinder, traverse the table manually while gently bringing the wheel down until the high spot is reached.
10. Adjust the cross-feed on the hydraulic grinder so that the wheel move crosswise about 0.025 in. after reaching the longitudinal stop previously set. The cross-feed will index automatically after tripping on the stop on a hydraulic grinder. On a hand grinder, turn the cross-feed handle about one-third of a turn after each longitudinal grind is made. This is where a beginner can make good use of the stops on a hand grinder—the stop functions as a signal to turn the cross-feed handle. Downfeed 0.002 in. with the downfeed adjustment by hand, then grind

across the part. Repeat each downfeed after completing the cross-feed grind until the part is ground flat and clean.

11. De-energize the chuck, remove the part, and check its size to see how much material remains to be removed. **Do not change any settings on the downfeed handle.**
12. Position the part on the chuck once more, this time with the finished side down and re-energize the chuck. Grind this unground side. Continue to downfeed the wheel 0.002 in. after completing each cross-feed grind until within 0.0005 in. of the finished size. Check the size and remove the last 0.0005 in. or whatever remains. This will give the desired flatness, finish, and size.
13. Before stopping the machine, turn off the coolant and run the grinding wheel without coolant to throw all the water out of the wheel; otherwise, any moisture that is left in the bottom of the wheel will cause an out-of-balance condition when it is next started up.
14. When step grinding, the same procedure applies as above until the shoulder of the step is reached. Prior to grinding, the shoulder should have a grinding relief machined to allow clearance for the wheel as it approaches the shoulder (Figure 6.11). When the wheel is about  $\frac{1}{32}$  in. from the relief, it is good practice to hand-feed the wheel the rest of the way to avoid crashing into the shoulder. When mounting the part for grinding, before starting, the wheel and part must be squared to each other to ensure straightness of cut. To do this:
  - a. Place the part on the de-energized chuck and with the wheel positioned at a level below the upper surface of the part and slide it gently until its side rests against the side of the wheel.
  - b. Traverse the wheel past the part by hand. If the part remains in the same position while the wheel moves past it, the lineup is correct; if not, repeat until it does.
  - c. Energize the chuck to hold the part in place. It is now ready to grind.



*Figure 6.11. Shoulder with relief machined for wheel clearance. Prior to grinding, machine a grinding relief on a shoulder to allow clearance for the wheel.*

## Surface Finish

No surface is exactly smooth and they all have some degree of roughness and inaccuracy. Under magnification, the surface of a part, even with a finish that

appears to be absolutely flat and smooth to the naked eye, resembles a series of jagged peaks and valleys.

Mathematically, the surface finish of any given part is both measured and described in terms of average heights and depths of these “peaks and valleys” on the surface of the workpiece.

## Specifying Surface Finish

Surface finish is specified in three areas: (1) roughness (average height), designated by the arithmetical average; (2) waviness (width and height); and (3) the roughness width (Figure 6.12).

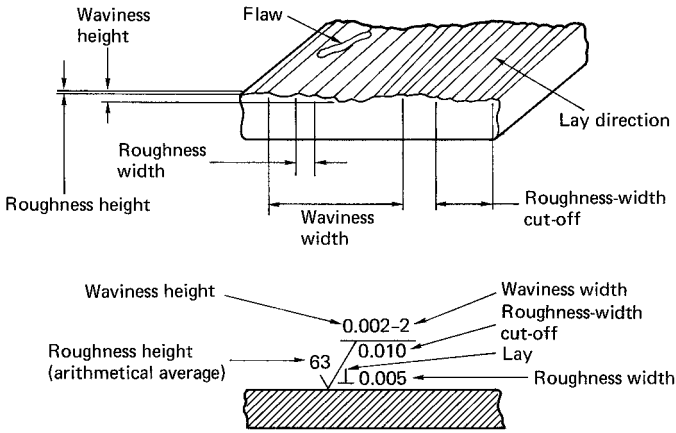


Figure 6.12. Surface finish, which is specified in terms of roughness, waviness, and roughness width.

**Arithmetical Average.** Measurements are in microinches (millionths of an inch). An imaginary centerline is imposed at a point representing the average midpoint or center of the distance between the peaks and the valleys.

These are measured for a specified area and the figures are added together and the total is then divided by the number of measurements taken to obtain the mean or arithmetical average (AA). (Roughness height is also measured in this fashion.)

**Waviness.** Waviness is determined by two measurements: (1) the height of the waviness and (2) its width. The greater the width, the gentler the slope from peak to valley, and therefore the smoother and thus more desirable the finish—the greater the width, the greater the difference between the size of the measurement units required to measure height (roughness) and those needed to measure waviness width—where height of roughness is measured in microinches ( $\mu$  in.) and waviness width is measured in thousandths of an inch.

To put that into perspective, on a scale where 1  $\mu$  in. (0.000001 in.) = 1 ft, 0.001 in. would be equivalent to 1000 ft.

**Roughness Width.** Roughness width is the measurement of a section of the peaks and valleys called *width cut-off*. This is measured by taking the square root of the width of the cut-off.

## Accuracy and Useful Life

Surface finish is important for such mating and loading bearing parts as gears, shafts, and axles for two basic reasons: accuracy and life.

The finer the finish, the less drag (friction) generated when part is put into use, and, everything else equal, the longer the potential useful life of the part. Accuracy also is tightly linked to finish—if surface finish cannot be maintained, it will be difficult or impossible to maintain grinding accuracies, especially when tolerances are specified in “tenths” (0.0001 in.).

With a good-quality surface grinder, cutting fluid and the correct grinding wheel, surface finishes of 6 microinches or better are possible.

## Finishing Guidelines

Guidelines for obtaining fine surface finishes include the following.

1. Select appropriate alloys. Material is important for obtaining a good surface finish; hardened material takes a finish better than soft material.
2. The grinding wheel should be friable, allowing the grains to break and grind cooler than tough abrasives that have a tendency to burn the material. Fine grit along with a rigid bond and a dense structure will produce better surfaces. Dressing the wheel properly is a must.
3. The machine must be rigid, accurate, and clean.
4. Using the correct cutting fluid, cross-feed, and longitudinal travel will guarantee good finishes.

## Maintaining Accuracy when Surface Grinding

Surface grinders are used in the toolroom, shop, and plant for flat grinding, form grinding step grinding, and crush grinding for every conceivable type of application. The requirements for each of these are as follows.

**Rigidity.** Sharpening dies is very common and requires good machine rigidity. If rigidity is inadequate, the wheel dips when it loses contact with the work. When the wheel on such a machine reaches the open part of a die where the punch enters, for example, as the upward pressure being exerted on the wheel by the workpiece is lost, the wheel will roll downward, dubbing the cutting edges of the die and significantly reducing its life.

Another result of inadequate or diminished rigidity is the raising of the column due to the column clearance being too great. This is reflected in an excessive

amount of sparkout, which means that less material than intended is being removed. The result is a raised belly in the center of the workpiece.

**Temperature.** If the wheel is not a suitable one—perhaps too hard a wheel for the material or one not dressed properly—then excessive heat that is likely to damage the work can be generated. Excessive heat can also result if cutting fluid is not applied properly.

In either case, the excessive heat causes the workpiece to expand and, consequently, more material is removed than intended, particularly in the center, resulting in a depression there.

With coolant properly applied, as much as 0.020 in. of material can be removed in one pass while holding accuracy to within 0.0001 in. flatness and size.

## Form Grinding

Ninety percent of all form grinding takes place in toolrooms for the various forms required on dies. The majority of the dressed wheels are made with a radius and angle wheel dresser. This is a diamond dressing attachment that forms a simple arc or radius on the grinding face of the wheel. The diamond pivots around a point on the attachment.

1. The dresser is set for (1) a concave or convex radius with a gage block that is the proper size and (2) an adjustment inward or outward of the pivot arm to allow rotation above or below the pivot point. (Follow the manufacturer's instructions.)
2. Center the diamond in the center of the wheel.
3. Feed the wheel slowly down toward the diamond.
4. Swing the dresser arm on which the diamond is mounted back and forth. Stops can be set to ensure that the arc does not overswing. This will form a convex or concave surface on the wheel, depending on whether it rotates above or below the pivot point of the dresser arm.
5. If an angle is also to be dressed, then set the rotating part to the angle required and lock it in position. A crank will now move the diamond in an angular position up or down. Position it at the proper spot on the face of the wheel and dress the angle on the wheel.
6. In all dressing, the same rules of thumb apply: Remove slight amounts until the wheel is dressed to satisfaction; use a sharp diamond dressing tool; and if you are grinding with coolant, then dress with coolant.

Once the wheel is properly dressed, it can be used to give a part that form on a surface grinder. The procedure for grinding a form differs from that for flat grinding. Instead of grinding across the work, the wheel is centered and fed directly downward. Known as *plunge grinding*, in this operation, the wheel is located directly over the part to be ground and while the table is reciprocating (traversing), the wheel is brought down directly onto the part. At the end of each stroke of the table, the wheel is lowered several thousands of an inch to remove the stock. This is repeated until the required depth is reached.

Coolant is required because, with the full width of the wheel grinding at one time, a lot of heat is generated. Coolant applied through the wheel is ideal because it emerges as mist, permitting a clear view of the part and wheel during grinding.

If more than one part is to be ground, frequent dressing of the wheel will be required.

## **Crush Grinding**

In crush grinding the shape desired on the wheel is first machined on the surface of a metal roll. The roll is usually 3–6 in. in diameter, slightly wider than the face of the wheel.

The roll is mounted on an attachment whose shaft is parallel to the spindle of the grinding wheel. The wheel and roll are brought into contact while they are both at rest, then pressure is applied at the point of contact and a drive for either the wheel or roll is engaged at a very low RPM. The wheel is then downfed until the profile of the roll is reproduced on the wheel.

This is accomplished by the crushing of the bond since the grains of the wheel are very hard. This process is usually used in high-production applications, with the rolls being made of high-speed steel.

Frequent dressing is required because of wear to the wheel, and frequent redressing of the rolls is necessary because they also wear rapidly.

Because of the pressures exerted, large rigid machines are most practical for this kind of work.

The procedure for forming a wheel is as follows.

1. Use a vitrified bonded wheel.
2. Bring the wheel and roll in contact with each other while they are both at rest.
3. While they are still at rest, move the wheel down about 0.005 in. This will exert enough pressure both to prevent slippage and to create a form. Slippage can distort the crush roll.
4. The roll or the grinding wheel should rotate at about 250–300 RPM.
5. Start downfeeding the wheel into the roll at a steady pace so no slippage occurs. Continue until the full depth of the roll is reached and then back the wheel out. Do not let it stay in the roll; it may lose its shape.
6. When grinding smaller parts, a slow table travel is best for taking full depth cuts. Larger parts can be ground with high table speeds, using automatic downfeed to remove small amounts on each pass.

## **Mounting and Dressing in a Magnetic Chuck**

Unless the magnetic chuck is properly mounted on the surface grinder, it will be impossible to achieve grinding accuracy. It's demanding, but is only necessary when setting up a new grinder or installing a new chuck to replace a worn out one.

In-transit damage aside, a new grinder is accurate to the manufacturer's specifications. Normally, the chuck is purchased separately, either from a supplier or a chuck manufacturer. When the new chuck arrives, check the packing list to be sure everything is there and then remove all kosmolene or other rust preventative and deburr the chuck to ensure contact with the table.

To install the chuck, use the following procedure.

1. Mount the chuck on the table of the grinder and block it at each end. Do not clamp it down with clamps.
2. Use a coarse, vitrified grinding wheel with a medium bond and dress it coarsely (that ensures a cooler grind).
3. Using plenty of coolant, grind the surface of the chuck, taking no more than 0.0005 in. at a time. Use a slow cross-feed and a relatively high table speed to minimize heat build-up. Do not energize the chuck. As in grinding parts, find the high spot of the chuck first. It takes several passes to grind the surface.
4. After grinding the top flat, release the blocking and turn the chuck over, cleaning everything before the next grind. Lay the ground side down on the table and block it so there is no movement of the chuck. Grind the bottom until it is completely flat and clean.
5. Deburr the newly ground bottom, clean the table pad thoroughly, and place the chuck on the pad with the bottom side down. Using the clamps that came with the chuck, clamp the left side of the chuck tightly. With the remaining clamp on the opposite side, finger tighten it. This will allow the chuck to expand horizontally, without buckling, if overheating occurs. With the chuck energized and coolant applied, grind the surface removing only about 0.0001–0.0002 in. at a time until the chuck is completely ground. Use a moderate table speed and a slow cross-feed. When finished, tighten the bolt previously only snugged with finger pressure so the chuck is secure.

Chucks wear with the mounting and removing of parts when grinding, requiring periodic redressing of the chuck. When the chuck needs regrinding, don't remove it—just grind it using coolant and the proper wheel, removing about 0.0001–0.0002 in. at a time until the chuck is flat and accurate again.

If the grinder does not have a coolant system, a good light grease can be used—any quality shortening will do. Smear it on the surface to be ground, applying enough to provide a cooling effect, but taking care not to overload the surface. This does a very satisfactory job and will provide a good finish for the chuck.

When the grinder and the chuck are not being used, keep the chuck well oiled or greased to prevent rusting, especially in humid climates.

## CYLINDRICAL AND CUTTER GRINDING

### Cylindrical Grinding

Cylindrical grinding encompasses two basic types of grinding: (1) *external grinding*, performed on plain and universal cylindrical grinders and on the

centerless type; and (2) *internal grinding*, performed on an internal grinder or, with an attachment, on a universal cylindrical grinder.

The plain grinder and the centerless grinder are designed for production work. The universal grinder, designed more for use in toolrooms and machine shops, is much more versatile than plain and centerless machines, since both its wheelhead and its headstock can be swiveled.

## The Cylindrical Grinder

The cylindrical grinder consists of a bed, table, headstock, tailstock (footstock), wheelhead, and its controls. The headstock and tailstock (footstock) hold the workpiece in the machine and revolve it while the wheelhead feeds in and rotates. The headstock and tailstock use dead centers—they are much more accurate than live centers—to hold the workpiece, with a dog and drive plate on the headstock rotating the part.

The table is driven longitudinally back and forth, with adjustable trip dogs, set to control length of travel, reversing travel direction. The table, which can be swiveled for grinding tapers and for alignment purposes, is mounted on a lower table and rides on accurately ground ways.

The wheelhead is mounted on a cross-slide at the back of the machine and can be fed into the work automatically or by hand. It is equipped with mechanical infeed stops; this allows a controlled amount of stock to be removed. In automatic operation, the recommended incremental infeed range is from 0.0002 to 0.0008 in.

The wheelhead may also be swiveled for steep tapers, which can then be plunge-ground.

A diamond dressing attachment that clamps to the table is normally used to dress the wheel.

Although the universal grinder has the versatility required of machine tools designed for low-volume and one-of-a-kind work, it also is capable of fully automatic grinding operations. It can be fitted with an internal grinding attachment that normally is mounted on the wheelhead for internal grinding, although some are designed to mount in the tailstock. The internal attachment is usually driven by a separate motor.

There is little variation in choice of wheel type: grinding wheels used on the cylindrical grinder are all the straight type. As in surface grinding, the grade, grain, and bond is determined by the material to be ground. There is no problem in grinding shoulders, radii, and fillets.

For form dressing as well as for dressing and trueing, the recommended increment for depth of cut per pass is 0.001 in.

Surface speeds are between 5500 and 6500 ft/min. Feeds are adjustable on the cylindrical grinder, permitting the use of slower rotational speed settings to achieve the recommended surface feet per minute (SFM) rates when the wheel is new; as the wheel diameter decreases with wear, rotational speeds must be increased to maintain the recommended SFM.



Cutting fluid should always be used, with plentiful amounts applied at the cutting edge. Quality and cleanliness must be maintained.

## Plain Cylindrical Grinding

Plain cylindrical grinding is straight grinding on the periphery of the work, with the part held by centers and driven with a dog and face plate. The center holes are the same ones used when rough-turning on a lathe. After heat treatment of the part, it may be necessary to grind the center holes for accuracy; this can be done on center hole machines constructed for that purpose.

While grinding, the work is traversed by the movement of the table. Both the part and the wheel rotate counterclockwise, with the part rotating slowly while the wheel rotates much faster. At the point of cut, the wheel is moving downward while the work is moving upward.

At the end of each stroke of the table traverse, the wheelface is fed inward without leaving the workpiece at either end of the part; this prevents wheel marks and helps ensure a smooth surface.

## Plain Grinding Procedure

The procedure for plain grinding is as follows.

1. After truing and dressing the wheel, keep the grinding wheel going, to warm up the spindle bearings.
2. Make sure the centers and center holes of the workpiece are clean and free of any damage.
3. Check the headstock and tailstock for alignment, using a test bar and indicator.
4. Mount the workpart between centers with a dog and the drive plate.
5. Set the speed of the wheel according to the condition of the wheel and check to ensure that the wheel and headstock rotation are correct.
6. Set the automatic infeed of the wheel to about 0.0002 in. and to the number of infeeds necessary to finish the work. The machine is equipped for a dwell at the end of its operation to spark-out and ensure size. The dwell time can be set by the operator.
7. Set the table traverse speed. The rule of thumb is about one-half the wheel width per revolution of the wheel. For a finer finish or final traverse, slow the traverse to about one-quarter the width of the wheel. Set the traverse so that the wheel will reverse when half of the wheel reaches the end of the traverse each way. Do not let the wheel leave the work.
8. Move the wheel until it just touches the work as evidenced by a slight spark. Turn on the coolant and engage the feed lever. Grind until the work is completely cleaned up. Check the size and readjust the number of infeed movements for the correct amount of material to be removed.
9. The infeed motion of the wheel will stop automatically when the correct size is reached.

## Taper Grinding

When grinding tapers, refer to the *Machinery's Handbook*, pp. 1739–1752 (22nd edition). These are standard tapers. The setting of the taper is in inches per foot. If tapers are referred to in decimal units, these must be converted to fractions and set according to the make of the grinder.

Several commonly used methods of grinding tapers are as follows.

1. Swiveling the table to the required tapers per inch, as shown in *Machinery's Handbook*. The grinder table is calculated in degrees or tapers per inch.
2. The tailstock can be offset and is calculated only in tapers per inch.
3. Taper attachments, similar to lathe taper attachments, can be used.
4. Swiveling the work head for steep angles (tapers) and plunge grinding.
5. Dressing the wheel to the angle required and then plunge grinding.

All must be checked after settings are made to ensure that the correct taper is being ground. A taper micrometer, ring gage, or taper gage must be used to check this.

When shoulder grinding, the procedure for grinding is the same as cylindrical grinding except that the wheel should be fed inward manually to minimize the possibility of ruining the part. A flared cup wheel is recommended to perform shoulder grinding as close to the headstock as possible to ensure maximum rigidity for the end thrust. The wheel should always be trued and dressed before grinding.

## Form Grinding

Cylindrical form grinding is similar to form grinding on a surface grinder in that the grinding wheel is shaped to the contour required. When grinding, the wheel is plunge fed. The main difference between the two types of form grinding is that in surface grinding, the forms are usually for tools that will be used to produce parts, whereas normally the purpose of form grinding cylindrically is to grind parts for such production assembly applications as (1) crankshafts, (2) crankpins, (3) gears, (4) threads, (5) camshafts.

## Internal Grinding

Internal grinding on the universal grinder is mainly done in the toolroom with the attachment mounted on the wheelhead. This is an advantage because, if necessary, both external and internal grinding can be accomplished in one setup.

The work is mounted in a chuck, in a collet, or on a face plate. The face plate method is the best one to use for parts that are thin and easily distorted. Parts mounted on a face plate must be firmly clamped to the plate and gaged with an indicator to ensure accuracy.

Heat is a big factor in grinding, especially so when grinding thin parts. Coolant must be used and plentifully applied; it must be clean and of proper type for the work.

The procedure for internal grinding is as follows.

1. Mount the workpiece in either a chuck or on a face plate. If on a face plate, be sure to indicate the part in the location wanted.
2. Mount the proper spindle in the quill of the internal attachment. The proper spindle is one with the correct speed range for the size of the grinding wheels to be used. Where several wheels are in the correct range, use the largest one. **Do not use a grinding wheel recommended for slower speeds and a spindle for speeds higher than the wheel requirements.** The grinding wheel should enter the work easily. The spindle should be as rigid as possible for the work being ground.
3. The height of the spindle must be adjusted so that its center is in line with the center of the work.
4. Dress the grinding wheel and set the speed to 5000–6000 SFM.
5. Set the headstock speed to 150–200 SFM.
6. Set the table travel so only about one-third of the wheel overlaps the ends of the work at each end of the stroke.
7. Start the grinding wheel and the headstock and touch the wheel to the work lightly. Turn on the coolant. Clean grind the hole and check the size to determine the amount of material to be removed. Set the automatic infeed for the number of strokes necessary to finish grind the part, removing about 0.0002 in. with each pass.
8. Grind until the hole is to size. Check for size and see if any taper occurred. If so, correct by additional grinding for the right size and straight hole.

When grinding a tapered hole, the procedure is the same as above except that the taper is set on the table. Remember that half the angle should be set to achieve the full taper.

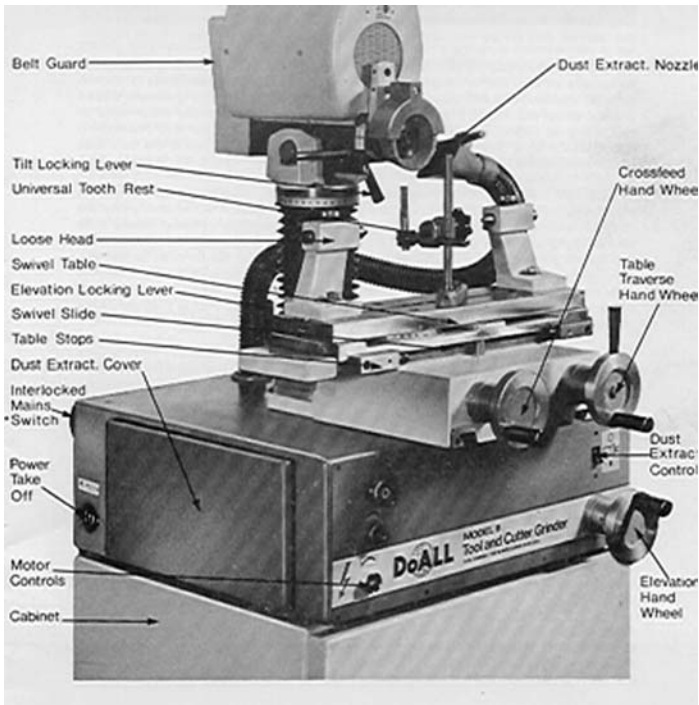
## Tool and Cutter Grinding

### Tool and Cutter Grinder

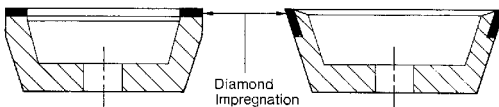
The tool and cutter grinder has become an integral part of the machine shop because of on-going production needs. Cutting tools such as milling cutters, slitting saws, side and face cutters, end mills, reamers, whether fabricated from high speed materials or carbide, have become very expensive and resharpening cuts costs by extending their useful tool lives. Despite the fact that numerical control cutter grinders have entered the picture, the basic tool and cutter grinder will continue to be used and may even remain the mainstay in most shops.

The principal parts of the machine are shown in the following illustration.

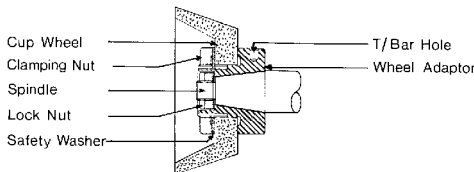
The following description explains some of the functions of each component.



*Tool and Cutter Grinder (Courtesy of DoAll).*



All grinding wheels should be mounted on a wheel adaptor which is secured to the tapered location on the front end of the spindle. The adaptor is arranged to receive wheels having 31.75 mm (1¼") dia. bore and up to 13 mm (½") thickness. Before mounting grit wheels, ensure that a wheel has cardboard clamping rings on both sides to prevent crushing of the wheel when tightening in position, and that the steel safety washer, which is keyed to the adaptor, is in position. This is to prevent the clamping nut unscrewing.



7

*(Courtesy of DoAll).*

The wheelhead provides two spindle speeds (3,500 and 5,200 rpm). These two speeds accommodate a variety of grinding wheels. The head can be lowered or raised vertically; the vertical movement is indicated by a micrometer with 0.001 in. or 0.02 mm adjustments.

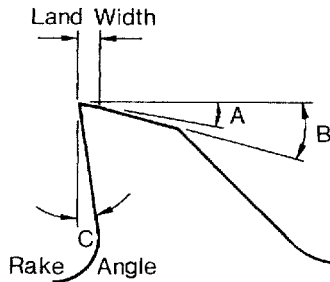
The work (or swivel) table can be arranged to swivel through 180 degree. Setting the swivel point off-center assists in the grinding of awkwardly shaped cutters.

Wheels used are cup and dish wheels. The standards are aluminum oxide and for carbide tools, diamond impregnated wheels. Depending upon the type of tools being sharpened, either a flaring cup [used for general purpose grinding] or a taper cup [for a finer finish] is available. The wheels are depicted in the sketch below. The wheels are mounted on a wheel adapter, which in turn is secured to the tapered section on the front end of the spindle.

The depth of cut should not exceed .005 in. or .012 mm.

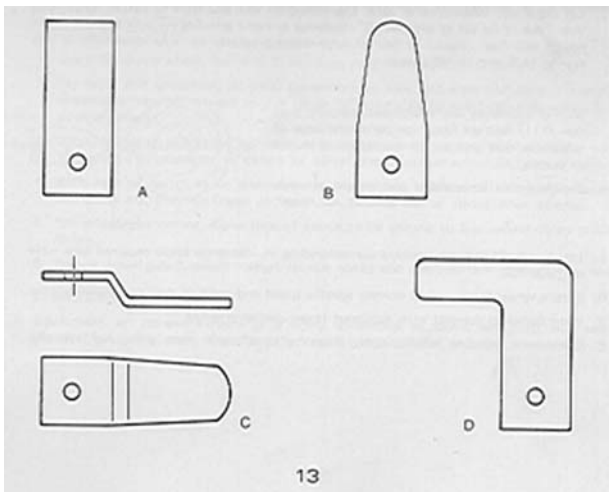
The grit cup wheel is the one most frequently used in tool and cutter grinding. It is advisable to dress this wheel with an angle on the face so it presents this wheel with a single edge to do the work, thus avoiding an excessive amount of wheel in

Most cutters have two clearance angles, a primary (A) and a secondary (B).



The main function of the secondary clearance is to relieve the primary clearance after repeated grindings have increased the width of the land.

*Clearance Angles (Courtesy of DoAll).*



*Fingers (Courtesy of DoAll).*

contact with the cutter since that can result in excessive heat that can distort and damage or ruin the tool.

When using diamond wheels, do not try to dress the wheel but check the wheel with a dial indicator for face and peripheral run-out. Run-out should not exceed 0.003 in. (0.008 mm) on any particular wheel.

Clearance angles are an important factor in the grinding of all types of cutters. Usually there are two clearance angles, a primary and secondary.

Tooth rests and fingers are used in the majority of operations. These are located as close as possible to the holder to ensure maximum rigidity and lend support to the cutter tooth during grinding. The finger should also be as near as possible to the grinding wheel without touching but never more than  $\frac{1}{32}$  in. (0.8 mm) away.

The following basic shapes will serve as a guide.

- A. *Straight blade.* Used for grinding all types of straight fluted cutters such as side and face cutters, end mills, reamers, etc. It should be capable of flexing to allow for moving from one tooth to another.
- B. *Radiused end.* Used for grinding all types of spiral fluted cutters.
- C. *Offset.* Used for certain spiral cutters and for sharpening of peripheral teeth of large cutters.
- D. *Right angle or "L" shaped.* Used for thin cutters and ones with very finely spaced teeth that will not accommodate the straight blade (a).

## Tool Grinding Skill

Tool and cutter grinding is a skill acquired through many years of experience. It requires manual dexterity, the ability to determine amounts of material to remove, and knowledge of various cutting tools. It is still a manual skill. Numerically controlled cutter grinders that are easier to operate and require less skill to master than the conventional, manually operated machines are now on the market, but they are extremely expensive and, as a result, have not yet found their way into cutter grinder departments in any greater numbers.

Virtually every type of cutting tool, from plain milling cutters through end mills, slotting cutters, form relieved cutters, staggered-tooth cutters, and reamers to single-point cutters can be sharpened on cutter grinders. The most popular manual machine is the Cincinnati Number 2 and its many clones.

The parts of a cutter grinder are:

- \* The *base*, which is a heavy, cast iron box that provides the rigidity needed.
- \* The *wheelhead*, which is mounted on a column at the back of the base. It can be raised or lowered by handwheels located on either side of the base. The wheelhead may be swiveled through 360 degrees.
- \* The *spindle*, located in the wheelhead head. Speed can be varied by means of stepped pulleys to suit the size of the wheels being used.
- \* The *saddle*, which is mounted on the ways of the base and is moved in and out by turning cross-feed handwheels. These handwheels are located in the front and back of the machine, making it easy for the operator to manipulate the machine from any position.

- \* An *upper* and a *lower* table. The upper table travels along the ways on rollers, usually balls, so there is little friction. The lower table can be swiveled to position the cutter being sharpened for easy access to the grinding wheel. Adjustable stop dogs are positioned to limit table travel while sharpening.

## Attachments and Accessories

Two *tailstocks* are available to hold cutters in centers when required; these locate on the table and are locked in place with bolts mounted in the T slots of the upper table.

Other commonly used accessories include: a *universal workhead* designed to accept cutters with Morse, universal, or other tapers that can be mounted on the table; a chuck to hold cutters; and a motorized headstock for cylindrical grinding.

A *centering gage* is used to align the headstock center with the center of the wheelhead spindle.

An *adjustable tooth rest* can be fastened to the wheelhead or the table. It is also available with a micrometer adjustment.

An *air-bearing spindle* is also used for holding and manipulating the sharpening of end mills. It increases the reliability of accuracy and speeds up the sharpening procedure.

*Grinding mandrels* are much more accurate than lathe mandrels and must be used for holding cutters.

## Cutter Grinder Wheels

Three basic types of wheels are used for grinding such differing types of cutters as end mills, milling cutters, and reamers—the flaring cup wheel; the saucer or dish wheel; and the straight wheel.

The problem is manipulating the cutter grinder with the wheel being used, to allow enough clearance for the wheel to grind without damaging the preceding cutting edge.

The wheel diameters come in sizes ranging from 3 to 7 in. Manual cutter grinders have a belt change arrangement (depending on the size of the machine) for speed changes, to increase or decrease the spindle speed for the smaller or larger wheels.

**Always check the speed on the machine prior to mounting a wheel. The speed of the spindle for the smaller wheels such as the 3-in. flaring cup wheel is twice as fast as the 6-in. straight wheel. If improperly set, the wheel can explode and do damage both to the operator and to the machine.**

## Adjustable Tooth Rests

As presented in the description of the tool and cutter grinder, the set-up of the cutter and the quality of the resharpened edges depends strongly on the type and

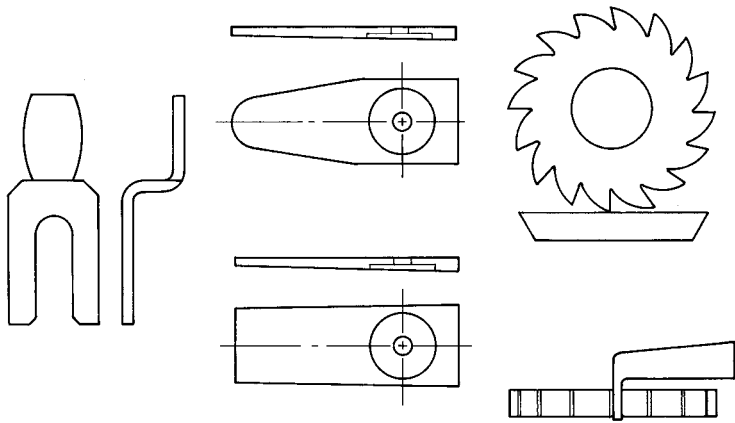


Figure 6.13. Tooth-rest shapes. These vary considerably, but are among the variations commonly used.

shape of the tool rests (fingers). These fingers support the cutter while it is being ground and are used for indexing each cutter edge for grinding.

Rest designs vary widely; there are, however, in addition to the four basic shapes depicted in the preceding segment, a fifth or “hook” type also is commonly used. The basic forms can also vary in both minor and major ways, (Figure 6.13). The fingers are usually made by the operator to suit the task at hand and can be made from any rigid, flexible, thin material such as a used hacksaw blade or flat ground stock, hardened and ground to shape. They should be able to flex to allow for tripping from one tooth to another.

The five basic shapes are:

- \* *Straight blade.* This is used for all types of straight fixed cutters, such as side and face cutters, end mills, and reamers.
- \* *Radiused end.* This is used for all types of spiral-fluted cutters. The exact shape and radius will vary to suit each application.
- \* *Offset.* This is used for certain spiral cutters and for the sharpening of peripheral teeth of large cutters.
- \* *Right angle or L-shaped.* These are used for thin cutters and for those cutters with teeth too finely spaced to accommodate a straight blade.
- \* *Hook finger.* This is used where a straight finger would impinge on other teeth. It can mount on either side of the tooth.

## Positioning the Tooth Rest

When grinding plain or straight cutters mounted on arbors or between centers with the table passing them along the wheel, the tooth rest should be mounted on the machine table.



Unless using an air-bearing spindle, when grinding helical- or spiral-tooth cutters or reamers, the tooth rest must be mounted on the wheelhead to keep it in line with the wheel as the cutter spirals over it; if an air-bearing spindle is used, the finger can be mounted on the table.

For the majority of operations, the tooth blade or finger should be located as close as possible to the holder for maximum rigidity and support for the cutter tooth during grinding.

The finger should be as near as possible to the grinding wheel. It should be  $\frac{1}{32}$  in. or closer to the wheel.

The time to sharpen a cutter is when the “wear land”—the area just in back of each cutting edge—is worn anywhere from about 0.010 in. to 0.032 in., depending on the material the cutter is being used to machine. Clues indicating that it’s time to resharpen a cutter are visual changes in the quality of the finish, high squealing, and heat build-up. In recurring production work, a specified number of pieces to be machined may be set to determine when to remove the tool for resharpening.

## Measurement

Relief angles are the angles behind and sloping down from the cutting edges of the peripheral teeth. They are critical for the machining of the various materials. There are two relief angles: (1) the *primary*, located just behind the corner of the cutting edge, and (2) the *secondary*, located just behind the primary.

The primary relief angle is for clearance of the cutting edge, but it cannot be too sharp because the tooth backing both must give the cutting edge support for strength to allow good metal removal rates and help dissipate heat to lessen build-up during machining.

The secondary relief is to give clearance to the back of the flute so the tool does not rub against the workpiece as it is removing material.

The face of the tool—the front part that runs down from the cutting edge—also has relief, and its angle depends on the material being machined.

Relief angle measurements on end mill teeth can be checked either on an optical or a template comparator or with dial indicators by what is known as the *indicator-drop method*. The indicator-drop method is inexpensive and can be done while the cutter is mounted between centers or by its shank in a rotating fixture.

The indicator-drop method uses two dial indicators, one with a sharp point to measure the relief drop from the cutting edge toward the back of the end mill tooth and the other to measure the amount the cutter is rotated (Figure 6.14).

The procedure for checking relief using the indicator drop is as follows.

1. Mount the end mill snugly between centers so it will rotate without any play.
2. With the shaft of the sharp-pointed indicator on a radial line to the mill center, set the point at the tooth edge.
3. Mount the other indicator at a 90-degree angle to the first indicator and the end mill axis near the outside diameter of the tooth face.

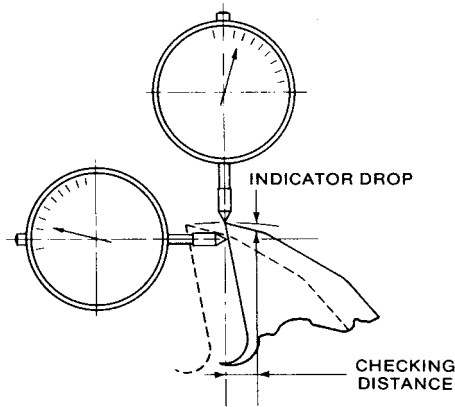


Figure 6.14. Indicator-drop method. Two dial indicators are employed in the indicator-drop method to measure relief angles when resharpening tools.

4. When checking end tooth relief angles, the same procedure can be used using a collet to hold the cutter so the end is free. The end mill can be mounted in a chuck or collet mounted in an indexing attachment and set on a surface plate.
5. Find the correct readings for flat or concave peripheral relief angles given in Figure 6.15.

## Setting the Tooth Rest

When using a straight wheel, use the following procedure.

1. Bring the center of the wheel and the work onto the same plane (Figure 6.16).
2. Fasten the tooth rest to the table of the machine and adjust it to the same height as the center of the work. Use a height gage to make this adjustment.
3. Use the graduated handwheel to either raise or lower the wheelhead (depending on the wheel rotation) the proper distance.
4. If using an adjustable tooth rest with a micrometer adjustment, you can raise or lower the tooth rest the correct distance instead of using the wheelhead. The formula to raise or lower the head when using a straight wheel is:

$$\text{Relief angle} \times \text{Wheel diameter} \times K(0.0087)$$

For example: Using a 6-in. diameter straight wheel and a 7-degree relief angle:

$$7 \times 6 \times 0.0087 = 0.365 \text{ in.}$$

Cutter Diameter (Inches)	Usual Range of Relief Angle (Degrees)	Flat or Concave Relief Indicator Drop for Relief Angles Shown		Checking Distance (Inch)
		Min.	Max.	
1/16	20–25	0.0018	0.0027	0.010
3/32	16–20	0.0017	0.0024	0.010
1/8	15–19	0.0021	0.0032	0.015
5/32	13–17	0.0020	0.0030	0.015
3/16	12–16	0.0020	0.0034	0.020
7/32	11–15	0.0020	0.0034	0.020
1/4	10–14	0.0019	0.0033	0.020
9/32	10–14	0.0020	0.0034	0.020
5/16	10–13	0.0020	0.0033	0.020
3/8	10–13	0.0025	0.0035	0.020
7/16	9–12	0.0025	0.0038	0.025
1/2	9–12	0.0027	0.0040	0.025
9/16	9–12	0.0028	0.0041	0.025
5/8	8–11	0.0028	0.0045	1/32
3/4	8–11	0.0031	0.0047	1/32
7/8	8–11	0.0033	0.0049	1/32
1	7–10	0.0028	0.0045	1/32
1 1/8	7–10	0.0030	0.0046	1/32
1 1/4	6–9	0.0025	0.0042	1/32
1 1/2	6–9	0.0026	0.0043	1/32
1 3/4	6–9	0.0027	0.0044	1/32
2	6–9	0.0028	0.0045	1/32
2 1/4	5–8	0.0023	0.0039	1/32
2 1/2	5–8	0.0024	0.0040	1/32
3	5–8	0.0024	0.0041	1/32
4	5–8	0.0025	0.0042	1/32
5	4–7	0.0021	0.0037	1/32
6	4–7	0.0021	0.0037	1/32
7	4–7	0.0021	0.0037	1/32
8	4–7	0.0021	0.0037	1/32
10	4–7	0.0021	0.0038	1/32
12	4–7	0.0021	0.0038	1/32

End relief angles are generally between 5 and 8 degrees or 0.003–0.004 indicator drop in 0.030 in. The actual value of the relief angle is normally kept within the range shown but may be varied to suit the cutter material, the work material, and the operating conditions.

Figure 6.15. Tables for readings for the indicator-drop method. (Courtesy of DoAll Co.)

Cutter Diameter (Inches)	Usual Range of Relief Angle (Degrees)	Eccentric Relief Indicator Drop for relief Angles shown		Checking Distance (Inch)
		Min.	Max.	
1/16	20-25	0.0036	0.0047	0.010
3/32	16-20	0.0029	0.0036	0.010
1/8	15-19	0.0040	0.0052	0.015
5/32	13-17	0.0035	0.0046	0.015
3/16	12-16	0.0042	0.0057	0.020
7/32	11-15	0.0039	0.0054	0.020
1/4	10-14	0.0035	0.0050	0.020
9/32	10-14	0.0035	0.0050	0.020
5/16	10-13	0.0035	0.0046	0.020
3/8	10-13	0.0035	0.0046	0.020
7/16	9-12	0.0040	0.0053	0.025
1/2	9-12	0.0040	0.0053	0.025
9/16	9-12	0.0040	0.0053	0.025
5/8	8-11	0.0044	0.0061	1/32
3/4	8-11	0.0044	0.0061	1/32
7/8	8-11	0.0044	0.0061	1/32
1	7-10	0.0038	0.0055	1/32
1 1/8	7-10	0.0038	0.0055	1/32
1 1/4	6-9	0.0033	0.0050	1/32
1 1/2	6-9	0.0033	0.0050	1/32
1 3/4	6-9	0.0033	0.0050	1/32
2	6-9	0.0033	0.0050	1/32
2 1/4	5-8	0.0027	0.0044	1/32
2 1/2	5-8	0.0027	0.0044	1/32
3	5-8	0.0027	0.0044	1/32
4	5-8	0.0027	0.0044	1/32
5	4-7	0.0022	0.0038	1/32
6	4-7	0.0022	0.0038	1/32
7	4-7	0.0022	0.0038	1/32
8	4-7	0.0022	0.0038	1/32
10	4-7	0.0022	0.0038	1/32
12	4-7	0.0022	0.0038	1/32

Figure 6.15. (cont'd)

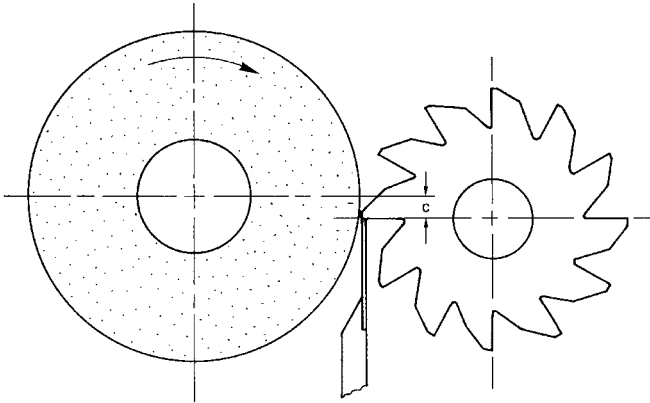


Figure 6.16. Setting the tooth rest when using a straight wheel.

Clearance angles for straight wheels are given in the following table.

Wheel Diameter (in.)	Four-Degree Relief Angle	Five-Degree Relief Angle	Six-Degree Relief Angle	Seven-Degree Relief Angle
3	0.104	0.131	0.157	0.183
3 $\frac{1}{4}$	0.113	0.141	0.170	0.198
3 $\frac{1}{2}$	0.122	0.152	0.183	0.213
3 $\frac{3}{4}$	0.131	0.163	0.196	0.227
4	0.139	0.174	0.209	0.242
4 $\frac{1}{4}$	0.150	0.185	0.222	0.259
4 $\frac{1}{2}$	0.157	0.195	0.235	0.274
4 $\frac{3}{4}$	0.165	0.207	0.248	0.289
5	0.174	0.218	0.261	0.305
5 $\frac{1}{4}$	0.183	0.228	0.274	0.319
5 $\frac{1}{2}$	0.191	0.239	0.287	0.335
5 $\frac{3}{4}$	0.200	0.250	0.300	0.350
6	0.209	0.261	0.313	0.365
6 $\frac{1}{4}$	0.218	0.272	0.326	0.381
6 $\frac{1}{2}$	0.226	0.283	0.339	0.396
6 $\frac{3}{4}$	0.235	0.294	0.352	0.411
7	0.244	0.305	0.365	0.426

When using a cup wheel, use the following procedure.

1. Fasten the tooth rest to the wheelhead and align the tooth rest and center of the cutter on the same plane, using a height gage (Figure 6.17).

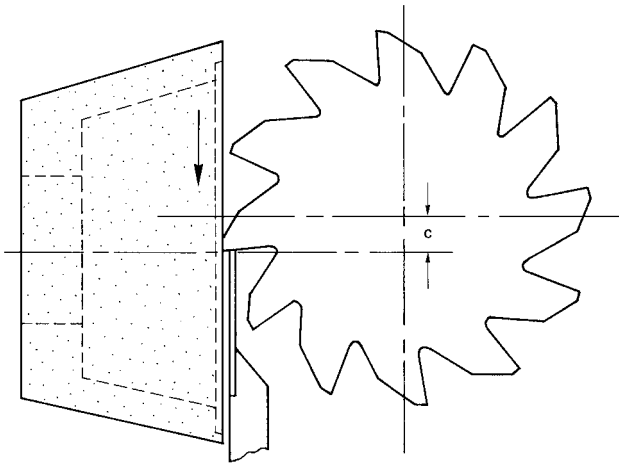


Figure 6.17. Setting the tooth rest when using a cup wheel.

2. Raise or lower the wheelhead the required distance. Use the same formula as above.
3. If using a micrometer-adjustable tooth rest, the tooth rest can be raised or lowered the correct distance instead of the wheelhead. The formula is:

$$\text{Relief angle} \times \text{Wheel diameter} \times 0.0087$$

For example: Using a 4-in.-diameter cup wheel and a 5-degree relief angle:

$$5 \times 4 \times 0.0087 \text{ in.} = 0.174 \text{ in.}$$

Clearance angles for cup wheels are given in the following table.

Cutter Diameter (in.)	Four-Degree Relief Angle	Five-Degree Relief Angle	Six-Degree Relief Angle	Seven-Degree Relief Angle
$\frac{1}{2}$	0.017	0.022	0.026	0.031
$\frac{3}{4}$	0.026	0.033	0.040	0.046
1	0.035	0.044	0.053	0.061
$1\frac{1}{4}$	0.044	0.055	0.066	0.077
$1\frac{1}{2}$	0.052	0.066	0.079	0.092
$1\frac{3}{4}$	0.061	0.076	0.092	0.108
2	0.070	0.087	0.105	0.123
$2\frac{1}{2}$	0.087	0.109	0.131	0.153
$2\frac{3}{4}$	0.096	0.120	0.144	0.168
3	0.104	0.131	0.158	0.184
$3\frac{1}{2}$	0.122	0.153	0.184	0.215

Cutter Diameter (in.)	Four-Degree Relief Angle	Five-Degree Relief Angle	Six-Degree Relief Angle	Seven-Degree Relief Angle
4	0.139	0.174	0.210	0.245
4 $\frac{1}{2}$	0.157	0.197	0.237	0.276
5	0.174	0.219	0.263	0.307
5 $\frac{1}{2}$	0.192	0.241	0.289	0.338
6	0.207	0.262	0.315	0.368

## Grinding Setups for Various Cutters

### Plain Straight Milling Cutter on the Outside Diameter

The procedure for using this cutter is as follows.

#### Equipment

- \* Universal tooth rest
- \* Centers
- \* Clearance-angle setting gage
- \* Cutter arbor
- \* 4-in. flared cup wheel
- \* Centering gage

1. Dress wheel square. Turn wheelhead 15 degrees, dress out clearance on inner taper, and leave  $\frac{1}{16}$  in. on the grinding face. Reset the wheelhead in its original position.
2. Clamp down tooth-rest base plate on table between center position.
3. Mount clearance-angle setting gage on right center.
4. Insert arbor into cutter hole and mount on centers.
5. Mount clearance-angle setting gage dog and secure it.
6. Set clearance-angle setting gage for desired primary angle on cutter and lock it in.
7. Mark a tooth for recall of starting point and set the tool-rest assembly under it.
8. Adjust the wheel height so that a tooth lines up with the centerline of the wheel.
9. Set the table travel dogs so the work clears the wheel after each grind.
10. Grind each cutting edge until all are clean and then spark out at the end. Remove only about 0.0004 in. on each pass. To prevent burning of the cutting tool or damaging the wheel, do not remove more.

### Grinding an End Mill on the Outside Diameter

The procedure is as follows.

## Equipment

- \* 4-in. flaring cup wheel
- \* Air-bearing spindle
- \* Collet
- \* Centering gage
- \* Finger
- \* Universal tooth-rest assembly

1. Mount the cup wheel on the spindle.
2. Dress the face of the wheel square.
3. Turn the wheelhead 15 degrees and dress out clearance on the inner taper of the wheel; leave a land of  $\frac{1}{16}$  in. for grinding. Set the wheelhead back to its original position.
4. Swivel wheelhead to 89 degrees (i.e., 1 degree from 90 degrees) to allow clearance for the cutter as it comes across the opposite face of the wheel.
5. Mount air-bearing spindle fixture and select collet to fit the end mill.
6. Set the tooth rest to zero with the centering gage provided with the attachment.
7. Set the tooth rest  $\frac{1}{64}$  in. from the edge of the tooth, or as close to the tooth edge as possible without hitting the grinding wheel.
8. Mark one tooth with layout ink to recall starting point.
9. Swivel the grinding collar to obtain the clearance angle. With the horizontal traverse of the table, move the table until the tooth-rest and the cutter are to the center of the cutting face of the grinding wheel. Lock in position with the table locking dogs. The movement of the cutter being ground will be actuated by the air-bearing spindle.
10. Set the wheel guard and dust collector.
11. Retract the tilt lever on the air-bearing spindle (the front collar must be set so the cutter clears the finger for indexing). Now raise the tilt lever.
12. Using the table cross-feed handles, feed the cutter inward to the wheel until sparks occur, then zero the handwheel. Grind all the flutes before touching the infeed handwheel. Feed from 0.0004 in. to 0.0005 in. for each complete cycle of grinding. Continue until cutter is sharpened.

## Face Grinding the End Mill

The procedure is as follows.

1. All the equipment is the same as given above.
2. Dress the wheel as above.
3. Set the table on a 20-degree angle.
4. Mount the air-bearing spindle fixture at a 20-degree angle.
5. Set the air-bearing spindle, using the graduated collar, at 1–3 degrees clearance so that the tooth is dished after grinding toward the center of the end mill.



6. Set, with the centering gage, the tooth cutting edge parallel to the table.
7. Mount the tooth rest and universal tooth-rest assembly and set it to the edge of the tooth.
8. Adjust the air-bearing spindle fixture to get the desired clearance angle.
9. Raise or lower the wheelhead so that the wheel centerline is aligned with the tooth edge.
10. Adjust the table travel to bring the grinding wheel edge to where it just clears the first tooth without hitting the opposite tooth.
11. Set the dogs to limit the table travel. The table will be used to pass the cutter across the grinding wheel.

## Grinding a Plain Spiral-Tooth Cutter

The procedure is as follows.

### Equipment

- \* 4-in. cup wheel
- \* Centers
- \* Cutter arbor
- \* Clearance-angle setting gage
- \* Centering gage
- \* Universal finger assemble
- \* Radiused end finger

1. Mount and dress wheel with a 15-degree inner angle, leaving a land of  $\frac{1}{16}$  in. or less.
2. Set the table to run parallel to the grinding wheel face, using an indicator to align it.
3. Reset the wheelhead in its original position.
4. Swivel the wheelhead 1 degree to allow cutter clearance on the opposite side of the wheel.
5. Set the finger to the approximate height using a centering gage.
6. Mount the cutter on the arbor and place it between centers.
7. Run the cutter over the finger to establish the contact point (marking ink on the cutter tooth helps pinpoint this).
8. Consult the clearance chart (or use the formula) for the offset required. Adjust the wheelhead by the amount indicated on the chart. (This offsets the finger since it is mounted on the wheelhead.)
9. Set the table travel dogs to prevent the cutter from running off the beginning or the high end of the finger.
10. Secure the wheel guard and position the dust collector.
11. Cross-feed in until sparking occurs on the tooth. Grind each tooth, making a pass across the wheel *without running off the finger*.

12. Return on the same tooth to the starting position, taking care to hold the cutter tooth against the finger to prevent it from rotating into the wheel.
13. Index to the next tooth and repeat the procedure.
14. After first complete grind remove 0.0004 in. of material for each grind until cutter is ground.
15. When all the teeth are ground, mount a dial indicator on the wheelhead and take a reading on each tooth. If the ends are within 0.003 in., the grind is satisfactory; if not, make the adjustment necessary to remove the taper and repeat the grinding.

## Grinding the Primary Relief on a Staggered-Tooth Cutter

This cutter presents problems other cutters do not because every tooth is set at the same but opposite angle. In addition the teeth are set close together. The traditional approach has been to use two setups, first grinding one side, then the other.

This sequence permits grinding all teeth in one setup.

### Equipment

- \* Centers
- \* Arbor
- \* Centering gage
- \* Universal tooth-rest assembly
- \* 4-in. flared cup wheel
- \* Inverted V-type finger

1. Mount and dress the cup wheel face  $\frac{1}{16}$  or less, with the head set at a 25-degree angle.
2. Mount the universal tooth rest on the wheelhead.
3. Adjust the V-type finger on the universal tooth rest assembly approximately one-third above the centerline of the mounted cup wheel; be sure the following tooth will clear the wheel. Adjust if necessary, to clear the wheel.
4. Raise the grinder wheelhead, with the finger mounted on it, so that the high point of the finger comes to zero on the centering gage.
5. Consult the chart—or use the formula—for the offset required to achieve desired relief angle. It may be necessary to use a slightly smaller angle to clear the following tooth.
6. Lower the wheelhead for the desired relief angle.
7. Set the table dogs to prevent the cutter from hitting the opposite face of the wheel.
8. Mark one tooth with layout ink to recall starting point.
9. Feed the table in until the grinding wheel is sparking on the tooth. Make a pass across the finger grinding the tooth.
10. Index for the next tooth; this will be the opposite-angled tooth. Now grind that tooth. Check these two teeth to be sure the lands are the same

size. If one land is wider than the other, shift the finger very slightly in the direction of the wider land, grinding and checking until the lands are of equal width.

11. Repeat the procedure for the other teeth and continue until the cutter is ground.

It is necessary to provide adequate chip clearance when milling deep slots; on Staggered-tooth cutters, that calls for 20–25 degrees.

Secondary clearance is ground in two setups, one for each row of equal-, but opposite-angled, teeth.

## Equipment

- \* 4-in. flaring cup wheel
- \* Two centers
- \* Wheel arbor
- \* Universal tooth-rest assembly
- \* Center height gage
- \* Flicker-type finger

1. Zero the wheel column and spindlehead—that is, zero the machine vertically and horizontally.
2. Dress the wheel until the face is square. Turn the wheelhead 15 degrees and dress the taper until the grinding face is  $\frac{1}{16}$  in. or less.
3. Reset the wheelhead in its original position.
4. Now swivel the head 1 degree so the edge opposite the one used for grinding will clear the cutter.
5. Swivel the table  $4\frac{1}{2}$  degrees from table zero and lock in place.
6. Mount the centers the proper distance apart for the mandrel holding the cutter. Positioning it on the left center holder, mount the clearance-angle setting gage. The full 20-degree range of the gage will be used.
7. Mount the cutter on the arbor and place it between the centers.
8. Select the starter tooth and mark it with layout ink.
9. Set the center height gage on the table and position it so contact is at the approximate center of the starter tooth.
10. With the tooth centered and held, mount the dog on the clearance-angle setting gage. Do not tighten.
11. While the tooth is firmly held on the height gage make sure the full 20 degrees is achieved. Since the full 20-degree range of the gage is needed, the movable base “0” should be set at 10-degrees to get a full secondary clearance.
12. Tighten the dog; the cutter, arbor, and gage are now locked together and move as a single unit.
13. Remove the center height gage.

14. Now set the clearance angle gage to 20 degrees (that is, move the base “0” to the opposite 10-degree position). Hold in the position.
15. Mount the tooth-rest assembly to the table T-slot bolt.
16. Set the assembly so that the tip of the finger is just short of contact with the center of the starter tooth.
17. Now turn the adjustable finger holder until the finger tip makes contact with the starter tooth center.
18. Remove the clearance-angle setting gage dog from the wheel arbor.
19. Use the table traverse wheel. Move the cutter toward the grinding wheel. Check to be sure the wheel will clear the following tooth. If not, adjust the wheel down until clearance is achieved.
20. Take a trial cut across the back of the starter tooth and inspect the land for uniformity of width from one side to the other. If not equal, a slight adjustment in the table swivel will correct that—swivel toward the wheel, using the wider land as a guide.
21. When the lands are given, grind all the teeth on one side.
22. Reverse the setup and grind the opposite side, again checking to make sure the lands are equal on this side. If not, adjust them.
23. Finish grinding the remaining teeth.

## Grinding the Side Teeth of a Staggered-Tooth Cutter

The sides of the teeth should be ground only when dullness makes this necessary or when the cutter width is to be reduced.

### Equipment

- \* 4-in. flaring cup wheel
  - \* Universal workhead
  - \* Stub arbor with #50 series taper shank or cutter adapter for fixture
  - \* Finger height gage
1. Dress the wheel square then dress the taper back, leaving a  $\frac{1}{16}$ -in. face.
  2. Mount the cutter on the stub arbor (#50 series). Mount the stub arbor with the cutter in the workhead spindle. Tighten the draw-in bolt.
  3. Using the center height gage, level one tooth parallel to the table plane. Once the tooth is level, clamp the workhead spindle.
  4. Mount the universal tooth-rest assembly on the bottom of the workhead. Use the support with the adjustable micrometer and a round top finger.
  5. Position the finger on the periphery of the cutter under the tooth leveled in step above. Mark tooth.
  6. Loosen upper wheelhead swivel clamp and tilt head to appropriate clearance angle. Tighten the swivel clamp.
  7. Grind clearance. Normally the clearance is approximately 1 degree of back taper. The tooth should taper from the outer diameter to the inner diameter.

When grinding form tooth cutters, a 6-in. saucer wheel is used so the wheel can pass between the face of the cutters. The wheel is ground square. The form tooth cutter is always ground on its face, never on the form.

When grinding reamers, only the chamfer is ground—never the outside diameter unless a new special-size diameter is required. The angle of the chamfer is set with the swivel table. A 4-in. cup wheel is used with a grinding face of  $\frac{1}{16}$  in.

If the outside diameter is to be ground on a reamer, the cylindrical grinding attachment is used and the table is fed longitudinally to grind the outer diameter. Light cuts are necessary to avoid burning.

## Inspection

The main item to inspect on a newly sharpened cutter is the clearance angle on the back side of the cutting edges. This is done by hand with a clearance-angle gauge (B & S or Starret) or by means of a dial indicator, (Figure 6.18).

For each  $\frac{1}{16}$  in. width of land, each degree of clearance is approximately equivalent to 0.001 in. on a dial indicator. The standard formula for the use of a dial indicator to check the amount of drop from the cutting edge back of the end of a given primary or relief land is:

$$C \text{ degrees} = \frac{57.32H}{L}$$

where  $C$  is the clearance in degrees,  $H$  is the indicator reading in inches, and  $L$  is the width of land in inches.

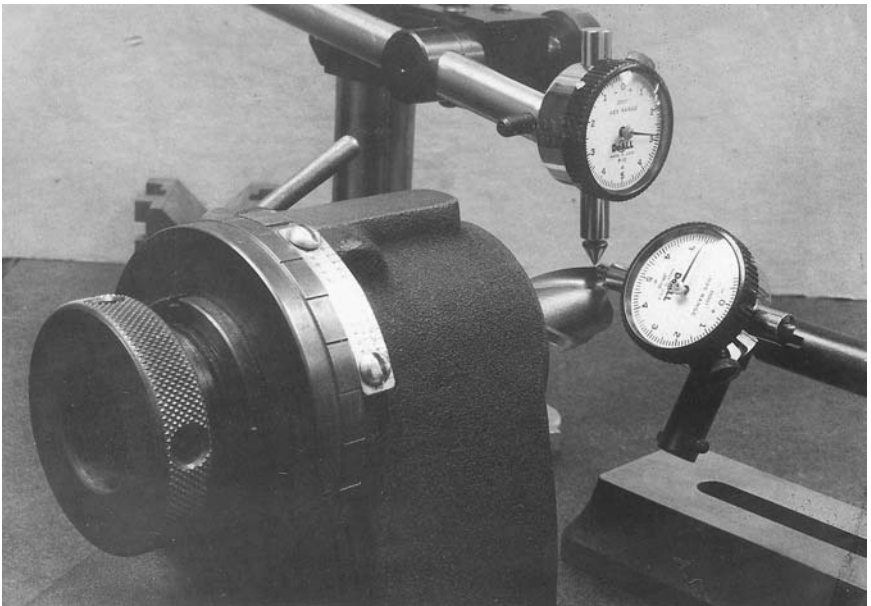


Figure 6.18. Checking clearance on a newly sharpened cutter with a dial indicator.

## Project Number 3: Toolmaker's Vice

### TOOLMAKERS VISE–JAWS

#### OBJECTIVE:

Using the knowledge and experience gained to make a precision toolmakers' vise with hardened "V" jaws, holding tolerances to  $\pm .001$  in. where required.

Review safety procedures for above machines.

Follow cleanup procedures at end of section.

#### TOOLS:

Parallels (2 – 1 × 3/16 × 6 in.)	Layout Ink
Edge/Center Finder	Scriber
7/16 in. 2-Flute End Mill	Center Punch
1/2 in. 4-Flute End Mill	Combination Square
17/32 in. Drill	6 in. Scale
3/16 in. Drill	5/8 in.-8 Tandem Acme Tap
13/32 in. × .250 Counterbore	1/4 in.-20 Tap
#7 Drill	# 10–24 Tap
2 "C" Clamps	0–1 Depth Micrometer
1/4 in. Drill	#360 Coolant/H <sub>2</sub> O 10/1
File	90° Angle Plate
Ink Remover	Tap Wrench

#### STOCK:

1–2-1/4 × 2-1/2 × 4-1/4 in. Long Cold Drawn Flat #1018 Green.

#### CUT-OFF SAW

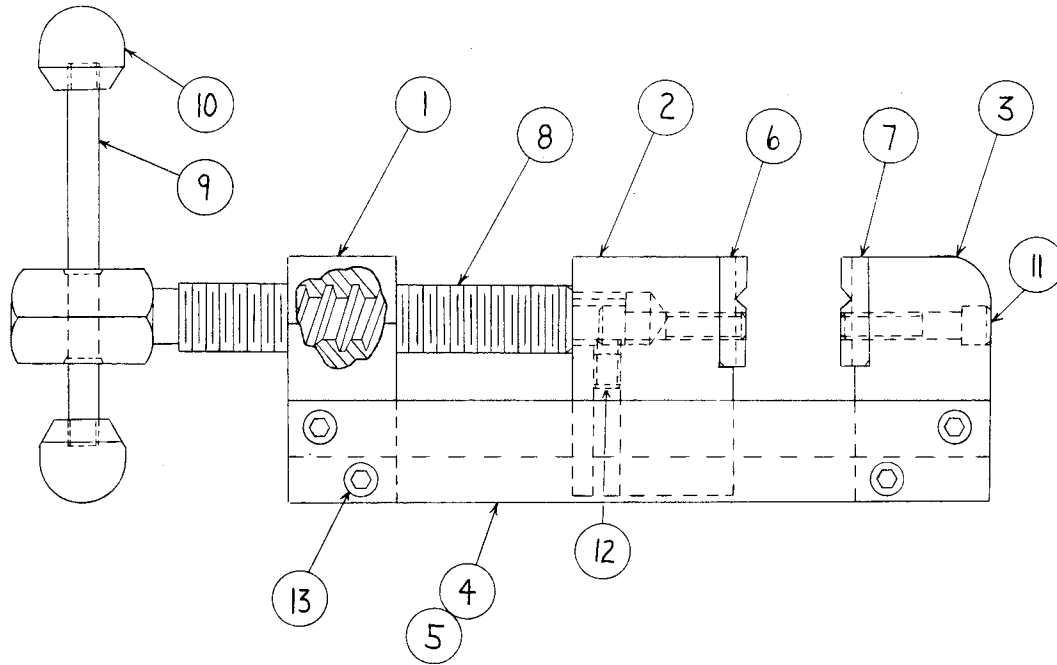
#### LATHE

#### MILLING MACHINE

#### SURFACE GRINDER

### PROCEDURE

1. Cut off 4-1/4 in. long. HD600/H<sub>2</sub>O 5/1. Leave Green Code.
2. Deburr 12 edges; deburr 4 finished sides on sanding plate.
3. Clean with ink remover. Placed on end on a rag, apply 3 light coats layout ink on 2-1/4 in. sides.
4. Scribe lines for .515 × 5/8 in. slot, .952 in. up from bottom 2 sides.
5. True up fixed vise jaw using indicator bolt tight and recheck.
6. Mount workpiece in vise on 3/16 × 1 in. parallels with slot layout toward fixed jaw. Both parallels must remain tight, After vise is tightened, remove vise handle.
7. With edge finder, locate slot edge of block. Feed workpiece in -1/2 D of edge finder +.952 in. -1/2 of slot width. \_\_\_\_\_ in. Lock cross feed.
8. Insert 2-flute 7/16 in. mill (tighten set screw on flat). Lock quill rough mill slot .200 in. on a pass .600 in. deep by raising table. Use coolant. \_\_\_\_\_ RPM \_\_\_\_\_ IPM. Check with depth micrometer.
9. Rough mill opposite slot by repeating details #6 and #8



TOLERANCES: DECIMALS  $\pm .001$   
 FRACTIONS  $\pm \frac{1}{64}$

DoALL TECHNICAL INSTITUTE		Material AS NOTED	Scale Qty. —
Part ASSEMBLY-TOOLMAKERS VISE			
Dr. A.G.K.	Date 2-6-80	Part Number	

DETAIL	QUANTITY	MATERIAL	ROUGH SIZE
1	1	AISI 1018	2 1/4 x 2 1/2 x 4 1/4
2	1	AISI 1018	CUT FROM DET #1
3	1	AISI 1018	CUT FROM DET #1
4	1	AISI 1018	1 x 1 3/4 x 6 5/8
5	1	AISI 1018	SEE DET. #4
6	1	O-1TL.STL.	1/4 x 1 x 2 3/8
7	1	O-1TL.STL.	SEE DET #6
8	1	AISI 1018	3/8 HEX x 6 1/2
9	1	AISI 1018	5/16 DIAM. x 3 3/8
10	2	AISI 1018	3/4 DIAM. x 1 3/4
11	4	COMM.	1/4-20 x 1 S.H.C.S.
12	1	COMM. (NYLOCK)	1/4-20 x 3/8 DOG PT. SET SCREW
13	8	COMM.	#10-24 x 1 7/8 S.H.C.S.

DoALL TECHNICAL INSTITUTE	Material AS NOTED	Scale — RES. —
Part BILL OF MATERIALS TOOLMAKERS VISE		
Dr. A.G.K.	Date 2-5-80	Part Number:

**TOOLMAKERS VISE—JAWS (Cont.)**

10. Insert 1/2 in. 4-flute milling cutter and finish slot .500 x .625 in. deep. One pass.  
\_\_\_\_\_ RPM \_\_\_\_\_ IPM.
11. Without changing table setting, mill opposite side.
12. Measure slot. Unlock table. Feed table out to machine .515 in. slot. Relock and mill .515 in. dim. Do not unlock table.
13. Repeat for opposite side.
14. Unlock cross feed. Move table out to mill off 7/16 in. shoulder until slot is .250 in. ±.001 in. deep. Use 0–1 depth micrometer.
15. Repeat milling operation on opposite side. Move parallel.
16. Deburr slot edges both sides of workpiece. Start rails.
17. See section on how to make “Rails.” Mark rail and matching slot before proceeding to element # 18.
18. Set workpiece between magnetic parallels on 5/8 in. surfaces. Block end of piece. Grind top surface to clean up.



**TOOLMAKERS VISE—JAWS (Cont.)**

19. Place workpiece on angle plate, ground side to plate, using adjustable parallel in 5/8 in. slot. Raise workpiece above the angle plate. Use an indicator in slot bottom to level. End of workpiece should extend to one side only. Secure two clamps tight.
20. Grind one side of workpiece to clean up.  $.625 \text{ in.} \pm .001 \text{ in.}$
21. Reset one clamp at a time. Set angle plate on its side. Block support on chuck. Do not let part move on angle plate.
22. Grind end to clean up.
23. Grind 3 opposite sides parallel to ones already ground. (Bottom to  $.952 \text{ in.} + .000 \text{ in.} - .005 \text{ in.}$ ) Block piece on chuck.
24. Deburr all edges, including slots.
25. Lay out 1-1/4 in. from end of workpiece and saw off stationary jaw.
26. Grind jaw to 1-1/4 in. Grind remaining piece to clean up.
27. Lay out lead screw jaw. 1-17/32 in. dim. is approximate. Hole *must* be on *centerline* of workpiece  $\times$  1-11/16 in. dim.
28. Drill 17/32 in. hole, 1-1/4 in. deep.
29. Cut off lead screw jaw and grind to 1 in. dim. Chamfer 17/32 in. hole and tap Acme 5/8-8. Mill  $45^\circ \times 5/8 \text{ in.}$
30. Mill sawed face, then grind moveable jaw to 1-1/2 in. Chamfer 17/32 in. hole.
31. Go back to "Rails," detail #16.
32. Mill  $1 \times 1/8 \text{ in.}$  step inner face 2 jaws.
33. Drill 17/32 in. hole, 7/8 in. deep, in moveable jaw and chamfer.
34. Drill #7 set screw hole on centerline of moveable jaw.
35. Drill 17/64 in., 1 in. deep.
36. Tap 1/4 in.-20 through.
37. Mill moveable jaw to  $.890 \text{ in.}$
38. Mill 3/8 in. R on nose of stationary jaw if desired.

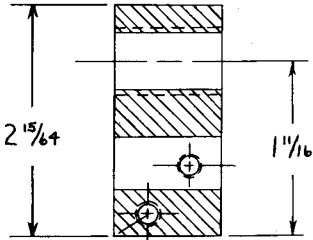
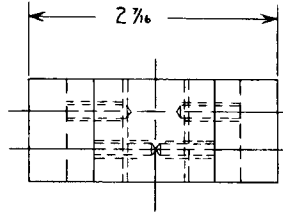
**CLEANUP**

*Clean* all machines, floors and toolracks immediately after use and before proceeding to next operation or machine.

*Return* all tools to designated locations.

*Report* any breakages or repairs needed.

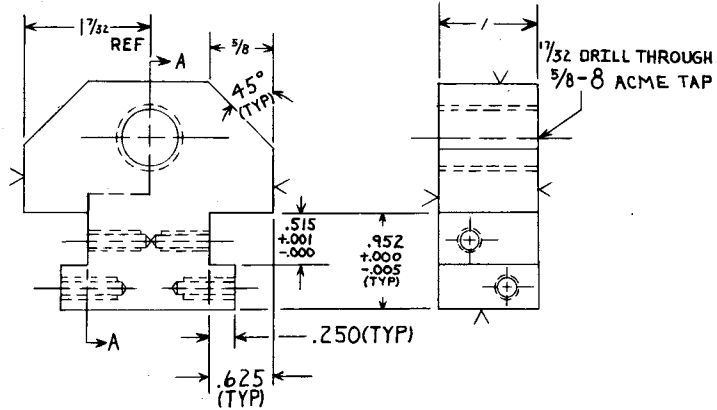
DETAIL# 1 LEAD SCREW BLOCK  
1 REQ'D - BREAK ALL CORNERS



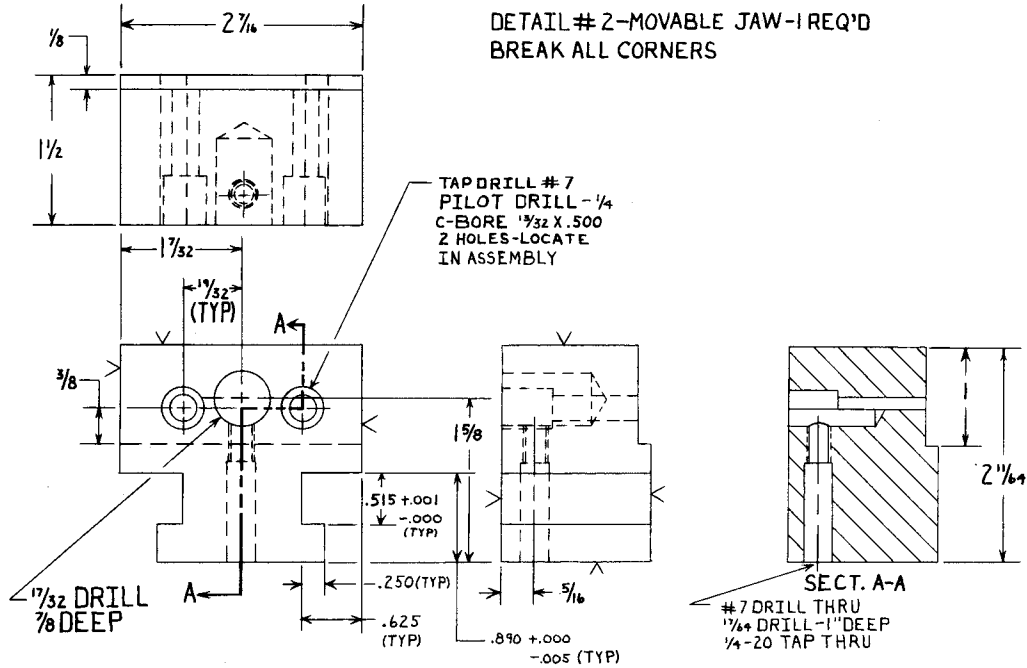
SECT. AA

#25 DRILL - 3/16 DP. - #10-24 TAP - 5/8 DP.  
(4 PLACES) - LOCATE AT ASSY WITH  
DETAILS # 4 + # 5

TOLERANCES: DECIMALS ± .001  
FRACTIONS ± 1/64

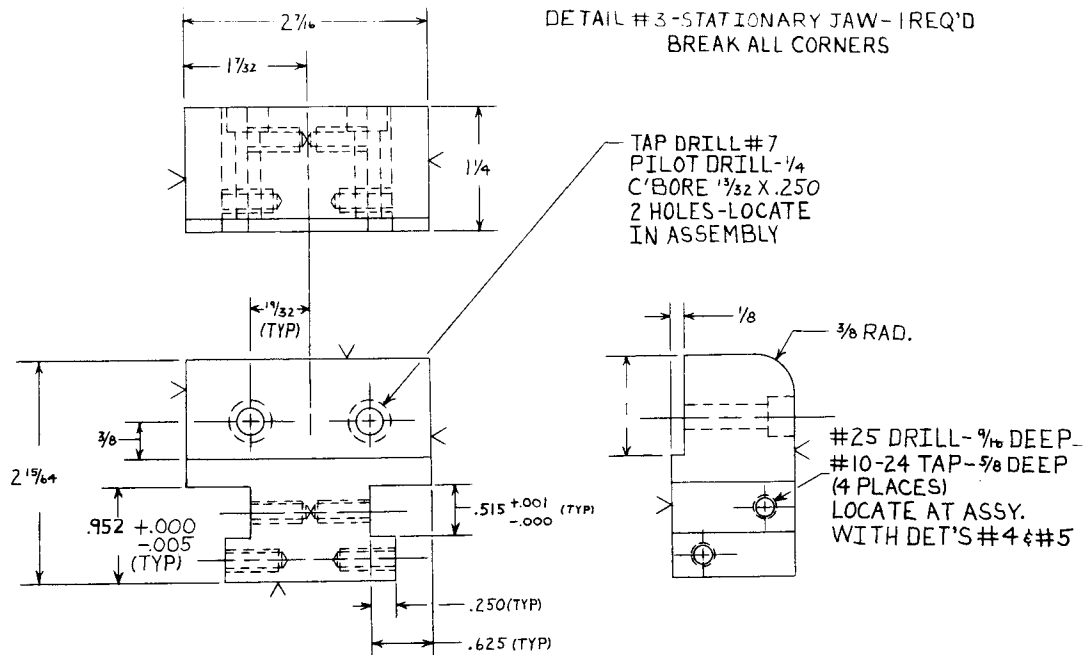


DoALL TECHNICAL INSTITUTE		Material AISI 1018	Scale 1:1 QTY.
Part LEAD SCREW BLOCK			
Dr. A.G.K.	Date 1-30-80	Part Number	



TOLERANCES: DECIMALS ±.001  
FRACTIONS ± 1/64

DoALL TECHNICAL INSTITUTE	Material AISI-1018	Scale   :   Qty.   1
Part MOVEABLE JAW		
Dr. A.G.K.	Date 2-1-80	Part Number



TOLERANCES: DECIMALS  $\pm .001$   
FRACTIONS  $\pm \frac{1}{64}$

DoALL TECHNICAL INSTITUTE		Material AISI-1018	Scale: 1:1 Qty. 1
Part STATIONARY JAW			
Dr. A.G.K.	Date 2-4-80	Part Number	

## TOOLMAKERS VISE-SIDE RAILS

### TOOLS:

Shell Mill	Center Punch
1/2 in. End Mill	Flat File with Handle
3/16 in. Drill	0–1 Depth Micrometer
21/64 × .190 in. Counterbore	0–1 Micrometer
#25 (.1495 in.) Drill	#10–24 Tap
#11 (.191 in.) Drill	

### STOCK:

1 × 1-3/4 × 6-3/4 in. Cold Drawn Steel

Review mill safety steps.

Follow cleanup procedures at end of section.

## PROCEDURE

*(Calculate speeds and fill in blanks before starting.)*

1. Secure 1 × 1-3/4 in. Cold Drawn Steel 1018 Coded Green cut to rough length 6-3/4 in. \_\_\_\_\_ SFM \_\_\_\_\_ Pressure.
2. True up fixed jaw of mill vise and tighten down. Recheck.
3. Clamp workpiece in vise on parallels with 1-3/4 × 6-3/4 in. surface above vise. Keep parallels tight.
4. Install suitable shell milling cutter on holder in mill. \_\_\_\_\_ RPM \_\_\_\_\_ IPM.
5. Take a skin cut .010-.015 in. off top surface.
6. Turn workpiece over to skin cut other side. Mill both ends to 6-1/2 in.
7. Skin cut edges the same.
8. Install 1/2 in. milling cutter.
9. With workpiece in vise on edge, mill .260 in. ±.001 in. step leaving .515 in. +.000 in. –.001 in. oversize. Check with 0–1 in. depth micrometer. Quill is locked and table is raised.
10. When step is .250 in. ±.001 in., crossfeed table to approach .515 in. +.000 in. –.001 in. dim. Make this *fit* one groove in block. Deburr top corners so as not to gall when fitting. Mark end of slot and end of matching rail.
11. Reverse rail repeating element #10 to *fit* a rail to the other slot, marking each.
12. On vertical saw, cut off each rail from center section.
13. Mill sawed face to 5/8 in. dim. Leave .010 in. stock for grinding.
14. Lay out each rail, 1 RH, 1 LH, for screw center lines.
15. Go back to “Jaws,” detail #18.
16. Position 3 jaws in relative positions.
17. Insert rails in matching slots.
18. “C” clamp, leaving hole locations clear.
19. Drill (8) #25 tap drill holes through rails and jaws 1-1/4 in. deep.

## TOOLMAKERS VISE—JAWS (*Cont.*)

20. Remove rails and tap holes in jaws 5/8 in. deep. Use tap magic. Be sure tap is square in 2 planes. Break chips.
21. Open rail holes to .191 in. and counterbore 21/64 × .190 × 1/4 in. deep.
22. Back to “Jaws,” detail #32.

### CLEANUP

*Clean* all machines, floors and toolracks immediately after use and before proceeding to next operation or machine.

*Return* all tools to designated locations.

*Report* any breakages or repairs needed.

## TOOLMAKERS VISE—INSERTS

### TOOLS:

Tap Wrench	Layout Ink
1/4 in.-20 Tap	Scriber
#7 Drill	Combination Square
1/2 in. End Mill	Toolmakers' Parallel Clamp

### STOCK:

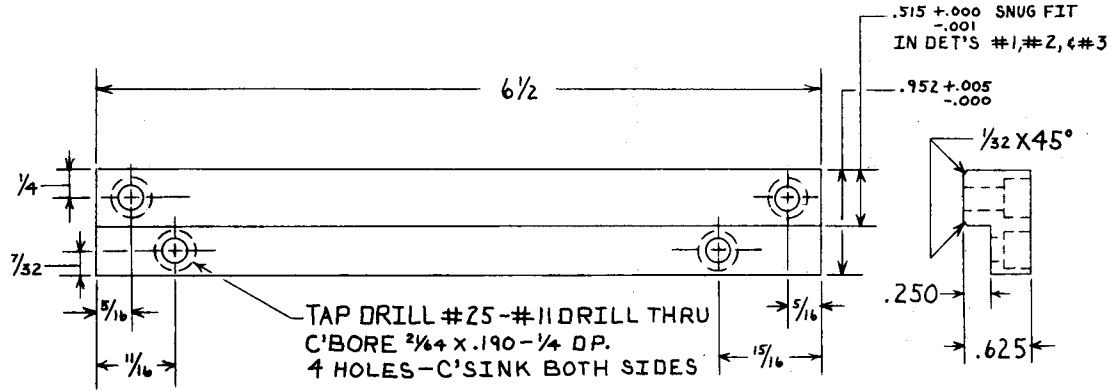
2-1/4 × 1 × 2-9/16 in. 0–1 Ground Flat Stock

4-1/4 in.-20 × 1 in. Socket Head Cap Screw

## PROCEDURE

1. Cut off to 2-17/32 in. long and file 1/32 in. × 45° one edge.
2. Lay out screw holes (2) center punch.
3. Clamp in step in vise jaw. 1/32 in. × 45° into corner.
4. Drill through insert and jaw with #7 drill, 2 holes each.
5. Mark inserts and matching jaws.
6. Tap holes in inserts.
7. Open 4 holes in jaws to 1/4 in.
8. Counterbore 13/32 × .250 in. to 49/64 in. dim. max.
9. Screw inserts into respective jaws.
10. Position jaws and inserts face to face clamping rails in position.
11. Lay out centerlines on ends and top of inserts for “V”s.
12. Tilt head 45°. Reposition vise truing up fixed jaw.
13. Clamp jaws in vise. Position edge of cutter over centerline of “V” (double check alignment before milling) raising table cut “V” .093 in. deep.
14. Repeat for other 3 “V”s.
15. Preheat furnace to 1400 F, heat inserts 60 min.

DETAIL #4 - SIDE RAIL L.H. - IREQ'D (SHOWN)  
 DETAIL #5 - SIDE RAIL R.H. - IREQ'D (SYM. OPP.)



NOTE: TAP DRILL #25 THRU RAIL  
 INTO JAWS  $\frac{1}{4}$  DP.

TOLERANCES: DECIMALS  $\pm .001$   
 FRACTIONS  $\pm \frac{1}{64}$

DoALL TECHNICAL INSTITUTE		Material AISI 1018	Scale 1:1 Qty. —
Part SIDE RAILS			
Dr. A.G.K.	Date 2-5-80	Part Number	

**TOOLMAKERS VISE—INSERTS (Cont.)**

16. Quench in oil bath. Wipe tongs of oil and cover tank.
17. Draw at 350° F, 60 min.; air cool. Shut off furnace.
18. Reassemble and grind; see "Final Grind."
19. Redraw to darken and reassemble.

**CLEANUP**

*Clean* all machines, floors and toolracks immediately after use and before proceeding to next operation or machine.

*Return* all tools to designated locations.

*Report* any breakages or repairs needed.

**TOOLMAKERS VISE—LEAD SCREW****TOOLS:**

Toolbit Ground for Acme Thread	Center Drill
Acme Center Gage	0–1 in. Blade Micrometer
Toolbit Ground for Turning	21/64 in. Drill
Toolbit Ground for Facing	Countersink
Toolbit Ground for Necking	Acme Thread Gage

**STOCK:**

7/8 in. Hex. × 6-1/2 in. Cold Drawn Steel #1018 Green

**PROCEDURE**

1. Cut hex stock to 6-1/2 in.
2. Chuck short, face one end. Turn 1/16 in. × 45°.
3. Reverse piece, chuck short, face and center drill.
4. Hold with chuck and live center.
5. Turn .623 in. ±.001 in. to 1-1/4 in. from hex end. Turn .468 in. × 1/4 in. Chamfer hex.
6. Turn 1/2 in. D to 5-1/8 in. from hex end. Turn 3/8 in., D × 1/2 in.
7. Turn Acme thread 5/8 in.-8.
8. Turn 120° point. Chuck over brass on threads.
9. Lay out, drill and chamfer 21/64 in. hole.
10. Deburr part.

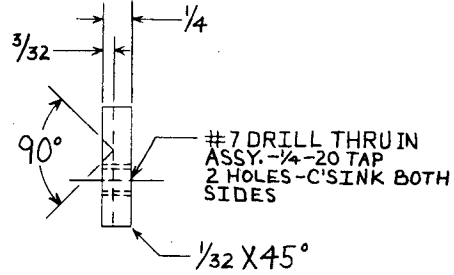
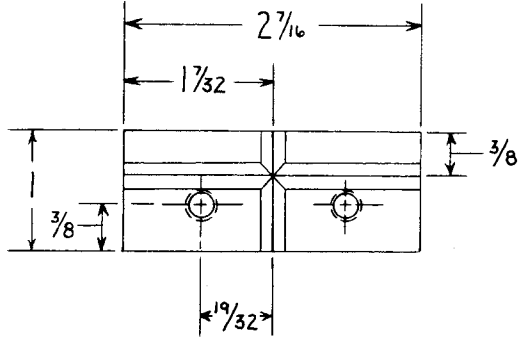
**CLEANUP**

*Clean* all machines, floors and toolracks immediately after use and before proceeding to next operation or machine.

*Return* all tools to designated locations.

*Report* any breakages or repairs needed.





DETAIL#6-JAW INSERT-1 REQ'D (SYM. OPP.#7)  
 DETAIL#7-JAW INSERT-1 REQ'D (SHOWN)  
 HARDEN-53-55 R.c & GRIND

BREAK ALL CORNERS  
 TOLERANCES: DECIMALS  $\pm .001$   
 FRACTIONS  $\pm \frac{1}{64}$

Refer to Pages 22-23

DoALL TECHNICAL INSTITUTE	Material O-1 TOOL STEEL	Scale 1:1 Qty. —
Part VISE JAW INSERTS		
Dr. A.G.K.	Date 2-5-80	Part Number

## TOOLMAKERS VISE–HANDLE

### TOOLS:

File with Handle	Center Drill
5/16 in.-18 Taps (Plug and Bottom)	Letter “F” Drill
5/16 in.-18 Stock and Die	3/4 in. Radius Gage

### STOCK:

5/16 in. D Drill Rod  
3/4 in. Cold Drawn Steel

## PROCEDURE

1. Cut 5/16 in. rod to length.
2. Thread 1/4 in. each end 5/16 in.-18.
3. Chuck 3/4 in. bar face off. Turn  $60^\circ \times 1/2$  in.
4. Center drill, “F” drill, 1/2 in. deep.
5. Tap full depth 5/18 in.-18. Use cream break chip.
6. Cut off 12/16 in. long.
7. Repeat elements # 1–#6.
8. Chuck 5/16 in. rod and screw on knob.
9. Using compound, rough out 3/8 in. R with several angular cuts.
10. Smooth out radius with file. Check with gage.
11. Polish.
12. Repeat details #8–#11 for other knob.
13. Insert rod through lead screw and tighten knobs finger tight.

### CLEANUP

*Clean* all machines, floors and toolracks immediately after use and before proceeding to next operation or machine.

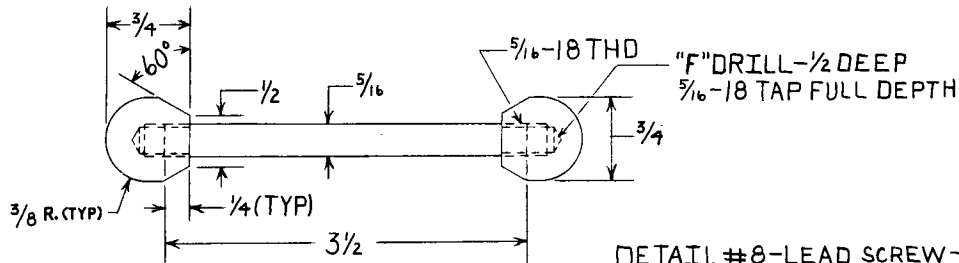
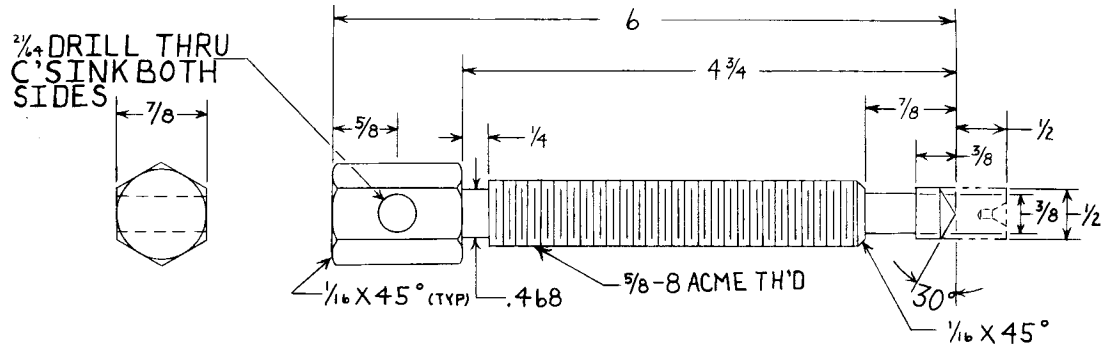
*Return* all tools to designated locations.

*Report* any breakages or repairs needed.

## TOOLMAKERS VISE–FINAL GRIND

### PROCEDURE

1. Assemble vise without lead screw. Snug, then tighten all 8 rail screws.
2. Mount upside-down with rails only resting on 2 magnetic parallels. Block end.
3. Clean up rails on bottom. OK if end block and jaw are slightly recessed.
4. Install screw, clamp vise shut and grind top.
5. Clamp vise base against angle plate. Indicate side of vise parallel to surface plate. Use brass shim, under clamps.



DETAIL #8-LEAD SCREW-1 REQ'D  
 DETAIL #9-HANDLE BODY-1 REQ'D  
 DETAIL #10-KNOB-2 REQ'D

BREAK ALL SHARP CORNERS & EDGES

TOLERANCES: DECIMALS  $\pm .001$   
 FRACTIONS  $\pm \frac{1}{64}$

DoALL TECHNICAL INSTITUTE		Material AISI-1018	Scale  : Qty. —
Part LEAD SCREW & HANDLE			
Dr. A.G.K.	Date 2-5-80	Part Number	

**TOOLMAKERS VISE—FINAL GRIND** (*Cont.*)

6. Grind on side until inserts and rail are flush with jaws.
7. Remove screw, readjust clamps, tip angle plate on side. Block on chuck.
8. Grind end (fixed jaw) and rails flus.
9. Remove from plate.
10. Grind opposite side.
11. Install lead screw and lock free with dog point set screw.
12. Spray with RPM to preserve finish.
13. Inspection.

**CLEANUP**

*Clean* all machines, floors and toolracks immediately after use and before proceeding to next operation or machine.

*Return* all tools to designated locations.

*Report* any breakages or repairs needed.

## Chapter 7

# STEELS, ALLOYS, AND OTHER MATERIALS

## INTRODUCTION

Every material has its own characteristics and peculiarities that affect both the way it performs as a tool, product, or building material and the way it machines as a workpiece:

1. *Ferrous metals.* Iron is brittle, rigid, and a good foundry and casting material that teams up with carbon to form steel, a strong, hard, and versatile metal that, when fortified by alloys, is good for use as an all-around tool, product, and building material. Both iron and carbon have machining characteristics that range from great to nearly impossible.
2. *Nonferrous metals.* Copper has good electrical and heat-transfer properties; it is soft and gummy, but has alloys as hard as brass and durable as bronze. Aluminum is light, machines well-to-terrible. Titanium is a light, extremely strong, and tough-to-cut, aerospace material, etc.
3. *Plastics.* There are infinite varieties, good for everything from cheap toys to exotic (and therefore, very expensive) “space age” materials, including metal-reinforced composites. Most are formed rather than machined, but many know the cutting edge.
4. *Woods.* Some are as light as feathers that cut with ease, others are tough and dense enough to earn the name “ironwood”—and they machine like it, too.

In just the metals alone—the basic machining materials—there are more than 33,000 separate and distinct alloys.

To be able to saw drill, bore, mill, grind, or turn such a wide-ranging collection of materials, one must know something about—and preferably understand—their structure and properties, how hard they are, and such other things as how they are affected by water, cutting fluids, and coolant.

Generally speaking, the harder the material, the more difficult it will be to machine. But soft materials can also be difficult to machine if they're gummy.

Alloying elements can markedly alter materials properties. Among the ones found in steels and in other alloys that are most commonly encountered in the shop are:

Boron(B)	Nickel(Ni)
Carbon(C)	Selenium(Se)
Chromium(Cr)	Silicon(Si)
Cobalt(Co)	Sulfur(S)
Lead(Pb)	Tungsten(W)
Manganese(Mn)	Vanadium(V)
Molybdenum(Mo)	

Most of these are added to improve the *performance* of alloys as tools or products, but lead, selenium, and sulfur are added to improve their *machinability*. With these additives, some alloys machine so much better that they're referred to as "free machining steels."

## STANDARD IRONS AND STEELS—AISI AND SAE

Standard carbon steels are listed by numbers assigned by the American Iron and Steel Institute (AISI) and the Society of Automotive Engineers (SAE). The only difference between AISI and SAE numbers is that AISI numbers may include a letter indicating the method of making that steel—B for Bessemer, C for crucible, etc. The numbers indicate the type of iron, the amount of carbon present, and the presence of major alloying elements (Figure 7.1). The first two numbers indicate the type of steel, the next two indicate its carbon content. The carbon content range is from less than one-tenth of one percent (0.10%) to one percent (1.00%). AISI and SAE 1045 steel, for example, indicates a carbon steel (the 10) with 0.45% carbon content (the 45).

Other elements either are present by design or are trace elements not removed during the steel-making process. For example, 1045 steel can have as much as 0.60 to 0.90% manganese (Mn), up to 0.040% phosphorous (P), and up to 0.050% sulfur (S). These are not indicated by the AISI and SAE number, but usually will be noted in machining recommendation tables for these materials.

Nothing's perfect; there is a spread (tolerance) in the amount of carbon and other materials present. The carbon content tolerance range for 1045 steel, for example, is 0.43–0.050%, so a batch of 1045 could have as much as 0.50% carbon and still be considered "1045."

As the carbon content increases, so does the difficulty of machining.

### 1000 Series

The standard steels start with the 1000 series and range from 1005 to 1095.

Materials over 1035 can be heat-treated to achieve various levels of hardness, and those with as much as 0.95% carbon (C) are used for carbon-steel band-saw blades, threading dies, and inexpensive drills, taps, and reamers.

**Carbon and Alloy Steels**

Type	AISI No.	Machinability		Comments	Common Uses
		Rating, %*			
Carbon	1010	50		Gummy	Sheet, nails, fence
	1020	65		Soft	Sheet, pipe
	1030	65		Machines best of carbon steels	Tubing, gears
	1040	60		Machines good	Crank, shafts, axles
	1050	50		Machines fair	Springs, hand tools
	1060	50		Machines fair	Washers, springs
	1070	45		Tough	Wrenches, knives
	1080	40		Hard and abrasive	Hammers, chisels
	1090	40		Hard and abrasive	Leaf spring, tools
Free Machining	1117	90		Machines very well	Screw machine parts
	1141	80		Machines very well	Screws, nuts, gears
	1112	100		Machines very well	Screw machine parts
	12L14	160		Machines excellent	Screw machine parts
Manganese	1360	60		Some work hardening	Axles, shafts, bolts
Nickel	2317	55		Work Hardens	Cables, plate
Nickel Chromium	3115	60		Some Ni machines fair	Armor plate
	3130	55		Some Ni machines fair	Gears
	3140	55		Some Ni machines fair	Springs
	3150	50		Some Ni tough	Shafts
Molybdenum	4063	50		Machines fair	Springs, hand tools
Chromium Molybdenum	4130	65		Machines fair	Gears
	4140	60		Machines fair	Bolts
	4150	50		Abrasive	Arbors, shafts
Chromium Nickel Molybdenum	4320	50		Work hardens some	Gears, forgings
	4340	50		Work hardens some	Forgings
Nickel Molybdenum	4620	60		Work hardens some	Coil springs, shafts
	4820	45		Work hardens	Gears, rock bits
Chromium	5140	60		Machines fair	Coil and flat springs
	5150	55		Machines fair	Gears, axles
	52100	30		Very tough and abrasive	Bearings
Chromium Vanadium	6150	30		Very tough and abrasive	Springs, pistons
Nickel Chromium Molybdenum	8620	60		Machines well	Carburized gears
	8630	65		Machines well	Gears, tubing
	8650	50		Tough	Bearing races
	8740	60		Machines fair	Bolts, shafts
Silicon & Silicon Chromium	9254	40		Abrasive and hard	Springs
	9255	45		Abrasive and hard	Axles
	9260	45		Abrasive and hard	Bolts, springs

\*Based on AISI 1112 steel as 100%.

*Figure 7.1. Carbon and alloy steel table. Machinability ratings are based on a scale with AISI B1112 as the base steel. If the machinability of AISI B1112 is 300 SFM, then an alloy with a machinability rating of 50% can be machined at 150 SFM under the same conditions.*

## 1100 Series

Sulfur aids machining and is added when fasteners and similar products are made to speed up machining rates; these “resulfurized grades” are known as *free-machining steels*.

In addition to the sulfur content, there is manganese in the composition for added strength. Examples:

1144: 0.40–0.48 C; 1.35–1.65 Mn; 0.040 P; 0.24–0.33 S

The 11 indicates iron. Steels of similar composition but without sulfur are referred to as nonresulfurized grades.

## 1200 Series

The 1200 series of carbon steel is also a free-machining grade, but is rephosphorized as well as resulfurized.

There are only a few grades in this category—1211, 1212, 1213, and 1215. The amounts of phosphorous and of sulfur are about double those of the 1100 series.

Lead also improves machineability, and standard carbon steels can be produced with lead in the range of 0.15–0.35.

These steels are identified by inserting the letter “L” between the second and third digits of the AISI/SAE number. Example: 12L14.

Boron is added in some cases to improve hardenability. A small amount is all that’s necessary: 0.0005–0.003%.

The letter “B” indicates boron content. Example: 10B45.

## 1500 Series

The 1500 series is a standard carbon-steel grade that has additional manganese (over 1.00%).

The other compositions are the same as regular carbon steel. Example:

1541: 0.36–0.44 C; 1.35–1.65 Mn; 0.040 P (max.); 0.050 S (max.)

Iron is represented by the number 15. The additional manganese makes the material more difficult to machine because of its abrasiveness.

Silicon also has a very adverse effect on machining, causing cutting tools to wear very rapidly. Despite the cost and difficulty of machining steels containing silicon, they nevertheless are used in applications where long wear life is required.

## STANDARD ALLOY STEELS

Standard alloy steels are identified by the numbers indicating that silicon (Si), nickel (Ni), chromium (Cr), and molybdenum (Mo) are present. Their amounts are indicated by the first two numbers—the 4000 series, for example—with the last two again indicating the nominal carbon content.



## 1300 Series

This is a standard alloy, which is why it comes after the standard carbon. The amount of manganese is higher than the 1500 series, and silicon is added. There are only four 1300 series available—1330, 1335, 1340, and 1345. An example of the chemical composition of the 1300 series is

1345: 0.43–0.48 C; 1.60–1.90 Mn; 0.035 P (max.); 0.40 S (max.); 0.15–0.35 Si

## 2000 Series

The 2000 series is a nickel alloy, with the first two numbers indicating the approximate amount of nickel. Examples:

2320: Ni, 3.50; C, 0.18–0.23, Mn, 0.3; Si, 0.15–0.35; P, 0.40; S, 0.050.

2500: 2500 series would have 5% Ni.

## 3000 Series

The 3000 series is a nickel–chrome group, with the number following the 3 indicating the nickel–chrome composition. Examples:

3120: Ni, 1.25; Cr, 0.65–0.80 C, 0.018–0.23

3240: Ni, 1.75; Cr, 1.07; C 0.37–0.44

33xx: Ni, 3.50; Cr, 1.50–1.57

34xx: Ni, 3.00; Cr, 0.77

## 4000 Series

The 4000 series alloy steel indicates molybdenum and ranges from 4023 to 4820. From 4023 to 4047, only molybdenum is added to the basic alloy consisting of carbon, manganese, phosphorous, sulfur, and silicon.

From 4118 to 4151, chrome is added to the molybdenum; from 4320 to 4820, nickel is added to the chrome and molybdenum.

All the additions to the basic steel promote a finer structure in the basic material while improving its heat treatment.

## 5000 Series

The 5000 series is a chromium series ranging from 5117 to what is considered high carbon–high chrome, used for bearings—E51100 and E52100. The E stands for the method of manufacture, electric furnace steel. These high-carbon steels have as much as, or more than, 1% carbon, as is the case for E52100, and as much as 1.30–1.60% chromium.

E53100: 0.98–1.10 C; 0.25–0.45 Mn; 1.30–1.60 Cr; 0.15–0.35 Si; 0.025 each, P and S

The advantage of the high-chrome steel is the hardenability and retainability of hardness with fine grain structure.

## 6000 Series

This series has two grades, 6118 and 6150. They are a chrome(Cr)–vanadium (V) grade of steel with a range of 0.50–0.70 and 0.80–1.10 chrome and 0.10–0.15 vanadium (min.).

6150: 0.48–0.53 C; 0.70–0.90 Mn; 0.035 P; 0.40 S; 0.15–0.35 Si; 0.80–1.10 Cr; 0.15 V (min.)

## 7000 Series

This is a tungsten(W)–chromium(Cr) alloy; 72xx indicates tungsten 1.75 and chromium 0.75.

## 8000 Series

The 8000 series is a nickel, chrome, and molybdenum group, with the whole group from 8615 to 8822 having the same proportion of each alloying element in all of the steels. Only the manganese varies slightly. Example:

8620: 0.18–0.23 C; 0.70–0.90 Mn; 0.035 P; 0.040 S; 0.15–0.35 Si; 0.40–0.70 Ni; 0.40–0.60 Cr; 0.15–0.25 Mo

## 9000 Series

This group is a silicon–manganese alloy with as much as 1.80–2.20 silicon. Many of the 9000 series give high hardenability. Others in this range provide medium hardenability—good for holding such cutting tools as diamonds for oil-well drilling, providing good wearability characteristics in ploughing through abrasive earth. Example:

9260: 0.56–0.64 C; 0.75–1.00 Mn; 0.035 P; 0.040 S; 1.80–2.20 Si

## Usage

The preceding steels are used in the manufacture of automotive components, such as gears, springs, suspension systems, crankshafts, camshafts, piston pins, connecting rods, tubing, and sheet metal for body parts.

These various types of standard carbon and standard alloy steels also are widely used for application in other industries that call for their respective combinations of tensile strength, wearability, formability, or other mechanical properties.

The high-strength steels range upward from the 3000 series.

## STAINLESS STEELS

Stainless steels are in a category by themselves and, appropriately, feature their own numbering system.

Used by virtually every industry, these unique alloys strongly resist atmospheric corrosion. The term “stainless steel” originally was a proprietary trade-name that has now become generic and is generally applied to any and all steels that do not rust.

### 200 Series

The most popular grades in the 200 series are 201, 202, and 205. These features about 18% chrome, but relatively low nickel compared to other stainless steels; they have the highest manganese (Mn) content of any of the SS grades and are the only group with nitrogen (N). Nitrogen is added to increase strength and has minimum effect on corrosion resistance.

201: 15–18 Cr; 2.50–5.5 Ni; 0.15 C; 5.5–7.5 Mn; 1.0 Si, 0.060 P; 0.030 S; 0.25 N

202: 17–19 Cr; 4–6 Ni; 0.015 C; 7.5–10 Mn; 1.0 Si, 0.060 P; 0.030 S; 0.25 N

205: 16.5–18 Cr; 1–1.75 Ni; 0.12–0.25 C; 14–15 Mn; 0.50 Si; 0.030 P; 0.030 S; 0.32–0.40 N

This series is not hardenable by heat treating but it does very readily “work-harden,” so care must be taken, when machining, to make sure the cutting edges continuously penetrate into the work and do not slip or rub; otherwise the tool may be damaged and tool life drastically reduced.

The chips from these alloys are stringy and the cutting tool must have a positive rake to shear the work.

201 is used for automotive wheel covers and some domestic flatware. 202 is used for kitchen equipment and hub caps, whereas 205 is less susceptible to work-hardening than the other two and consequently is used for spinning and special drawing applications.

### 300 Series

The 300 series range from 301 to 384 and is a particularly popular group of alloys. Most grades in the series contain 10–15% nickel, but 330 has 34–37% Ni whereas 329 has only 3–6%—but whereas chromium content in the others averages 20% or less, 329 has 25–30% Cr.

Manganese content is low in all 300 grades, and in those in which nitrogen has been added—316, for example—an N is placed after the number: 316N. Nitrogen is added to 316 to increase its strength with minimum effect on ductility and corrosion resistance.

304 is widely used for chemical and food processing equipment, brewing equipment, gutters and downspouts, and tanks.

304: 18 Cr; 8–10.5 Ni; 0.08 C; 2.0 Mn; 1.0 Si; 0.045 P; 0.030 S

302 is another widely used stainless steel, with applications ranging from trim, aircraft cowlings, antennas, springs, building exteriors, tanks, hospital equipment, household appliances, cookware, and food handling equipment to signs, jewelry, and oil refining equipment.

302: 17–19 Cr; 8–10 Ni; 0.15 C; 2.0 Mn; 1.0 Si; 0.045 P; 0.030 S

Although this series cannot be hardened by heat treating, a few of the grades will age-harden, so working parts made of these alloys will become stronger with age.

## 400 Series

The 400 series is a heat-treatable group of stainless steels that can be used in applications in which wearability and the ability to harden with exposure to heat is desirable. Uses include pump shafts, bolts, mining machinery, rifle barrels, springs, tempered rules, cutlery, steam turbine blades, and surgical tools.

Few grades in the 400 series contain nickel, but their greater-than-usual levels of carbon, along with high chromium content, result in good hardenability.

440C: 16–18 Cr; 0.95–1.20 C; 0.1 Mn; 1.0 Si; 0.040 P; 0.030 S; 0.75 Mo

410: 11.5–13.5 Cr; 0.15 C; 0.1 Mn; 1.0 Si; plus small amounts of P and S

There are three versions of 440 stainless: 440A, 440B and 440C. The one used in the example, 440C, yields the highest hardnesses of the hardenable stainless steels and is used for ball bearings, raceways, valves, and valve seats.

## 500 Series

There are only two grades in the 500 series. These exhibit good mechanical properties at moderately elevated temperatures and so they are used in petroleum refining equipment, heat exchangers, and gaskets.

## 18-8 Stainless

This stainless is cited here because it is commonly used in commercial flatware. The 18-8 refers to 18% chromium and 8% nickel. Either 0.30% sulfur (S) or 0.25% selenium (Se) are added to improving machineability, making this stainless as easy to machine as any steel screw stock.

## TOOL STEELS

Tool steels, as the name implies, are primarily used to make cutting tools for manufacturing processes of all kinds, including cutting and forming of woods, plastics, and other industrial materials, as well as for machining and forming metals.

In general, these steel withstand high-impact loading, have high heat resistance, and can perform under adverse conditions. Because they retain their integrity at the relatively high temperatures typically generated when machining

at high cutting and metal removal rates, these tool steels are also referred to simply as high-speed steels.

There are a number of tool steel classifications that indicate general properties—water or oil hardening, shock resisting, hot or cold working, etc.—with selection of the alloy for various applications depending on the needs of the specific application.

## Water Hardening

Water-hardening tool steels are primarily high-carbon steels that may have a small amount of vanadium or chromium added. Although tools made of these alloys take and hold excellent cutting edges, they must be operated at relatively low cutting speeds because their heat resistance runs only to about 350°F. They are designated by a W.

W1: 0.04–1.40 C

W2: 0.60–1.40 C, 0.25 V

W3: 0.10 C; 0.50 Cr

They are used for blanking dies, blacksmith tools, drills and other cutting tools, tools used in the glass and jewelry trades, razor blades, and taps and dies.

## Shock Resisting

This group of tool steels exhibit toughness but at the expense of wear resistance. Hardness and toughness generally come at the expense of one another; in general, the harder a material, the more brittle it is and, consequently, the less shock it can absorb. Hardness implies rigidity and brittleness, whereas toughness implies a degree of flexibility.

The S group is ideal for applications involving great shock, such as cold forming. The group also feature good hot hardness, abrasion resistance, and machinability. These steels are used for bolt header dies, hand and pneumatic chisels, drift pins, forming tools, shear blades, rivet sets, chuck jaws, and similar tools.

These steels have a low tempering range and therefore must be run at low speeds. They are not used for cutting tools. Examples:

S2: 0.50 C; 0.50 Mo; 1.0 Si

S5: 0.55 C; 0.40 Mo; 2.0 Si; 0.80 Mn

S6: 0.45 C; 0.40 Mo; 2.25 Si; 1.40 Mn; 1.50 Cr

## Oil Hardening

Oil-hardening tool steel, as the name indicates, is quenched in oil. These steels have little tendency to shrink or warp, making them very popular for blanking dies, drawing dies, and for cutting tools that do not require high cutting speeds, including taps, reamers, and shear blades. These alloys also are excellent for mandrels, toolholders, and similar devices.

In general, oil-hardening tool steels are not used for applications generating high heat. Because of their low hardening temperatures, these steels exhibit a low tendency to warp or crack, suiting them for applications in which the ability to hold dimensions is important.

The designated letter for this grouping, O, precedes the number. Some have tungsten (W). Examples:

O1: 0.90 C; 0.50 W; 0.50 Cr; 1.00 Mn

O2: 0.90 C; 0.25 W; 0.80 Mn; 1.00 Si

Oil-hardening steels are available as flat ground stock—that is, in bars already ground to standard sizes—making it unnecessary for a shop to perform that preliminary step in the preparation of dies.

## **Cold Work** (Medium Alloy, Air Hardening)

This medium, cold-work material, also available as ground stock, is a high-chromium type with a substantially lower alloy content than other tool steel groups. These steels harden with exposure to air and are very tough and strongly resist deformation. The letter designating this group is A. Examples:

A2: 1.00 C; 1.00 Mo; 5.00 Cr

A3: 1.25 C; 1.00 Mo; 5.00 Cr; 1.00 V

A6: 0.070 C; 1.25 Mo; 1.00 Cr; 2.00 Mn

A7: 2.25 C; 1.00 Mo; 1.00 Mo; 5.25 Cr; 4.75 V

Low cost (less chromium) and resistance to deformation during heat treating plus good machining characteristics make this a frequently used tool steel for punches for blanking and forming and for plastic molding dies. Those qualities make it unsuitable for cutting tools.

## **Cold Work** (High Carbon, High Chromium)

These are popular steels for dies because they possess rather good machining characteristics yet provide great wear resistance. Although expensive, the long life of dies and punches made with these alloys more than compensates for their high initial cost.

Characterized by their very high chromium (12%) and carbon (as much as 2.35%) content, these tool steels are identified by a D. Examples:

D2: 1.50 C; 1.00 Mo; 12 Cr; 1.00 V

D3: 2.25 C; 12 Cr

D5: 1.50 C; 1.00 Mo; 12 Cr; 3.00 Co (Cobalt)

D7: 2.35 C; 1.00 Mo; 12 Cr; 4.00 V

In the past, machining with tools made of the D-series tool steels was usually performed dry, but the use of a cutting fluid with a high water: oil ratio both improves tool life and permits much faster machining rates.

## Hot Work (Chromium, Tungsten, Molybdenum)

This group is designated with an H. The key property of these steels is their ability to withstand elevated temperatures, so tools made of these alloys can sustain high cutting rates without serious impairment. The category consists of three divisions, each with differing dominant properties: (1) chromium, (2) tungsten, and (3) molybdenum.

1. *Chromium.* Features about 5% chromium. Some also have tungsten and molybdenum, and one has cobalt. This group has good abrasion resistance and hot work capabilities; used for extrusion dies, particularly in the aluminum extrusion industry; H13 is the most popular one for this used. Other applications include mandrels, punches, and press liners. Examples:

H12: 0.35 C; 1.50 W; 1.50 Mo; 5.00 Cr; 0.40 V

H13: 0.35 C; 1.50 Mo; 5.00 Cr; 1.00 V

H19: 0.40 C; 4.25 W; 4.25 Cr; 2.00 V; 4.25 Co

2. *Tungsten.* These contain 9–18% tungsten and, with the exception of H23, about half the amount of chromium as the first group. H23 has 12% chromium in addition to 12% tungsten. Examples:

H22: 0.35 C; 11.00 W; 2.00 Cr

H24: 0.45 C; 15.00 W; 3.00 Cr

H26: 0.50 C; 18.00 W; 4.00 Cr; 1.00 V

Although essentially used for the same kinds of tools as the chromium series, these steels have been developed for applications in which the tool is in contact with hot work over extended periods of time, making resistance of the steel to the softening effects of high temperature of prime importance.

3. *Molybdenum.* This group has good resistance to high temperatures. It is low in ductility, however, so it is used primarily for applications where there is a tendency for the dies surfaces in contact with the work to wear at elevated temperatures. These steels excel as hot heading dies for nuts, bolts, and punches used in hot work. Example:

H42: 0.60 C; 6.00 W; 5.00 Mo; 4.00 Cr; 1.00 V

## HIGH-SPEED TOOL STEELS

This series is designated by a T or an M. Primary applications are tools used for machining metals at high cutting speeds. They are exceptionally good at resisting heat and retaining good hardness and strength; their “red heat hardness” is outstanding—they can withstand temperatures of up to 1100°F before the cutting edges soften. They also are highly abrasion resistant. The T identifies tungsten and the M identifies molybdenum as the dominant element in their respective alloys.

## T—Tungsten

The T series is used for all kinds of cutting tools, including bits for lathes and planers, milling cutters, drills, taps, and reamers.

T1: 0.75 C; 18.00 W; 4.00 Cr; 1.00 V

T5: 0.80 C; 18.00 W; 4.00 Cr; 2.00 V; 8.00 Co

T15: 1.50 C; 12.00 W; 4.00 Cr; 5.00 V; 5.00 Co

## M—Molybdenum

The “moly” high-speed tool steel series requires more precise control of heat treating than the T series, and the list of applications is much more extensive, too—in addition to cutting tools for the lathe, milling machine, and other machine tools, the majority of taps, reamers, drills, and end mills are made with M tool steels.

M1: 0.85 C; 1.50 W; 8.50 Mo; 4.00 Cr; 1.00 V

M6 has cobalt for high heat resistance and is used for lathe and milling cutters. Form M30 to M47, cobalt is used extensively. The range from M41 to M47 can be hardened to 67–70Rc (“C” scale, Rockwell hardness test), giving them ability to machine materials above 40 Rc. The most popular of this group is M42, used in drills, reamers, end mills, and lathe tools.

M42: 1.10 C; 1.50 W; 9.50 Mo; 3.75 Cr; 1.15 V; 8.00 Co

## “L” Series Tool Steel

The “L” series is a low-alloy special-purpose tool steel used for automotive gears and forgings, arbors, blanking dies, spindles, wrenches, machine shafts, and other applications requiring a reasonable balance between such properties as wear resistance (needs hardness) and toughness (requires ductility). Hardening depths are intermediate; these tools are less expensive than the highly alloyed types.

## Mold Steels

The mold series steels are identified by a P. Used in plastic molding dies, they exhibit good core strength and wear resistance at high temperatures while providing excellent surface finish. Dies made using mold steels in the P2–P5 range are not machined, but, because of their low carbon content, are bobbed and then carburized to “case harden” (creates a hard skin to various depths) them.

P2: 0.07 C; 0.20 Mo; 2.00 Cr; 0.50 Ni

The most popular mold material is P20, used for zinc and plastics. This alloy is carburized to give the mold a very hard case, and since P20 has a substantial carbon content at 0.35%, the hardened substrate runs relatively deep.

P20: 0.35 C; 0.40 Mo; 1.70 Cr



## NONFERROUS ALLOYS

The nonferrous alloys include aluminum (Al), copper (Cu), magnesium (Mg), zinc (Zn), and lead (Pb). Magnesium, zinc, and lead alloys are used mainly in die casting, whereas aluminum and copper alloys are used in a great multitude of products, including structural and electrical applications.

## ALUMINUM ALLOYS

The aluminum alloy numbering system looks much like the AISI/SAE system for iron and steel. But the similarity ends with the numbers. The aluminum numbering system goes from the 1000 to the 7000 series.

### The 1000, 1100, 1200, and 1300 Series

Aluminum for 1050–1350 is 99% or more pure aluminum and is mainly used for foil or tubing. A few can be acquired in bar stock and other shapes; they are used for coiled tubing, reflector sheets, and for some sheet metal work.

### The 2000 Series

The 2000 series ranges from 2011 through 2024. All have (Mn) copper—from 2.50 to 6.60%—in addition to manganese and magnesium (Mg). Some also have iron (Fe), nickel (Ni), zirconium (Zr), titanium (Ti), and other elements. One of the most popular of this series is one with medium machining characteristics, 2024. Used for things like screw machine products, aircraft parts, and truck wheels, 2024 is readily available from supply houses.

2024: 4.4 Cu; 0.6 Mg; 1.5 Mn; balance is Al

### The 3000 Series

The 3000 series consists mainly of aluminum, with some manganese and magnesium. 3003 is the only one to have copper added instead of manganese.

3003: 1.2 Mn; 0.12 Cu; balance is Al

3003 is used for cooking utensils, builder's hardware, sheet-metal work, and storage tanks.

### The 4000 Series

This group has silicon, which fosters good heat resistance and wear characteristics. Except for 4032—used to make forgings for pistons—alloys in the series are produced in wire form and used for brazing.

## The 5000 Series

The 5000 series contains chromium and is used for welded tanks, automobile trim, and some appliances. Only the two lowest numbered alloys in the 5000 series do not contain chromium. Example:

5083: 0.7 Mg; 4.45 Mg; 0.15 Cr; balance is Al

## The 6000 Series

The 6000 series consists of popular alloys used in all types of manufacturing, from furniture through automobiles to building materials. In addition to chromium, alloys in this series have many elements not found in the lower series, including bismuth (Bi).

One of the most widely used alloys in the 6000 series is 6061. Available from supply houses, it is used by the building industry to make extruded window and door frames and for heavy-duty construction components.

6061: 0.6 Si; 0.28 Cu; 1.0 Mg; 0.2 Cr; balance is Al

## The 7000 Series

The 7000 series has physical properties that have resulted in extensive use for aircraft components, truck bodies, trailer parts, and similar structural items but also make it difficult to machine. Alloying additives include zinc (Zn), zirconium (Zr), and titanium (Ti). Example:

7075: 1.5 Cu; 2.5 Mg; 0.23 Cr; 5.6 Zn; balance is Al

# COPPER ALLOYS

The copper alloys are quite extensive and have a variety of uses. The most popular alloys in industrial use are the brass, bronze, and copper–nickel combinations.

In pure form, copper is a very difficult material to machine because it is gummy and retains heat. Adding lead, tin, and zinc converts copper into brass, small amounts of iron, nickel, and aluminum are also used. The concentrations of these determine the type of brass produced and such properties as its tensile strength.

Brass is much easier to machine than copper, and has a multitude of uses.

Bronze has a lesser amount of copper and lead, and higher concentrations of zinc, manganese, aluminum, and iron. It is utilized where high strength and toughness are required, including gear shifter forks in automotive parts, brackets, castings for aircraft, shafts and gears, and bearings and bushings.

The addition of nickel to copper strengthens the material while improving its formability, suiting it for fasteners, costume jewelry, optical parts, and other applications where both strength and the ability to bend and form the material are important. Varying amounts of nickel, ranging from 10 to 30% are used in

conjunction with zinc, tin, iron, or lead. Machineability to these alloys is not particularly good.

Machineability ratings are based on brass as the standard (100%). Since its rating is 100, if a copper–nickel alloy is rated at 20, then it machines at only one-fifth the efficiency of the base material.

## **DIE CASTING ALLOYS**

Die casting is a procedure in which the alloy is injected in molten form into a die to create the desired configuration, then cooled to a solid state. The alloys commonly used for die casting are aluminum, magnesium, zinc, copper, and combinations of lead–tin–antimony. Applications include such things as carburetors, door mechanisms, fuel pumps, and similar parts for the automotive and other industries.

These alloys are mentioned here because some uses—carburetors, for example—usually require at least drilling for mounting purposes and occasionally require milling and other machining steps. These alloys typically machine easily at very high speeds.

## **SUPERALLOYS**

Superalloys are being used more and more by industry because of their high heat resistance, strength, and excellent corrosion resistance. Applications include turbine wheels, aircraft engine and structural parts, gas turbine blades, and jet engine parts.

### **Nickel-Based Alloys**

These include alloys with such proprietary names as Hastelloy, Inconel, Waspoly, and Udimet. They contain carbon, manganese, silicon, chromium, cobalt, molybdenum, tungsten, titanium, aluminum, with the balance—extremely high—nickel. All are very difficult to machine.

### **Iron–Nickel-Base Alloys**

Although these alloys contain many of the same elements as the nickel-based alloys, they are very high in iron content. Such compositions as A-286, Incoloy, Pyromet, Unitemp, and Duraloy are very dense and difficult to machine, requiring high horsepower for efficient metal removal. Nickel content is less than that of the nickel-based alloys whereas the iron present will run 50% or more.

### **Cobalt-Base Alloys**

Iron and nickel content is lower in this alloy, with cobalt—whose content is as high as 75%—being the dominant element. To say it is difficult to machine is to understate the problem.

## PLASTICS

Plastics are widely used by industry, including many reinforced with such materials as glass, metal wire, and graphite. These give plastics the strength and many of the properties of steels. Glass-reinforced plastics and composite materials typically must be machined with diamond-impregnated tools; otherwise, the cutting tool wears very rapidly. Many of these must be cut by a water jet, since conventional cutting tools tend to snag the reinforcing strands and ruin the workpiece.

Most plastics are molded to the size and shape desired and do not require machining.

## HEAT TREATMENT OF METALS

To cut bread, a knife must have a sharp cutting edge and be harder than the bread. In fact, to cut something—anything—the tool must (1) have sharp cutting edges and (2) be harder than the material to be cut. This applies to machining: To be able to cut steel and other alloys—or wood, plastic, and other workpiece—at all, let alone machine them “efficiently,” the tool used must have a sharp cutting edges and be harder than the material to be cut.

The method of machining, the size and design of the machine, and its power and method of applying that power determine such things as the shape cutting tools can have.

The workpiece material and its density and physical properties determine its machineability—how easy or difficult it will be to machine—and the configuration of the cutting tool.

But the hardness of the alloy determines how well the tool will cut the workpiece—and whether it can be cut at all. Heat treatment is a process used to give steel alloys the hardness required. Heat treating also serves such other purposes as relieving internal stresses and strains—as the names given to the various forms of heat treatment of metals imply:

- \* Hardening
- \* Quenching
- \* Tempering (drawing)
- \* Annealing
- \* Normalizing
- \* Casehardening

### Hardening

Hardening is the process of raising the temperature of steel to its critical point: That is where the grain pattern formed by groups of molecules in the steel are reduced to their smallest condition. At this point, the alloy is densest and hardest, and the material is “frozen” to keep it in that condition. To “freeze” the heated material, it is immersed in water, oil, or air. The alloy, however, tends to return to its original condition.

To be able to stay in the pattern achieved at that critical point, in addition to being cooled rapidly before it can revert back to its original condition, the steel must contain a sufficient amount of carbon.

Each steel or alloy has its own critical temperature and such things as the size and shape of the workpiece and how slowly or quickly it transmits and radiates heat affect the ability of the material to retain the results of the transformation.

The size, shape, and use of the part to be hardened determines the type of alloy to be used. But the amount of carbon in the steel or alloy is the single most important factor determining the ability of the alloy to achieve and retain high hardness. A steel with a carbon content of more 0.50%, for example, can be hardened to Rockwell 65 on the C scale, whereas combinations of iron and other elements with less than about 0.35% carbon cannot be hardened unless casehardening, described below, is performed.

Size and shape are important in heat treating; if a part is thin, heat treatment will penetrate completely through. If, however, the part is thick, the center portion will cool slowly and will be softer than the surface. While that might be acceptable or even desirable for some applications, it would be wholly unsuitable for others. For the latter, the addition of molybdenum, chromium, or tungsten changes the characteristics of the steel and its hardening process, allowing the heat treatment process to penetrate completely. That, in turn, permits the steel to achieve its full hardness all the way through.

To improve such properties as red heat hardness—the ability to withstand high temperatures without softening—abrasive resistance, and other desirable cutting tool attributes, the steel mill adds various alloying elements, as presented earlier in this chapter. These change heat treatment procedures as well as the properties of the alloys in which they are used. Carbon, for example, greatly affects steel alloy properties. The harder the part, the more brittle it is—a difference of one-tenth of one percent, for example, can be noticed. During heat treating, decarburization (removal of carbon) of the material takes place at and near the surface of the steel as it is subjected to high temperatures. The simplest way to overcome this problem is to make the dimensions of the material before heat treating a little larger than the final dimensions so it can be machined after the hardening process. Parts to be heat treated should be about 0.015–0.020 in. oversize.

When heat treating steel from 10xx to 93xx, the appropriate temperature range is from 1400–1450 to 1650–1700°F. The lower-range steels are usually quenched in water or oil whereas the upper range is usually quenched in oil. The higher temperatures tend to produce deeper hardness penetration whereas the lower temperatures tend to result in shallow hardness penetration but increased resistance to stress that cause splitting or bursting. A rule of thumb is:

Carbon Content (%)	Temperature (°F)
0.65–0.80	1550–1550
0.80–0.95	1410–1460
0.95–1.10	1390–1430
1.10–over	1380–1420

When heat treating stainless chromium irons and steels, the temperature range is slightly higher, 1750–1850°F, and oil or air hardening is applied. They can be air cooled or cooled in the furnace for tempering.

When heat treating high-speed steels, the hardening temperatures range from 2200 to 2500°F and are usually quenched in oil.

If materials are thin and heated at high temperatures, the proper length of time for heat treating is normally only on the order of minutes. But if the material is thick, the time the material needs runs into hours. As a general rule, a workpiece should soak in the oven approximately 1 hr for each inch of thickness.

## Quenching

Effective heat treating methods require proper rates of cooling as well as correct heating rates and temperatures. Proper cooling rates depend on the size of the part, the alloy and its element percentages, and the amount of carbon. This requires quenching the material to a completely hardened state and then tempering it to the desired hardness and toughness.

The method of quenching—water, oil, or air—depends on the alloy contents and size of the part and the amount of agitation required.

The heat removal rates and characteristics of each medium determine which type of quench should be used. In water and oil quenching, the bath should be large enough so the medium can dissipate the heat rapidly.

When quenching in water, the water should be kept at a minimum temperature of 70°F, with 100°F being preferable. The water should be soft; salt is often added to water to eliminate the danger of cracking the part being cooled. About 8% rock salt is used.

Oil is the most common quenching medium since it results in even cooling that imparts good hardness and toughness to the material. All types of oil are used, but mineral oil is the most common because of its low cost and the fact that does not have objectionable odor. The temperature of the oil should ideally be about 125°F.

When quenching in a water or oil bath, the part should be agitated (moved around) to evenly dissipate the heat.

Air cooling is used where the degree of distortion resulting from oil quenching is objectionable or where the alloy content of the part will permit full hardening by air quenching.

## Tempering (Drawing)

Tempering is the process of reducing the brittleness and internal strains that develop in many alloys during hardening and quenching of the part. It is also used to bring the part to a desired hardness and toughness.

After quenching, the part is reheated in the furnace at anywhere from 300 to 750°F and then cooled slowly either by air in the furnace or in an oil bath. This removes any strains introduced in the heat treating process.

The part should soak in the oven at the recommended temperature for approximately 1 hr for each inch of thickness. The temperature used is determined by the

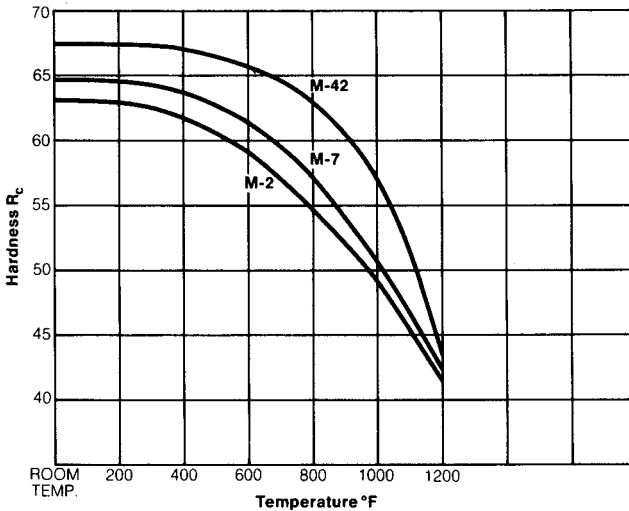


Figure 7.2. Graph for hardness and tempering.

hardness desired. That temperature is usually recommended by the steel manufacturer (Figure 7.2).

The following chart applies when tempering by color:

Temperature (°F)	Color of Steel
440	Light yellow
480	Dark yellow
500	Brown-yellow
540	Full purple
560	Full blue
570	Dark blue
640	Light blue

The tempering process is commonly referred to as drawing the material after heat treatment.

## Annealing

Annealing is a softening process in which the alloy is heated above the critical temperature and then cooled slowly to a temperature below the transformation range. It returns the material very nearly to its original state and is normally done to permit additional work to be done on it.

In general, the slower the cooling rate, the greater the resulting softness and ductility.

Steels of high carbon content should be cooled more slowly than the steels of lower carbon content; also, the higher the alloy, content, the slower the cooling

rate required. In air-cooled materials, the recommended practice is to encase the heat-treated material in asbestos and bury it in sand to fully anneal it.

Annealing temperatures recommended by the American Society of Testing Materials are:

Carbon Content (%)	Temperature (°F)
Less than 0.12	1600–1700
0.12–0.29	1550–1600
0.30–0.49	1500–1550
0.50–1.00	1450–1500

Slightly lower temperatures are recommended for steels having more than 0.075% manganese content.

## Normalizing

Normalizing is usually done to refine grain structures and place the material in its best condition for machining. It is differentiated from stress relieving or annealing in that it is carried out at approximately 100–200°F above the critical point representing the regular hardening temperature. The full air-hardening steels are exceptions.

Normalizing procedure produces good strength and ductility. Nonalloy steels such as 1040 are not good candidates for normalizing since they would only tend to harden and defeat the purpose of the normalizing process.

## Casehardening

To harden low-carbon steel, it is necessary to increase the carbon content on the surface of the steel; when that is accomplished, a thin outer layer or “case” can be produced by utilizing normal hardening procedure.

Casehardening thus requires two operations:

1. *Carburizing*. The process of impregnating the outer surface with carbon.
2. *Hardening*. Once sufficient carbon has been added to the surface, hardening can take place. The process of hardening also gives the core proper physical properties.

There are several methods of carburizing:

- \* Using carbonaceous mixtures
- \* Liquid carburizing
- \* Gas carburizing
- \* Cyanide hardening
- \* Nitriding



**Carbonaceous Mixtures.** These are applied at 1650–1700°F. The steel parts to be carburized are placed in metal boxes or pots and fully immersed in the compound.

There are many commercial carburizers on the market, including hard and soft wood charcoal, animal charcoal, coke, coal, beans and nuts, bone and leather, and various combinations of these. Energizers to activate these compounds under heat include ammonium compounds, various salts, soda ash, lime, and hydrocarbons.

The length of time heat is applied is determined by the depth of carbon penetration required. Carbon from the compound is absorbed by the surface of the steel and, depending on the alloy and carburizing time, can penetrate sufficiently deep to create a  $\frac{1}{8}$ -in. (0.125-in.) case in heat treatment. It is then heat treated to harden the carburized area and simultaneously toughen the core, resulting in a steel with a hard surface and a somewhat softer, more ductile, and flexible substrate.

The combination of hard surface and flexible substrate is useful in many applications. In die work, for example, if there is an accident a casehardened punch will bend rather than break, minimizing the chance of damage to the die as well as making it possible to salvage the punch.

**Liquid Carburizing.** This method uses a salt bath mixture of sodium cyanide (8–12%) and barium chloride. The temperatures used are 1600–1700°F, attained by means of electrodes immersed in the bath. A stirring action is used to ensure uniform temperature.

This is a faster method of carburizing and produces more uniform carbon penetration. There is less distortion of the part, and it takes much less time to immerse the workpiece than to pack carbonaceous materials around it. The depth of the case following hardening is from 0.035 to 0.062 in., or anywhere from three to six times as deep as that produced using carbonaceous mixture.

**Gas Carburizing.** Natural gas (methane), some propanes, and butanes are used. The gas is mixed with air or other gases. This method uses a slow revolving type of gas carburizing furnace for small parts and uses a vertical pit-type furnace for large parts in which the gas is circulated to minimize the possibility of damage while still ensuring proper carburization.

This type of carburizing is used for production work on bearings, gears, or large shafts.

**Cyanide Hardening.** This method is used where the low-carbon steel requires a very hard outer surface but internal hardening is unnecessary because high shock resistance qualities are not needed.

The process creates a very thin outer skin (only a few thousandths) by immersing the steel in a bath consisting of sodium cyanide.

The temperatures employed are from 1450 to 16500°F.

**Nitriding.** This is surface hardening effected by heating the steel in an atmosphere of nitrogen (ammonia gas) at about 950°F. The steel is cooled slowly.

This method is frequently used to harden the edges of cutting tools; the depth of the hardness is about 0.0005 in., increasing the life of the tool substantially without affecting its ability to absorb shock.

## TESTING HARDNESS OF MATERIALS

After heat treatment for hardening, the methods used to check hardness in the machine shop fall into several categories. The two main hardness tests used are Brinell and Rockwell.

### BRINELL HARDNESS TEST

Hardness of the steel is one of the factors that determines the feeds, speeds, and cutting tools used. The Brinell hardness test is the one manufacturers use to specify their steel or alloy.

The Brinell hardness number (BHN) scale ranges from 152 BHN on the low side to 630 BHN on the high side. Up to about 250 BHN, materials are relatively easy to machine. Above that, machining becomes more difficult.

The Brinell test for determining hardness on metal materials consists of applying a known load to the surface of the material through a hardened steel ball of a given diameter. The impression that occurs on the metal is measured and the BHN is then calculated. The formula for calculation is

$$\text{BHN} = \frac{\text{Load on indenting tool (kg)}}{\text{Surface area of imprint (mm}^2\text{)}} = \frac{P}{\frac{1}{2}D(D - \sqrt{D^2 - d^2})}$$

where  $D$  is diameter of ball in millimeters,  $d$  is measured diameter at rim of impression in millimeters, and  $P$  is applied load in kilograms.

The range mentioned above is:

- \* With a steel ball, the top hardness is 450 BHN.
- \* With a Hultgren ball, the top hardness is 500 BHN.
- \* With a tungsten carbide ball, the top hardness is 630 BHN.

The ball is 10 mm in diameter and the loads used are 3000 kg, 1500 kg, or 500 kg.

### ROCKWELL HARDNESS TEST

This is the most common test used in toolrooms and machine shops. It can measure hardness in the high range easily and quickly by reading off a scale or digital readout with no computing necessary.

The tester is essentially a machine that measures hardness by determining the penetration of a diamond, or in other cases a ball, into the material under fixed

conditions. The hardness number is related to the depth of indentation as follows: the higher the number, the harder the material.

The procedure is as follows.

1. Set the scale to zero with the diamond on the material. This puts a minor load of 10 kg on the material. The minor load is always 10 kg for all Rockwell tests.
2. Next, apply the load (150 kg to the material) by pushing the outer lever down to release the weight on the material. The 150 kg is for high hardness materials.
3. The third step is to release the side handle toward the front. This allows the arrow to the scale to revolve until it reaches its peak.
4. Now read the hardness off the point of the arrow.

This is the Rockwell C scale. This hardness can be read up to 70 on the C scale. This indicates maximum hardness.

The Rockwell B scale uses a ball with a  $\frac{1}{16}$ -in. diameter and is for softer material. The load is 100 kg, and reading it is accomplished the same way.

The Rockwell A scale is for tungsten carbide and other extremely hard materials. It is also used for thin hard sheets. The load is 60 kg and a diamond penetrator is used.

The Rockwell D scale is used when a somewhat lighter load is desired on the C scale. The load is 100 kg using a diamond penetrator.

The Rockwell E scale is for very soft materials and uses a  $\frac{1}{8}$  ball with a 100-kg load.

The Rockwell F scale is the same as the E scale except it uses a  $\frac{1}{16}$ -in.-diameter ball.

The Rockwell G scale is for metals harder than tested on the B scale using a 150-kg load with a  $\frac{1}{16}$ -in.-diameter ball.

Rockwell H and K scales are used for softer metals and use a  $\frac{1}{8}$ -in.-diameter ball with a 60-kg load on the H scale and a 150-kg load on the K scale.

When a shallow impression or a small area for hardened steel and hard alloys is needed, then 15-N 30-N, and 45-N scales are used, with each number representing the load applied and a Brale (diamond) penetrator being used.

The 15-T, 30-T, and 45-T scales are used for materials softer than hardened steel for shallow impressions and penetrated with a  $\frac{1}{16}$ -in. ball.

Other hardness tests are as follows.

## SCLEROSCOPE

This is based on the height of a rebound of a diamond-tipped hammer falling from a fixed height. The hammer is approximately  $\frac{1}{4}$  in. in diameter and  $\frac{3}{4}$  in. long with a striking tip rounded to 0.10-in. radius mounted in its downward end. It falls in a glass tube that has a scale graduated in 140 divisions.

This is a portable hardness tester, and the hardness is based on an average number of readings.

## VICKERS HARDNESS TEST

This test is based on applying a known load for a specified time to the surface of the material. The indentation is then measured. This test is very similar to the Brinell test except for the indenter used.

## KNOOP HARDNESS NUMBERS

This test is applicable to very thin metal-plated surfaces, exceptionally hard and brittle materials, and shallow carburized or nitrided surfaces.

The test consists of applying a known load for a specified time to the surface of the material through a diamond having unequal longitudinal and traverse included angles. The longitudinal angle is 172 degrees and the traverse angle is 130 degrees. The applied load must be below 3600 g. This is called a Tukon tester when used with the Knoop indenter; it is fully automatic.

There are other tests, but they are not used as frequently.

## HARDNESS CONVERSION

Hardness conversion scales are available in the *Machinery's Handbook* and can be used when wanting to convert Brinell to Rockwell or other hardness test or vice versa.

## SPEEDS AND FEEDS

Speeds and feeds to machine materials on the various machine tools are determined by the materials being machined, the cutting tools being used to machine the steels, the type and horsepower of the machine being used, the condition of the machines, and the operator of those machines.

There are literally thousands of alloys on the market today made by hundreds of manufacturers. All the alloys to be machined are determined by their chemical composition, the density, and hardness. Extreme softness or gumminess makes it difficult to machine the material.

Many of the materials can be grouped together for similar cutting speeds since the characteristics of machining are closely related. However, there are some guidelines one can follow when determining cutting speeds.

### Cutting Speeds

Tool life is influenced most by cutting speed, then by the feed rate, and least by the depth of cut. After a depth of about 10 times greater than the feed rate, a further increase in the depth of cut will have no significant effect on tool life.

The first step is to start with the cutting speed, which is always specified in surface feet per minute.

When converting from surface feet per minute to revolutions per minute the formula is

$$\text{RPM} = \frac{\text{SFM} \times 12}{\pi \times D}$$

where SFM means surface feet per minute, RPM means revolutions per minute,  $\pi$  equals 3.1416, and  $D$  is the diameter of the work or of the tool, whichever is rotating.

Always start with the given speed recommended on the charts. This speed is only a recommended guideline. From there on, speeds and feeds may have to be adjusted for the material.

Adjustment of speeds and feeds are necessary when under vibration occurs, color of the chip indicates too much heat being generated, or the chip formation itself is unsatisfactory.

Always use cutting fluids when possible. Cutting fluids lubricate, reduce heat, and help in removing metal. They all use chemical enhancing materials. Some materials will machine very poorly without cutting fluids.

Select the proper cutting tool for the material being machined. Some materials require a positive rake angle, others require a negative rake angle, others require no rake angle at all.

High-speed tools are great for many materials whereas carbide cutting tools may be the only economical way to go for others. Cast alloy cutting tools have their place as well as ceramics, cubic boron nitride (Borazon is one trade name indicating this type of cutting material), and diamond cutting tools.

The amount of nickel and chrome can indicate a work-hardening type of metal. This means using slower cutting speeds and penetrating the work at all times. If a dwell is allowed of the tool doing the cutting then the part will work harden and ruin the cutting tool.

Cutting fluid should be used. A straight sulfurized oil is recommended for some materials. The sulfur aids in removing metal.

Density is another troublesome factor when machining. Materials such as A286 are very dense, thereby creating problems. When machining this type of material, horsepower in sufficient quantity is necessary.

The feed should be constant and relatively deep to ensure penetration for both work-hardening and dense materials.

The more carbon in a material, the more difficult it is to machine. Speeds must be reduced to minimize heat, and a good quality soluble oil is recommended.

When heat becomes a major factor, chemical cutting fluids mixed with water can be very effective.

Tool steels can present various problems because of the amount of carbon mixed with chrome, tungsten, or molybdenum.

Cutting D2 (a tool steel) in the past has been to machine it dry to avoid work-hardening problems. Because of the heat generated it has been found to

machine faster and easier by using a high mixture of water with a chemical cutting fluid, as much as a 15:1 ratio, giving good cooling action without any change in the speeds and feed recommended. As a result, cutting rates and life of the cutting tool were increased almost double without changing speeds or feed.

Although this may not be applicable to all tool steels, it indicates that experiments with cutting fluids can prove quite beneficial.

## Depth of Cut

This, while having the least effect on tool life, may sometimes be the first step in selecting the cutting conditions for the material to be machined. The reason is that it determines the amount of metal to be removed.

It is also limited by the available power, rigidity of the workpiece, tool, and setup. Depending on the above limitations, always take the heaviest depth of cut that is possible with the power available because it has the least effect on life of the cutting tool.

A rule of thumb is 10 times the feed rate.

## Feed Rate

When selecting feed rate, the power, rigidity, and surface finish are the prime considerations. This all must be in consideration of the depth of cut as outlined above.

When surface finish is of prime importance, then select the maximum feed rate that will produce the acceptable finish.

When roughing a part first, use the maximum feed rate compatible with the machine conditions.

Cutting tools can be varied in configuration for roughing and finishing cuts, allowing for heavy feeds and depth of cut for material being machined.

The changing of roughing and finishing tools can be economical when removing large amounts of material and used on machines with quick-change tooling.

High-speed tools are used frequently; however, when carbide tools are used, the cutting speeds can be increased up to four times with the resulting feed rate increasing that amount.

Use the cutting speed charts that are available in *Machinery's Handbook* for general guidelines.

## CUTTING FLUIDS

Cutting fluids can be critical for most machining operations. They are used to cool both the cutting tool and the part being machined. They lubricate the cutting tool to prevent chip welding of the cutting edge, reduce or eliminate heat build-up in the tool and workpiece, and the cutting fluid cleans the area of chips. These features contribute to more accurate and markedly faster machining—as much as 30% faster than cutting dry.

Synthetic fluids, available both in very heavy-duty applications and very light duty work, now dominate the cutting industry. Oils are still used, many of which are soluble in water. These are called emulsified oils and they mix very well. They are used where medium and low speeds are required such as when machining stainless steels, tool steels, and tough die steels, including the A steels, H steels, and T steels. Straight oil is sometimes used but many try to avoid it because it is messy and it requires cleaning the steel afterwards when heat treatment is mandated.

A soluble oil formulated with sufficient emulsifiers and rust inhibitors and mixed with the proper ratio of water is excellent for grinding applications. It helps prevent premature wheel breakdown and metallic wheel loading and it also can provide better surface finishes. We have listed a cutting fluid selector. To help better evaluate cutting fluid applications, we've included a brief explanation of machining rating applications with an example of the rating system, and recommend that the two be used go together.

Machine rating involves nothing more than determining how difficult it is to cut the metal. The easier to machine the metal is, the higher the percentage it is given. Free machining metals, such as many of those in the 1100 range, are rated at 100%. Carbon steels at the lower end of the amount of carbon content mixed with iron are rated somewhere in the 70% category.

If, however, the amount of carbon is very low, the metal can be more difficult to machine. Because of a gummyness that occurs in very low carbon steels, they may fall into the 40% or 50% range. Some examples of machine rating are listed below.

## Cutting Fluid Types

Straight cutting oils are refined and otherwise processed to provide heat absorption and lubrication properties. They are usually of three types: mineral oils, fatty oils, and combinations of fatty oils and mineral.

They are used principally on metals, and have a higher lubricity than any other cutting fluid types. Their heat transfer characteristics, however, are the lowest among cutting fluid types; consequently, cutting speeds must be held in the low-to-intermediate range. Very little straight cutting oil is used today. To improve straight cutting oils, additives such as sulfur are used to improve resistance to chip-welding. To lower cost, mineral seal or paraffin may be used to dilute the oil.

Soluble oils are mineral oils and fat mixtures with water. An emulsifier is added to the oil, which allows the oil to be held in microscopic droplets throughout the water. Soluble oils have less lubricity than straight oils, but provide much better heat transfer capability. They generally perform best at intermediate and higher cutting rates.

Synthetic fluids are not formulated from petroleum oil bases and therefore are not true oils. They are designed for a full range of light-to-heavy duty-cutting jobs, and they may have synthetic additives that duplicate the effect of additives

used in soluble oils. Heat transfer features are superior to those of soluble oils, making it possible to achieve high cutting speeds with carbide tools, especially when used in a mist form.

## Rules to follow

- \* Always flush out the coolant tank before putting fresh fluid in.
- \* Always keep the reservoir full to ensure full volume flow.
- \* Keep the system free of chips by removing them from the reservoir daily.
- \* Thoroughly mix the properly measured amounts of fluid concentrate and water in a separate container.
- \* Before putting the mix in the reservoir, check the mix ratio with an Industrial Fluids Tester.
- \* When mixing, do not simply dump the concentrates and water together—prepare them in line with the supplier's ratio and mixing recommendations.

## Selecting the Proper Cutting Fluid

The following chart presents suitable cutting fluids for a variety of machining operations. Where different materials are to be machined in relatively small quantities, the recommended procedure is to base the selection on the material most commonly machined.



**GENERAL MACHINING SELECTION CHART**  
**Machinability Ratings of Materials**

**Cutting Fluids Selection**

	<b>1</b> <b>Greater Than 70%</b>	<b>2</b> <b>50%–60%</b>	<b>3</b> <b>40%–50%</b>	<b>4</b> <b>Less than 40%</b>	<b>5</b> <b>Greater than 100%</b>		<b>6</b> <b>Less than 100%</b>
Machine Operations	Cast Steels Free Cutting Carbon Steels	Alloy Steels Nickel Steels Nickel Chrome	Stainless Steels (Free Cutting)	Stainless (austenitic) Hi speed Tool Steels	Magnesium Alloys	Aluminum Alloys Brass	Monel Inconel Nickel
Power Sawing	Heavy duty Synthetic Cutting Fluid	Heavy Duty Synthetic Cutting Fluid	Heavy duty Emulsified Oil	Heavy duty Emulsified Oil	Light duty Straight Oil	Heavy Duty Synthetic	Heavy Duty Emulsified Oil
Grinding (Surface, Cylindrical)	Light to Medium Duty Synthetic or Light Oil	Light to Medium Light Straight Oil	Light to Medium Light Straight Oil	Light Straight Oil Heavy Duty Emulsified	Medium Duty Oil Emulsified	Light to Medium Duty Synthetic or Oil	Light Oil Mixture Heavy duty Oil Mixture
Tapping	Heavy duty Synthetic Cutting Fluid	Heavy Duty Emulsified Oil	Extra Heavy Duty Emulsified Oil	Extra Heavy Duty Emulsified Oil	Straight Oil	Heavy duty Synthetic Cutting Fluid	Cutting Oil HD Synthetic Tapping Compound
Threading	Heavy Duty Synthetic	Heavy Duty Synthetic	Extra Heavy Duty Emulsified Oil	Extra Heavy Duty Emulsified Oil	Straight Oil	Straight Light Oil	Extra Heavy Duty Emulsified Oil
Deep Drilling Milling Boring	Heavy Duty Synthetic Oil Light Oil	Heavy Duty Synthetic Oil Light Oil	Extra Heavy Duty Synthetic Oil Straight Oil	Emulsified Oil Straight Light Oil	Straight Light Oil	Heavy Duty Synthetic Oil	Straight Oil
Drilling Turning	Heavy Duty Synthetic Oil Or Straight Oil	Heavy Duty Synthetic Oil Or Straight Oil	Straight Oil	Straight Oil Or Dry	Straight Oil Or Dry	Heavy Duty Synthetic Oil	Straight Oil Or Dry

## *Chapter 8*

# NUMERICAL CONTROL AND CNC

## INTRODUCTION

The purpose of a basic machine-shop book is to lead into the higher technical aspect of machining. It is futile to attempt to machine anything in any method without understanding the steps required to accomplish that machining need. Numerical control is the perfect example.

When programming a part to be machined with the aid of electronics, the steps required are very precise to accomplish what is needed.

While step-by-step procedure can be varied, the speeds, feeds, tools, materials, cutting fluids, and capabilities of the machine must be understood.

To help bridge the gap to numerical control, we will show three methods of programming: One is the manual method used by many shops today. The second is simplified programming with help from the controller. The third is computer assist. The same part will be used to show the three methods and how the time for programming a part is progressively reduced. All three methods use the same program, but the three methods shown will exhibit what is simplifying the program.

New methods are rapidly being developed and should be investigated before making a final decision about what may be best for you.

Project No. 2 in this book, located after the milling machine chapter (Chapter 4), is for constructing a wheel puller.

The step-by-step procedure for machining that part is given along with the drawings of each component. If two or more wheel pullers are required, then numerical control should be considered. Costs will be reduced for set-up and machining time along with tool costs.

We will program the wheel puller manually, showing the procedure to be used; then we will show the program that is generated.

## AXES

In programming, it is necessary to label the direction that the tool travels to give commands to the machine.

In machines such as the milling machine that move the table instead of the tool, it must be understood that the direction the tool is cutting is what is important, not the table movement. Therefore, on a milling machine, as the tool cuts from right to left or left to right this is called the  $X$  axis and is the longitudinal movement (Figure 8.1). As the table moves in or out toward the column and away from it (crosswise), this is the  $Y$  axis. When the tool moves vertically with the spindle, it is called the  $Z$  axis.

When turning on a lathe, the  $Z$  axis is movement of the carriage toward or away from the spindle and the  $X$  axis is the movement of the carriage toward or away from the operator (crosswise). There is no  $Y$  axis on a lathe unless an attachment may be put on it.

Note that the movement of the tool represents a quadrant (Figure 8.2). The programmer controls the movement of the tool using the coordinate system.

Once understood, programming arcs or straight lines is easy. In an arc, for example, the points where the arc starts are the beginning, as plotted out on the coordinates, then the point where the arc ends are plotted. The radius is given in distance. Now the arc can be plotted, or machined very easily.

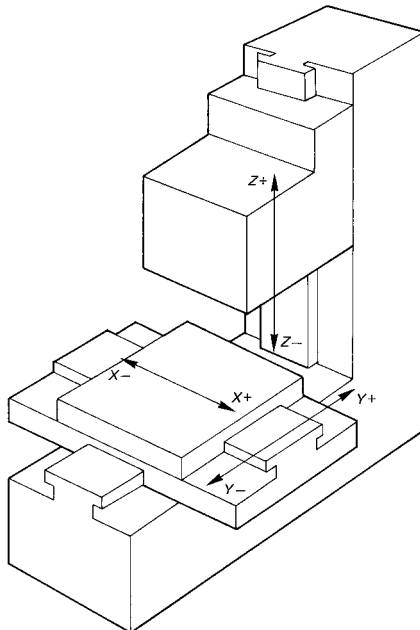


Figure 8.1. The axes on a vertical milling machine.

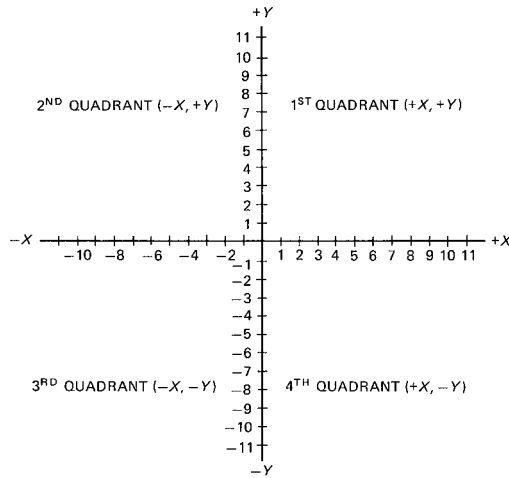


Figure 8.2. Four quadrants in the Cartesian coordinate system.

To determine what to call the movement of the axis when going from one direction or the other on a milling machine, the following is used:

1. The X axis is positive (+) when the tool is cutting to the right and negative (-) when the tool is cutting to the left (Figure 8.3).
2. The Y axis is positive (+) when the tool is cutting away from the column and negative (-) when the tool is cutting toward the column.
3. The Z axis is positive (+) when the tool is moving up or away from the workpiece and negative (-) when it is plunging in or moving into the workpiece.

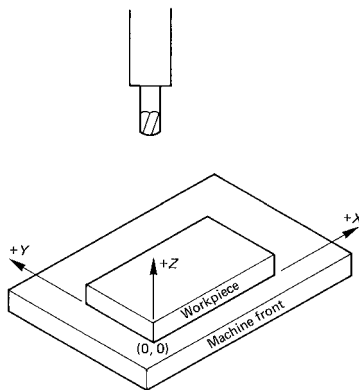


Figure 8.3. Vertical Z axis in relation to X and Y axes, the workpiece, and the table.

On the lathe, the following directions are used:

1. The X axis is positive (+) when the tool is cutting toward the operator or the carriage is moving away from the operator. It is negative (-) when

the tool is cutting away from the operator or the carriage is moving toward the operator.

2. The Z axis is positive (+) when the cutting tool is moving to the operator's right or away from the spindle. The Z axis is negative (-) when the cutting tool is cutting toward the spindle or moving to the operator's left.

## PROGRAMMING

Once the axes are understood along with mathematics and blueprint reading, programming a part can readily be accomplished. There are some very basic machine codes that are used and reference to them is necessary. First are the miscellaneous functions used to start the machine, select the tool, and start the spindle turning. In addition, if cutting fluid is used it starts and stops the flow of coolant. These machine functions are called "M" functions.

## MISCELLANEOUS FUNCTIONS

The miscellaneous functions are as follows (asterisks mean "most commonly used").

- \* M00 (Program stop). This code stops the program and will also stop the spindle and coolant if activated. The operator must cycle start in order to continue with the remainder of the program.
- \* M01 (optional stop). This code functions as above but only if the control unit optional stop selector is in the "on" position.
- \* M02 (end of program). After all movement of the program is completed, this code will stop everything and may include rewinding the tape.
- \* M03. \* Start spindle in clockwise direction.
- \* M04. \* Start spindle in counterclockwise rotation.
- \* M05. \* Spindle stop.
- \* M06 (tool change). \* This code stops the spindle and coolant and retracts the spindle to allow a tool change.
- \* M07. \* Turn flood coolant on.
- \* M08. \* Turn mist coolant on.
- \* M09. \* Coolant off.
- \* M10. Automatic clamping of slides, fixtures, workpiece, and spindle.
- \* M11. Unclamping of above.
- \* M12. Synchronize multiple sets of axis such as on a lathe having four axis and two independent operated heads or slides.
- \* M13. Combines simultaneous clockwise spindle and coolant on.
- \* M14. Same as M13 except it is for counterclockwise rotation of spindle.
- \* M15. Rapid traverse in positive (+) motion.
- \* M16. Same as M15 but negative (-) motion.

- \* M17 and M18. Unassigned.
- \* M19. Spindle stop at a predetermined angle.
- \* M20 to M29. Unassigned.
- \* M30. Rewind tape and automatically transfers to second tape if in the system.
- \* M31. Interlock bypass for temporarily circumventing normally provided interlock.
- \* M32 to M39. Unassigned.
- \* M40 to M46. Signal gear changes are required at the machine; otherwise unassigned.
- \* M47. Continues program execution from the start of the program unless inhibited by an interlock signal.
- \* M48. Cancels M49
- \* M49. Deactivates a manual spindle or feed override and returns to the original program value.
- \* M50 to M57 Unassigned.
- \* M58. Cancels M59
- \* M59. Holds RPM constant at program value.
- \* M60 to M99. Unassigned.

## PREPARATORY FUNCTIONS

Next are the “G” codes, known as the preparatory functions. These codes determine such modes of operation of the system as drill cycles, dwells, rapid traverse, cutter compensation, and many others. The following G codes are as follows (asterisks mean “most commonly used”).

- \* G00. \* Rapid traverse with point-to-point positioning.
- \* G01. \* For describing linear interpolation blocks and reserved for contouring.
- \* G02. \* Clockwise circular interpolation.
- \* G03. \* Counter clockwise circular interpolation.
- \* G04. \* Time delay (dwell).
- \* G05 to G07. Unassigned. Can be used by machine tool builder or systems builder.
- \* G08. Acceleration code. Causes the machine to accelerate at a smooth rate.
- \* G09. Deceleration rate.
- \* G10 to G12. Not used with CNC. Used with hardwire systems.
- \* G13 to G16. Used to direct the control system to operate on a particular set of axes.
- \* G17 to G19. To select a coordinate plane for such functions as circular interpolation or cutter compensation.
- \* G20 to G32. Unassigned. Can be used by the machine manufacturer.

- \* G33 to G35. \* Used for CNC lathes for thread cutting. G33 is for constant lead on threads, G34 is for increasing lead, and G35 is for decreasing lead on threads.
- \* G36 to G39. Unassigned.
- \* G40. \* Terminate cutter compensation.
- \* G41. \* Cutter compensation, when the cutter is on the left side of the work surface looking in the direction of the cutter motion.
- \* G42. \* Cutter compensation, when the cutter is on the right side of the work surface.
- \* G43. \* Cutter compensation, to an inside corner.
- \* G44. \* Cutter compensation, to an outside corner.
- \* G45 to G49. Unassigned.
- \* G50 to G59. Reserved for adaptive control.
- \* G60 to G69. Unassigned.
- \* G70. \* Inch programming.
- \* G71. \* Metric programming.
- \* G72. Three-dimensional circular interpolation, clockwise.
- \* G73. Three-dimensional circular interpolation, counterclockwise.
- \* G74. Cancel multiquadrant circular interpolation.
- \* G75. Multiquadrant circular interpolation.
- \* G76 to G79. Unassigned.
- \* G80. Cancel cycle.
- \* G81. \* Drill cycle.
- \* G82. \* Drill with dwell.
- \* G83. \* Intermittent or deep hole drilling.
- \* G84. \* Tap cycle.
- \* G85 to G89. Boring cycles.
- \* G90. \* Absolute programming.
- \* G91. \* Incremental programming.
- \* G92. Preload values such as axis-position registers.
- \* G93. Inverse-time feed rate.
- \* G94. \* Inches or millimeters feed rate.
- \* G95. \* Inches or millimeters per revolution feed rate.
- \* G97. \* Spindle speed in RPM.
- \* G98 and G99. Unassigned.

## OTHER ADDRESS CHARACTERS

The other address characters used are as follows (asterisks mean “most commonly used”).

- \* A. Angular dimensions about the *X* axis.
- \* B. Angular dimensions about the *Y* axis.

- \* C. Angular dimensions about the Z axis.
- \* D. Can be used for an angular dimension around a special axis, for a third feed function, or for tool offset.
- \* E. Same as D but for a second feed function.
- \* F. \* Feed rate.
- \* H. Unassigned.
- \* I, J, K. \* Used with circular interpolation. You will see these most frequently.
- \* L. Not used.
- \* M. \* Miscellaneous functions.
- \* N. \* Sequence number.
- \* O. Used in place of the customary sequence number word address N.
- \* P. A third rapid traverse code parallel to the X axis.
- \* Q. Second rapid traverses code parallel to the Y axis.
- \* R. First rapid traverse code parallel to the Z axis.
- \* T. \* Tool change number.
- \* U. Secondary motion parallel to the X axis.
- \* V. Secondary motion parallel to the Y axis.
- \* W. Secondary motion parallel to the Z axis.

## **ABSOLUTE AND INCREMENTAL PROGRAMMING**

These are the two methods used in programming a part.

Absolute programming is using a reference or common point and positioning the cutter in all positions with reference to that starting point. The advantage of absolute positioning is that an error in positioning that occurs in one place will not progressively recur.

Incremental positioning is positioning the cutter from the last or previous position. If an error is made in incremental positioning, then all subsequent positions will be in error. This is not to say either method is better for they both are used, depending on what the program requires.

In programming, these can be interchanged throughout the program. It all depends on the type of program desired.

## **INTERPOLATION**

This is simply a function of the control to generate simultaneous movement of two or more axes (if machine is capable) such as angular or circular requirements.

## **SUBROUTINES**

These are instructions or commands that can be recalled during a program to repeat a portion of that program without being required to rewrite it every time it is needed. For example, if a slot is to be put into a program at certain points, the



slot program (subroutine) can be put into the program and then recalled when needed.

## PROGRAMMING THE WHEEL PULLER

The print of the wheel puller shows the calculations that must be made to properly place the cutting tool for machining the part. These are essentially the same calculations that were made earlier for machining the part manually.

The following calculations should be made and the results should be kept for use when programming the part: the radii the bolt circle diameter, the length of the slots and their angular location and the type and size of the milling cutters that will be used.

Sequence numbers are started with an N and are four digits. The practice is to leave space between each sequence number so commands later needed can be inserted where they will be used. For example:

```
N0010 XXXXXXXXXXXX
N0015 XXXXXXXXXXXX
N0020 XXXXXXXXXXXX
etc.
```

This leaves four spaces in between each sequence number to insert. Some may leave as many as nine spaces in between.

The tool is usually placed at a point called the zero-zero point. This can be home; that is, when the part is finished the tool moves back to the starting point, thus allowing the part to be clear of the cutting tool to be removed and a new part is inserted to be machined.

### Sample Program

From the zero-zero point the tool is raised up in the spindle to avoid having the tool crash with the work, and a feed rate is given for that movement (see Figure 8.4).

```
N0015 G00 Z0 F35
```

Next stop the spindle from moving:

```
N0020 M5
```

And turn the coolant off.

```
N0025 M9
```

Now, move the cutting tool to the zero-zero point to the X and Y axis in rapid traverse so a cutting tool can be put in the spindle:

```
N0030 G00 X0 Y0
```

Call out tool number 1 and label it 2-flute ball end mill.

```

(***05-16-86 set-up=D2000D.S)
(***5 Gm S)
(PULL )
N0010 (9) T0
N0015 G00 Z0. F35
N0020 (9) M5
N0025 (9) M9
N0030 G00 X0. Y0.
N0035 (T) 1/2FBEM
N0040 (9) M0
N0045 (9) M3 T1
N0050 (T) R764
N0055 (9) M0
N0060 (9) M8
N0065 G00 Z0.1
(88810 Gr111 111 o0. s).25 G40p003s1 Z0. T1)
N0070 G00 X0. Y0.0002 Z0.1
N0075 G01 Z0. F10
N0080 G01 X6. F35
N0085 G01 Y0.2502
N0090 G01 X0.
N0095 G01 Y0.5002
N0100 G01 X6.
N0105 G01 Y0.7502
N0110 G01 X0.
N0115 G01 Y1.0002
N0120 G01 X6.
N0125 G01 Y1.2502
N0130 G01 X0.
N0135 G01 Y1.5002
N0140 G01 X6.
N0145 G01 Y1.7502
N0150 G01 X0.
N0155 G01 Y2.0002
N0160 G01 X6.
N0165 G01 Y.2502
N0170 G01 X0.
N0175 G01 Y2.5002
N0180 G01 X6.
N0185 G01 Y2.7502
N0190 G01 X0.
N0195 G01 Y3.0002
N0200 G01 X6.
N0205 G01 Y3.2502
N0210 G01 X0.
N0215 G01 Y3.5002
N0220 G01 X6.
N0225 G01 Y3.7502
N0230 G01 X0.
N0235 G01 Y4.0002
N0240 G01 X6.
N0245 G01 Y4.2502
N0250 G01 X0.
N0255 G01 Y4.5002
N0260 G01 X6.
N0265 G01 Y4.7502
N0270 G01 X0.
N0275 G01 Y5.0002
N0280 G01 X6.
N0285 G01 Y5.2502
N0290 G01 X0.
N0295 G01 Y5.5002
N0300 G01 X6.
N0305 G01 Y5.7502
N0310 G01 X0.
N0315 G01 Y6.
N0320 G01 X6.
(***15 Gp Z0.1 T1)
N0325 G00 Z0.1
N0330 G00 X6. Y6.
(***20 Gr 121 o0.03 s0.25 G41p001p003s21 Z-0.5 T1)
N0335 G00 X0. Y0. Z0.1
N0340 G01 Z-0.5 F10
N0345 G01 X6. F35
N0350 G01 Y0.095
N0355 G01 X0.
N0360 G01 X6.
N0365 G00 Z0.1
N0370 G00 X0. Y0.095
N0375 G01 Z-0.5 F10
N0380 G00 Y0.345
N0385 G01 X2.3755 F35
N0390 G02 X2.1907 Y0.595 I3. J1.
N0395 G01 X0.
N0400 G00 Y0.845
N0405 G01 X2.1084
N0410 G02 X2.095 Y1. 13. J.1.
N0415 G01 Y1.095
N0420 G01 X0.
N0425 G00 Y1.345
N0430 G01 X2.095
N0435 G01 Y1.595
N0440 G01 X0.
N0445 G00 Y1.845
N0450 G01 X2.0834
N0455 G03 X2.0295 Y2.095 I0.625 J1.661
N0460 G01 X0.
N0465 G00 Y2.345
N0470 G01 X1.9261
N0475 G03 X1.7601 Y2.595 I0.625 J1.661
N0480 G01 X0.
N0485 G00 Y2.845
N0490 G01 X1.4962
N0495 G03 X1.157 Y3.0313 I0.625 J1.661
N0500 G02 X1.0255 Y3.095 I1.4845 J3.875
N0505 G01 X0.
N0510 G00 Y3.345
N0515 G01 X0.7509
N0520 G02 X0.6239 Y3.595 I1.4845 J3.875
N0525 G01 X0.
N0530 G00 Y3.845
N0535 G01 X0.3799
N0540 G02 X0.3795 Y3.875 I1.4845 J3.875
N0545 G02 X0.6066 Y4.095 I1.4845 J.3.875
N0550 G01 X0.
N0555 G00 Y4.345
N0560 G01 X0.7111
N0565 G02 X0.9362 Y4.595 I1.4845 J3.875
N0570 G01 X0.
N0575 G00 Y4.845
N0580 G01 X1.6986
N0585 G03 X2.0253 Y5.095 I1.3638 J5.6208
N0590 G01 X0.
N0595 G00 Y5.345
N0600 G01 X2.1633
N0605 G02 X2.3181 Y5.595 I3. J5.
N0610 G01 X0.
N0615 G00 Y5.845
N0620 G01 X2.674
N0625 G02 X3. Y5.905 I3. J5.
N0630 G01 X0.
N0635 G01 X6.
N0640 G01 X0.
N0645 G01 X6.
N0650 G01 Y6.
N0655 G01 X0.
(*** 25 Gp Z0.1 T1)
N0660 G00 Z0.1
N0665 G00 X0. Y6.
(***30 Gr 121 o0.03 s0.25 G41p003p001s2 Z-0.5 T1)
N0670 G00 X6. Y6. Z0.1
N0675 G01 Z-0.5 F10
N0680 G01 X0. F35
N0685 G01 Y5.905
N0690 G01 X6.
N0695 G01 X0.
N0700 G00 Z0.1
N0705 G00 X6. Y5.905
N0710 G01 Z-0.5
N0715 G00 Y5.655
N0720 G01 X3.6245 F35
N0725 G02 X3.8093 Y5.405 I3. J5.
N0730 G01 X6.
N0735 G00 Y5.155
N0740 G01 X3.9312
N0745 G03 X4.1872 Y4.905 I4.6362 J5.6208
N0750 G01 X6.
N0755 G00 Y4.655
N0760 G01 X4.9745
N0765 G02 X5.2491 Y4.405 I4.5155 J3.875
N0770 G01 X6.
N0775 G00 Y4.155
N0780 G01 X5.3761
N0785 G02 X5.42 Y3.905 I4.5155 J3.875
N0790 G01 X6.
N0795 G00 Y3.655
N0800 G01 X5.3934
N0805 G02 X5.2889 Y3.405 I4.5155 J3.875
N0810 G01 X6.

```

Figure 8.4. Manual program of the wheel puller.

```

N0815 G00 Y3.155
N0820 G01 X5.0438
N0825 G02 X4.843 Y3.0313 I4.5155 J3.875
N0830 G03 X4.5919 Y2.905 I5.375 J1.661
N0835 G01 X6.
N0840 G00 Y2.655
N0845 G01 4.292
N0850 G03 X4.1072 Y2.405 I5.375 J1.661
N0855 G01 X6.
N0860 G00 Y2.155
N0865 G01 X3.9905
N0870 G03 3.9254 Y1.905 I5.375 J1.661
N0875 G01 X6.
N0880 G00 Y1.655
N0885 G01 X3.905
N0890 G01 Y1.405
N0895 G01 X6.
N0900 G00 Y1.955
N0905 G01 X3.905
N0910 G01 Y1.
N0915 G01 X3.9 Y0.905 I3. J1.
N0920 G01 X6.
N0925 G00 Y0.655
N0930 G01 X3.8367
N0935 G02 X3.6819 Y0.405 I3. J1.
N0940 G01 X6.
N0945 G00 Y0.155
N0950 G01 X3.324
N0955 G02 X3. Y0.095 I3. J1.
N0960 G01 X6.
N0965 G01 X0.
N0970 G01 X6.
N0975 G01 X0.
N0980 G01 Y0.
N0985 G01 X6.
(***35 Gp Z0.1 T1)
N0990 G00 Z0.1
N0995 G00 X6. Y0.
(***40 Gc1 o0. G41p024s2 Z-0.5 T1)
N1000 G00 X2.125 Y1. Z0.1
N1005 G01 Z-0.5 F10
N1010 G01 Y1.661 F35
N1015 G03 X1.1678 Y3.0593 I0.425 J1.661
N1020 G02 X1.4242 Y4.7479 I1.4845 J3.875
N1025 G03 X2.1819 Y5.3104 I1.3638 J5.6208
N1030 G02 X3. Y5.875 I3. J5.
N1035 G02 X3.8181 Y5.3104 I3. J5
N1040 G03 X4.5759 Y4.7479 I4.6362 J5.6208
N1045 G02 X4.8322 Y3.0593 I4.5155 J3.875
N1050 G03 X3.875 Y1.661 I5.375 J1.661
N1055 G01 Y1.
N1060 G02 X3. Y0.125 I3. J1
N1065 G02 X2.125 Y1. I3. J1.
(***45 Gp Z0.1 T1)
N1070 G00 Z0.1
N1075 G00 X2.125 Y1
(***50 Gm CT)
N1080 (P) T0
N1085 G00 Z0. F35
N1090 (P) M5
N1095 (P) M9
N1100 G00 X0. Y0.
N1105 (T) 3/8FBEM
N1110 (P) M0
N1115 (P) M3 T2
N1120 (T) R1019
N1125 (P) M0
N1130 (P) M8
N1135 G00 Z0.1
(*** Gc1 o0. G42p027s3 Z-0.5 T2)
N1140 G00 X2.9687 Y1. Z0.1
N1145 G01 Z-0.5 F10
N1150 G01 Y2.25 F35
N1155 G02 X3.0312 I3. J2.25
N1160 G01 Y1.
N1165 G02 X2.9687 I3. J1.
(***60 Gp Z0.1 T2)
N1170 G00 Z0.1
N1175 G00 X2.9687 Y1.
(*** Gc1 o0. G42p043p043s6 Z-0.5 T2)
N1180 G00 X1.5001 Y3.9021 Z0.1
N1185 G01 Z0.5 F10
N1190 G01 X2.3661 Y3.4021 I2.3505 J3.375
N1195 G02 X2.3349 Y3.3479 I2.3505 J3.375
N1200 G01 X1.4688 Y3.8479
N1205 G02 X1.5001 Y3.9021 I1.4845 J3.875
(***70 Gp Z0.1 T2)
N1210 G00 Z0.1
N1215 G00 X1.5001 Y3.9021
(*** 75 Gc1 o0. G42P031P031s4 Z-0.5 T2)
N1220 G00 X3.0312 Y5. Z0.1
N1225 G01 Z00.5 F10
N1230 G01 Y3.75 F35
N1235 G02 X2.9687 I3. J3.75
N1240 G01 Y5
N1245 G02 X3.0312 I3. J5.
(***80 Gp Z0.1 T2)
N1250 G00 Z0.1
N1255 G00 X3.0312 Y5
(***85 Gc1 o0. G42p037p037s5 Z-0.5 T2)
N1260 G00 X4.5312 Y3.8479 Z0.1
N1265 G01 Z-0.5 F10
N1270 G01 X3.4651 Y3.3479 F35
N1275 G02 X2.6339 Y3.4021 I3.6495 J3.375
N1280 G01 X4.4999 Y3.9021
N1285 G02 X4.5312 Y3.8479 I4.5155 J3.875
(*** 90 Gp Z0.1 T2)
N1290 G00 Z0.1
N1295 G00 X4.5312 Y3.8479
(*** 95 Gc1 o0. G42p049p049s7 Z-0.5 T2)
N1300 G00 X3.0781 Y3. Z0.1
N1305 G01 Z-0.5 F10
N1310 G02 X2.9219 I3. J3. F35
N1315 G02 X3.0781 I3. J3.
(***6p Z0.1 T2)
N1320 G00 Z0.1
N1325 G00 X3.0781 Y3
(***105 Gm E)
N1330 (P) T0
N1335 G00 Z0.
N1340 (P) M5
N1345 (P) M9
N1350 G00 X0. Y0.
N1355 (P) M30
END

```

Figure 8.4. (cont'd)

N0035 (T) 1/2FBEM

Next, make sure the program is stopped to put the tool in the spindle:

N0040 M0

Next turn the spindle on and set the RPM:

N0045 M3 T1

N0050 R764

Turn the spindle off:

N0055 M0

N0060 M8. Turn the flood coolant on.

With the flood coolant on, the spindle will revolve; therefore we have the command to stop the spindle from revolving.

Rapid traverse in and continue with the program. If no sign is before the axis such as *Z* then it is positive (+). A negative sign must be inserted before the axis.

N0065 G00 .1

N0070 G00 X 0. Y0.0002 Z 0.1

N0075 G01 Z 0. F10

N0080 G01 X 6. F35

N0085 G01 Y 0.2502

Notice the feed rate inserted for the *Z* axis and the *X* axis.

The *X*-axis feed rate also controls the feed rate for the *Y* axis unless otherwise stated. The feed rate is for machining and changes any other feed rate that may have been inserted for the *Z* axis, for rapid traverse, or for any other reason.

There are 271 commands to be made manually to complete the part. The time required for manual programming can take many hours.

When doing circular interpolation, the *X* and *Y* coordinates are the end point of the arc. Therefore the starting points of the arc cannot be *X* and *Y*, for that would confuse the machine. *I* is used to replace the *X* coordinate for the start point and *J* is used to substitute for the *Y* coordinate starting point. The radius *R* is then called out.

## SIMPLIFIED PROGRAMMING ON A CONTROLLER

Using a controller that is called “user friendly,” meaning it can be “talked to” with menu prompting is somewhat faster.

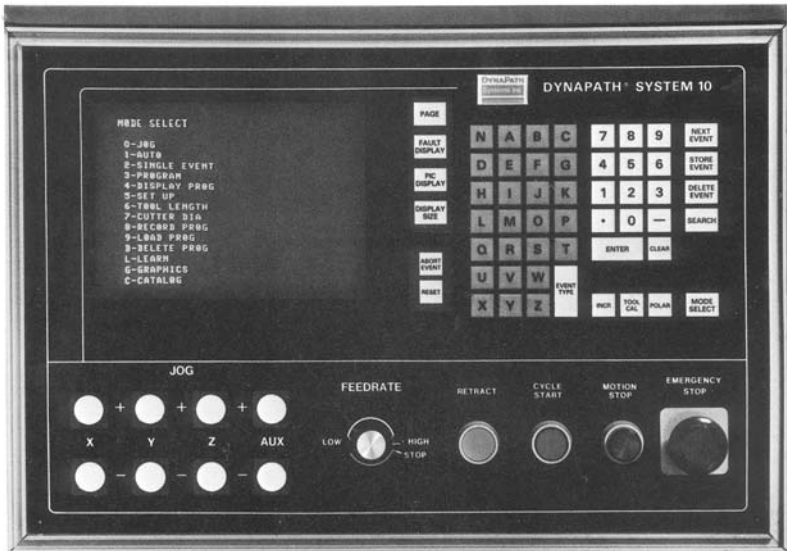
It must be pointed out here that while the programming is easier the number of commands are going to be the same as in manual programming. The timesaving comes in this order:

1. Sequence numbers do not have to be specified. The controller automatically does it.

2. M functions do not have to be spelled out. Specify what is wanted and the function is recorded.
3. G functions do not have to be spelled out. Select what is wanted and the controller will select the right G function.
4. All of this does require pushing the correct button and following the menu.
5. The program must be developed by the operator or the programmer and will have as many commands as the manual program.

This is very popular type of controller, and there are many styles made by various manufacturers (Figure 8.5).

### First, you tell it what you want.



To program the System 10, you just follow the step-by-step instructions displayed on the big 9-inch CRT. The information you'll need to program a part is shown in menu form.

Fill in the required details one line at a time using the codes shown on the screen. You don't have to respond to all the details—just those you need to program your part.

Input code numbers are displayed next to each operation and "point" to the correct key-switches at the operator's station.

It's all there on the CRT and it's presented in the right sequence for generating a part program. You provide the data, System 10 does the rest.

When you are sure the data you've entered is correct, press the STORE EVENT switch and go on to the next event. To construct a program, each event has to be stored before the next can be programmed.

You can tell the control what you want in polar coordinates and in inch or metric dimensions, absolute or incremental. In the SET-UP mode you can program mirror imaging. Another function control, "TOOL CAL," calibrates tool length.

Begin operation of the machine by selecting AUTO or SINGLE EVENT from the mode menu and pressing the CYCLE START button.

All you have to know before you start are the part dimensions and tolerances. A fault detection feature simplifies troubleshooting. Fault messages are displayed on the CRT so you can zero in on the problem.

Three controls to the right of the CRT screen let you double display size, search through multi-screen information a page at a time, and read the fault messages.

Once the control is installed, your operators won't have to leave for programmer's school. They can use shop floor language in writing programs and running parts. Either create a program before running the first part, or enter data and commands as you cut the first part.

Figure 8.5. Simplified controller assists.

# Then it tells you how to do it.

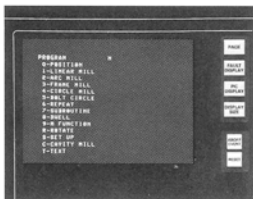
## MODE



Start programming by selecting the operational mode from the list displayed on the CRT when the control is first activated. Pressing "3" will call up the PROGRAM mode. Immediately a list of program events will appear on the CRT, replacing the MODE menu.

Multiple part programs can be stored, displayed and selected in catalog mode. Up to 99 programs can be stored. The part program identifier, number of characters used per program and storage remaining are displayed.

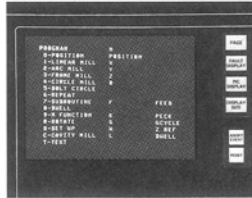
## PROGRAM



Select events by pressing the corresponding keyswitch. A data menu will appear to the right of the PROGRAM menu. Most of the time, programming is begun by pressing "9", the M FUNCTION event, to select the tool and start the spindle turning.

## O - POSITION

Position the tool to start the cut.



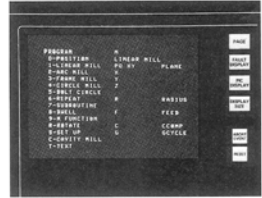
After you have programmed the tool and spindle speed, the next job is to get the tool to the beginning of the cut. So, you program the X, Y, and Z coordinates of that point. Or you can program POSITION using polar coordinates. Press "POLAR" before entering the dimensional information.

Then you enter the feedrate, "F", the depth of peck if required, and the G cycle if you need it. With one POSITION event, you can use any of six G cycles: drill, bore, counterbore, tap, peck drill, and off. "W", the Z reference plane, is the point above the work surface to which the tool will rapid advance before starting the feed. The tool will rapid back to W after the cut.

A lot of the time, the POSITION event is simply used to direct the tool to the point at which it will begin a cut. Then another event begins.

## 1.-LINEAR MILL

Linear mill for borders and straight cuts.



You can use "P" to select the plane you want the cutter to operate in. But it's not always necessary. Generally, to program LINEAR MILL you will enter the distance the cutter will travel in the X-direction, the feedrate in ipm, "F", and the distance in the Y-direction.

If you wish to blend two consecutive linear moves with an arc, simply enter an "R," value in the first linear event. This defines the radius of the arc.

Once feed is programmed in any of the events, it will continue that way unless or until it is specified differently. The feed is considered "modal." With feedrate override, though, you can increase programmed feeds by up to 150 percent.

In milling you may want to enter a cutter compensation to the left—"C1"— or the right—"C2"— or to cancel it, "CO".

LINEAR MILL can be programmed in polar coordinates. Press "POLAR" before entering the dimensions.

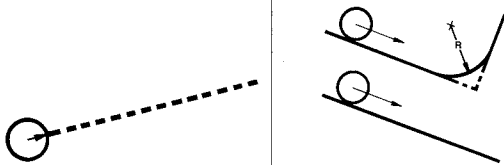
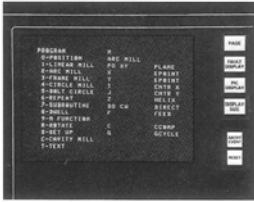


Figure 8.5. (cont'd)

**2.- ARC MILL**

Arc milling is simpler than it looks.



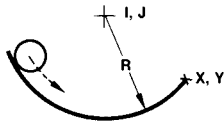
Press the "2" keyswitch to call up the ARC MILL event.

On the menu, "X" and "Y" are the axis endpoints; "I" and "J" are the distances in each of the axes to the center of the arc. "D" is the direction of the cutter path. Enter "D1" for the counterclockwise and "D0" for clockwise.

Program "F"; feedrate; use "C" to program cutter compensation. You can program a helical arc, too. With "Z," program for the depth of the cut desired at the "X" and/or "Y" endpoint.

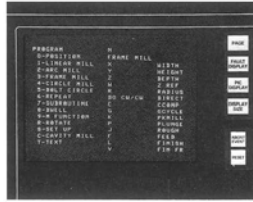
With "P" you select the plane you want the cutter to operate in.

You can program ARC MILL in polar coordinates, too. Just press "POLAR" before entering the dimensions, endpoints or radius.



**3.- FRAME & POCKET MILL**

Program complex tool paths with simple data.

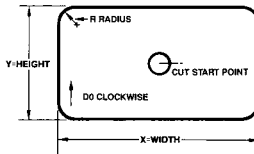


Frame and pocket milling are programmed with about the same kinds of data. The difference is the G cycle used for each. Frame milling is "G0" and pocket milling is "G7".

On the menu, "X" and "Y" are the width and height of the frame or pocket; "Z" is the finish Z-axis depth. "R" lets you program the corner radius of the frame or pocket.

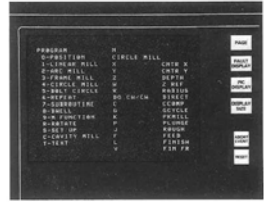
"D" the direction select, has its own short menu to program a clockwise or counterclockwise cutter path and "C," again, is for cutter compensation. With "W" you establish the Z-axis reference plane to which the tool will rapid advance before the feed begins.

"J" and "K" assign the amount of material removed in the "XY" and "Z" roughing passes and "L" assigns the amount of material removed in the finish pass. "P" is the Z-axis plunge rate, "F" is the roughing feedrate and "V" is the finish feedrate.



**4.- CIRCLE & POCKET MILL**

Circular interpolation made simple.



To circle mill, enter "G0" for the circular frame and "G7" for milling the circular pocket. "D" is the direction of the cutter path, and with "C" you program cutter compensation.

"J" and "K" assign the amount of material removed in the "XY" and "Z" roughing passes and "L" assigns the amount of material removed in the finish pass. "P" is the Z-axis plunge rate, "F" is the roughing feedrate and "V" is the finish feedrate.

In operation, the X- and Y-axes are fed around the rough X and Y dimensions you program into the control. The rough cut leaves enough stock for the finish pass. After it completes the finish cut, the tool exits automatically at a tangent and rapid returns to the start position.

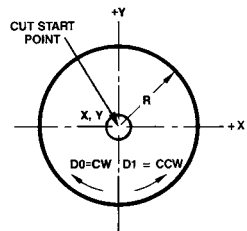


Figure 8.5. (cont'd)

**5.-BOLT CIRCLE**

Cut full circles or any size segment.



Pressing "5" will call up the BOLT CIRCLE event. "X" and "Y" are the center coordinates of the circle and "Z" is the finish depth of the holes.

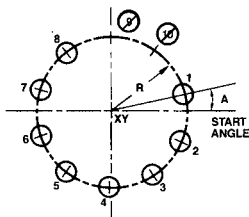
"A," the starting angle, is the angle at which the first hole will be drilled, and "R" is the radius of the pattern. You program feedrate, "F," and peck increments, "K."

You can use any G cycle—drill, bore, counterbore, tap, and peck drill.

The Z-axis reference plane, "W," is the point to which the tool will rapid advance before the actual feedrate begins.

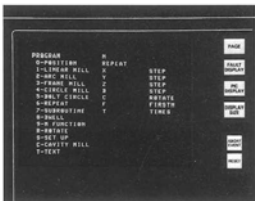
"L," dwell, is used only if you've programmed tap or counterbore.

Tell the control the number of holes you want in the circle, "H." In the drawing below, only 8 holes will be drilled in a 10-hole pattern. So, you would program 10.08.



**6.-REPEAT**

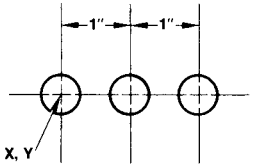
Repeat as many cuts as you want.



All you have to know to program REPEAT is the distance from one pattern to the next (the size of the incremental steps), the number of times it has to be repeated, and the sequence number in the program at which the repeat starts. "C," the rotation angle, allows you to rotate the pattern.

Substantial set-up time can be saved with this type of event. Each cut will be just like the last.

Enter "F," the starting sequence number of the repeat, and "T," the number of times you want to repeat that part of the program. You can program REPEAT in polar coordinates, too.



**7.-SUBROUTINE**

Subroutine means you can replay part of the program later on.



A simple way to repeat parts of a program, enter the first sequence number of the subroutine you want to call up and the last number of the sequence.

A repeated sequence of events in a part program can be prepared once, stored in the control, and "called" as a subroutine from a "main" part program whenever required.

Up to three levels of subroutine nesting are permitted.

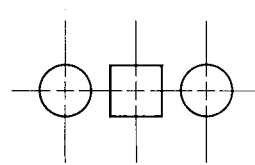


Figure 8.5. (cont'd)

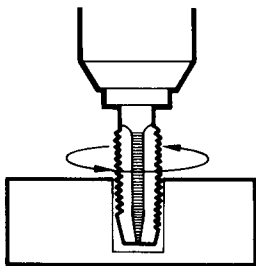


**8. -DWELL**

Dwell gives the spindle time to change direction.



Press "8" while in the PROGRAM mode and you can program a dwell time in seconds for the tool once it has reached depth. Use it to allow the spindle to change directions.



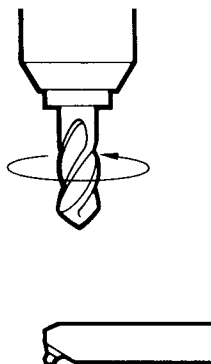
**9. -M FUNCTIONS**

Special codes make programming even faster.



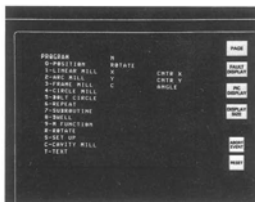
The M codes are special functions the System 10 passes on to the machine. Six of them are already defined and decoded by the control: M00 is for program stop; M03 cues the machine to start the spindle clockwise; and M04 starts the spindle the other way. M05 stops the spindle and M06 signals a toolchange. M30 is the end of a program.

There is a lot of room for assigning your own numbers.



**R-ROTATE**

Rotating part details saves set-up time.



Rotating the part detail lets you machine repeat cuts around a part. All the details will be the same, all will be concentric when you program the ROTATE event.

To use this powerful event, enter the centers of rotation in the X- and Y-axes and the angle of rotation, "C".

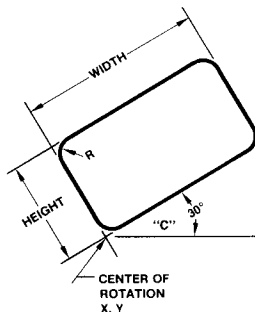


Figure 8.5. (cont'd)



Although there are some differences of approach, for most of the controllers, the basic idea is the same and they are very helpful when programming a part on a machine. Nevertheless, the basic principles of machining are still necessary: part dimensions, tolerances, speeds and feeds, and the tools to be used. Nothing comes easy, but maybe some things are a little easier.

The controller, located on the machine, has a cathode ray tube (CRT). It looks like a small television set. When the machine is turned on, a menu is shown, and the operator must make a selection for what he wants to do. In the picture shown, the mode select is on the screen. There are twelve selections to choose from, from jog to learn. Just about all are self-explanatory.

To write a program, select number 3, PROGRAM. When the PROGRAM select button is pushed, another menu appears: The first button selected is usually the M function number 9. Then the shape is called out, such as arc milling, frame milling, bolt circles, etc. Each time a selection is made, the figures are entered into the machine. These calculations must still be made by the programmer, but G codes and other codes are not necessary to record because the controller will automatically enter them when the type of machining is listed off of the menu.

In addition, some controllers supply graphics, which allows the operator to prove his or her program out on the CRT in graphic form. This is a real advantage because proving out a part usually requires one to make a part to see if it's OK. Usually, machining wax or plastic material is used to prove out a part, speeding up machining and saving the steel that is normally used when a part is made. The time to program a part with the controller is usually cut by about 25%.

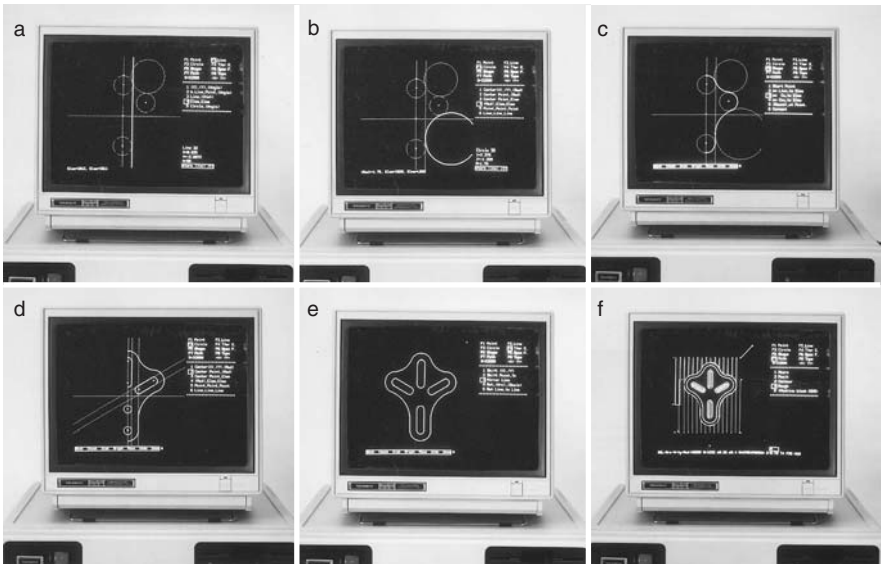
## PROGRAMMING WITH COMPUTER ASSIST

The final type of programming explained here is computer assist. Various manufacturers sell this type of software, used with a computer; it virtually eliminates math calculations, sines, cosines, angles, and other calculations. Programming is done directly by reading the drawings of the part. Because the software provides it, fractions are converted into decimal equivalents if wanted.

As shown in the photographs of one of these computer-assist systems, graphics and text are displayed simultaneously. Then geometric elements (points, lines, and circles) are defined. This geometry is used to define a shape and instruct the system for the tool path (which way to go) required for proficient machining. When defining the path, the system prompts the user about the radius of the tool, the feed rate wanted, and when to change the tool, if necessary.

The wheel puller is programmed following the sequence depicted in Figure 8.6. After the elements—geometry, shape, and tool path—are entered in the program, the computer generates a generic NC code for the machine.

The program can be proved out and edited, if desired, then converted to just about any format for a wide variety of NC machine controls. When printed out, the program looks exactly like the sample program for the other two methods of programming.



*Figure 8.6. Development of computer-assist programming. (1) Using the keyboard command of “F2 LINE”, a vertical line is placed tangent to the circles. (2) Similarly, “F3 CIRCLE” places a circle of known radius tangent to other elements—in this case a line and another circle. (3) Next, the computer “connects-the-dots” to outline the shape. (4) Shortcuts, specialized computer functions, complete the shape (here, using the mirror image step), still with minimal keyboard input. (5) The roughing pass of the pocket cut for slots is shown. (6) A complete tool path is shown that includes roughing and contouring passes.*

There are a few messages included in the program that alert the controller to the geometry generated. For example, the numbers in parentheses preceding the M function are only used to alert the machine for the M function being called out. The programmer does not enter them.

The program developed for the wheel puller takes about 20 min or less to do, depending on the experience of the operator and the skill he or she has in reading a print.

## CNC/CAD CAM

Before the advent of computer programs, numerical control described parts to be manufactured. Many more manual steps were required to create machining commands. Programmers converted machining commands into a computer language and keyed them into a controller, as shown on the previous pages. Next, the commands were transferred to a post processor, which converted the computer language to digital codes that, in turn, were punched into a tape. Next a tape reader transformed the digital codes into electric pulses that operated the spindle and drive motors.

Today, engineers create both part designs and manufacturing programs to make the parts on a computer. The engineer designs the part using a computer-aided design (CAD) program. This program then goes to a computer-aided manufacturing (CAM) program that, with subroutines that accommodate a wide range of parameters, generates the tool path. The computer then sends the CAM program as data signals to the spindle and machine drives.

NC (Numerical Control) became CNC when suppliers introduced computers into control systems. Increased computer memory meant added power that allowed for more sophisticated calculation, interpolation, and expanded ability to generate toolpath commands. When the computer is powerful enough to control several machine tools, the resulting system is called distributive numerical control (DNC). This eliminates a lot of paper and disk handling because one source generates the commands and distributes them electronically.

Specific commands, known as data blocks, such as G codes, preparatory commands, S codes, spindle commands, etc are handled with speed and accuracy by the computer. The lists of commands shown in this chapter comprise the data blocks. The computer can “think ahead” when making sharp turns or angles. This can prevent tool marks that, in traditional machining, typically would have to be polished out. Today, the computer can sense and handle hundreds of blocks ahead when necessary.

Prompting the computer through a series of screen prompts and questions is called conversational programming. This can be very helpful when the operator may know the workpiece material, the tool to use, and the toolpath, but not the best speeds and feeds. The program can then suggest the appropriate speed and feed based on the other operating conditions.

As we have shown in the lathe section, the computer can copy a manually made part. As the operator makes a part using manual control, the computer copies the steps and creates a program for all subsequent parts made on the same or equivalent machine. In addition, if the part has already been produced, it can be transformed into a program to reproduce that part by digitizing.

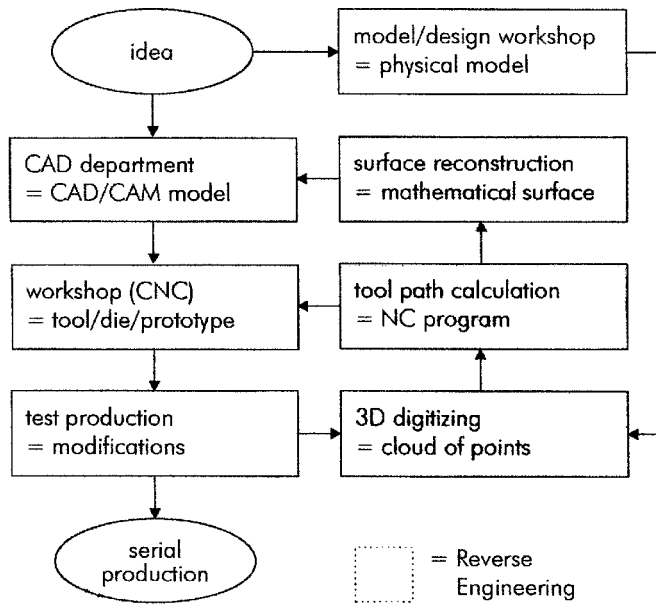
## REVERSE ENGINEERING

Today reverse engineering is becoming popular in many instances, and while it can be more sophisticated, it operates on the same principle as the operations cited above.

The method of reverse engineering is to make the first piece, many times called the prototype. With the development of digitizing, the computer can record the complete part with all of its required profiles using a scan-cad scan, a continuous tactile 3D digitizing procedure. To process the 3D data from, for example, a mill, three-dimensional digitized points are utilized. Using the scan system based on the points, surface reconstruction is then entered into the computer. The process is depicted in abstract form:

The systems can be adapted to any digitizing machine, machine tool, or coordinate measuring machine. The computer platform is a PC with DOS, Windows 95, 98, 2000, or NT. A probe, laser, or image systems are among the various methods used for digitizing.

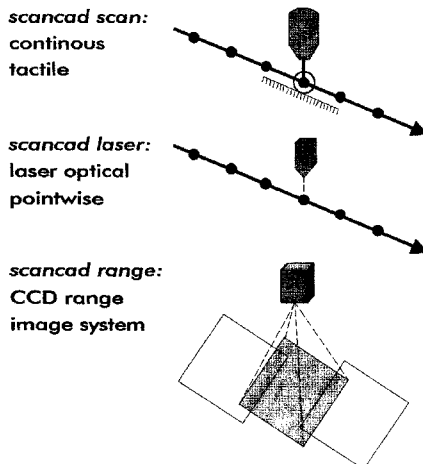
# Process Chain for Reverse Engineering



Reverse Engineering

(Courtesy of BCT GmbH, Dortmund, Germany).

## Measuring Methods for 3D Digitizing



(Courtesy of BCT GmbH, Dortmund, Germany).

The advantage using the digitizing method is that the part can be made out of wax or some very easy-to-machine material, allowing for fast and inexpensive production of the part and eliminating programming from a print or design. Digitizing can take place at night after the part has been made and production can take place the following day. All codes, movements, and calculations are automatically recorded in the computer.

The examples shown throughout the book all emphasize that whether making a part for machine repair or for production the initial part must be made.

## *Chapter 9*

# **COST PER CUT IN THE COMPUTER AGE**

## **INTRODUCTION**

Since the computer has become so important in manufacturing, speeds and feeds are included in many applications to help compute costs as well as machining factors.

We have added a run-down on basic computer operation and terms at the end of this chapter. This information may be helpful to those whose experience is more in use of computers than in the theory of their operation may find it useful, while those who are computer literate or expert can simply skip that portion of the chapter.

The one area that is generally ignored is the cost per cut for when computing the machine and other items used such as cutting fluids, overhead, tool costs, etc. Some companies make these calculations when they are actually manufacturing a particular part, but seldom do it prior to the acquisition of a machine when it can have the most beneficial effect upon subsequent machining operations by ensuring appropriate operating factors for the materials most often machines. Programming can help provide the most cost-effective operation of the cut-off machine being used, but it cannot compensate for inadequate rigidity, power, table travel, and similar considerations.

Below are some examples of cost per cut being applied on an overall basis of present manufacturing costs and anticipated cost. The following example can shorten the process of estimating a job and while estimates only work perfectly in perfect worlds, help ensure the calculations will be as accurate as the part, machines, cutting tools, fluids, and similar factors permit. Imputed costs for overhead, machine, and similar cost factors strongly influence cost per cut accuracy: even when computers are not used, the wry computer slogan, GIGO [garbage in, garbage out] still applies.



There are differences when estimating the cost per cut when turning or milling and when cutting off a part on a saw. When sawing, because of the nature of the removal of material, the amount of material is calculated directly in square inches per minute. For example, a 5" round of steel has 19.635 square inches of material to be removed. To compute the cost per cut we must add the cost of lubricant, overhead, machine cost, tool cost, etc.

If we are turning the part on a lathe then we must calculate the amount of cubic inches to be removed on each pass until arriving on our finished part. Each turn represents a cost per cut. The formula there is depth of cut, times RPM, times the feed rate. The number of cuts to the desired diameter represents the cost per piece. Again the actual cost must take in the following considerations, tool costs, lubricant costs, if used, the cost of the machine, overhead, etc.

For milling the tool feeds and speeds are governed by the rotation of the tool, differing in that respect from the lathe where the work is rotated. The computations are, however, the same when working with cubic inches being removed. Certainly when using a small fragile cutting tool the speeds and feeds will vary from those providing the optimal advantage in terms of material removal.

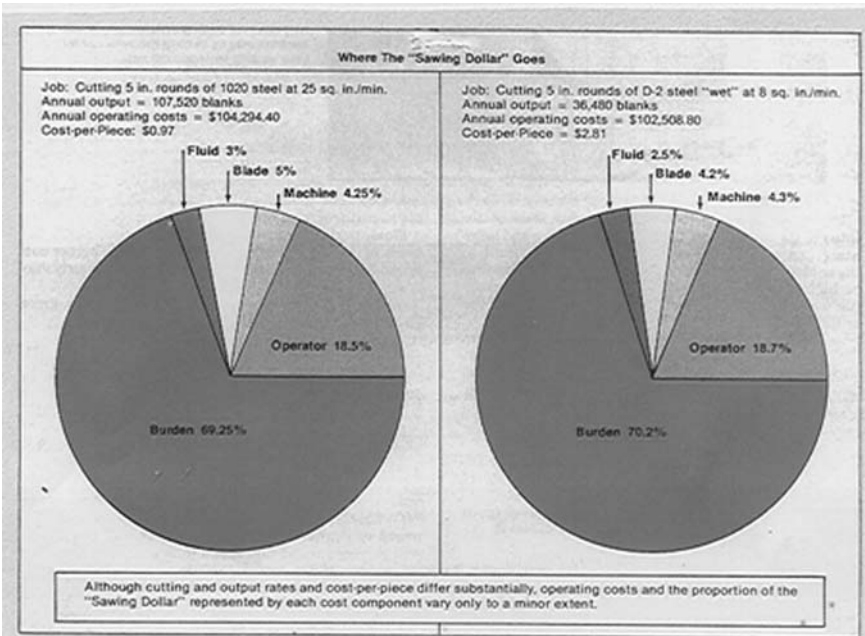
The pie charts depicting costs for power sawing operations in the accompanying illustration indicate that while component costs will vary, the contribution for each component typically varies only to a minor extent. The comparison of costs for a heavy duty saw and a medium duty saw illustrate how cost-per-cut for longer runs can be sharply reduced by using more rigid, high production equipment. By extension, it also indicates the cost-reducing effects of using the proper cutting fluid, blade or tool break-in where appropriate, and keeping the machine in proper adjustment. Computers can readily generate these kinds of graphic presentation that underscore cost component relationships to help guide sound decision-making.

Below are examples of computerized cost per cut. It does not matter what type of tool being used because the computer will take that into consideration when entering a highspeed cutting tool or Carbide cutting tool. The same applies to the machine being used. Each cost is taken into consideration. The computer calculates all figures entered into it.

The first cost per cut shown is based on sawing. Initially Material groups are entered.

## Main Material Groups

1. Carbon Steels	1008–1095
2. Free Machining Steels	1108–1213
3. Manganese Steels	1320–1345
4. Nickel Steels	2317–2517
5. Nickel Chrome Steels	3115–3315
6. Molybdenum Steels	4017–4150
7. Chrome Moly Steels	4130–4150
8. Nickel Chrome Moly	4317–4340
9. Nickel Moly Steels	4608–4820
10. Chrome Steels	5045–5160 (50100–52100)
11. Chrome Vanadium	6117–6152
12. Nickel Chrome Moly	8615–8750



**Table 1. Comparison of Costs for Gear Blanks Run on Medium vs. Heavy-Duty Cutoff Band Machine**

**THE JOB**  
Sawing a total of 1014 gear blanks, 1 1/2 inches thick, from 5-inch rounds of 4140 steel. Cross-sectional area per cut: 19.63 square inches. Total for run: 19,905 square inches.

	HEAVY-DUTY POWER SAW (\$10,000 model)	MEDIUM-DUTY POWER SAW
Machine Cost per hour	\$4.00, single-shift use	\$3.20, single-shift use
Blade	1.14 inch by 162 inches by 0.042 inch	1 inch by 144 inches by 0.035 inch
Cost per Blade	\$39	\$33
Blade Life	8000 square inches, 2 1/2 blades used for full run	4200 square inches, 4 7/10 blades used for full run
Cutting Fluid Replacement Cost per hour	\$0e	\$5e
Cutting Rate	16 square inches per minute	12 square inches per minute
Time Required for Full Run	25 hours	35 hours

**Table 2. Cost-Per-Cut Analysis**  
Burden and Operator Costs (\$35/hr and \$10/hr). Same for both Machines. Machine costs calculated on basis of one 8-hour shift per day.

	HEAVY-DUTY POWER SAW	MEDIUM-DUTY POWER SAW
BURDEN	25 x \$35.00 = \$875.00	35 x \$35.00 = \$1225.00
OPERATOR	25 x \$10.00 = \$250.00	35 x \$10.00 = \$ 350.00
MACHINE	25 x \$ 4.00 = \$100.00	35 x \$ 3.20 = \$ 112.00
CUTTING FLUID	25 x \$ 0.80 = \$ 20.00	35 x \$ 0.65 = \$ 22.75
BLADE COST	2.5 x \$39.00 = \$ 97.50	4.7 x \$33.00 = \$ 155.10
TOTAL OPERATING COSTS:	\$1342.50	\$1864.85
COST-PER-CUT	\$1342.50 / 1014 = \$1.3239 or rounded, \$1.32 per gear blank	\$1864.85 / 1014 = \$1.8391 or rounded, \$1.84 per gear blank

COST-PER-CUT DIFFERENCE: 52 cents per gear blank

*Where the "Sawing Dollar" Goes (Courtesy of DoAll).*

- |                         |           |
|-------------------------|-----------|
| 13. Silicon Steels      | 9255-9850 |
| 14. Miscellaneous       | .         |
| 17. Stainless Steels    | .         |
| 18. Copper Based Alloys | .         |
| 19. Nickel Based Alloys | .         |
| 20. Titanium Alloys     | .         |

Number of your Choice? This is your first selection

Assuming you select number 1, Carbon Steels 1008-1095 the following Material sub-groups will appear.

- |              |             |
|--------------|-------------|
| 1. 1008-1013 | 150-175 Bhn |
| 2. 1015-1035 | 160-175 Bhn |

- 3. 1040–1064      160–189 Bhn
- 4. 1065–1095      180–205 Bhn

Number of your choice. This is your second selection

After your second selection you are asked

What is the cutoff length of the part (inches). In this case we say 2.

The next question is what is the Part Shape?

- 1. Round Solid
- 2. Rectangular Solid
- 3. Round Tubing
- 4. Rectangular Tubing
- 5. Hex Solid
- 6. Angle
- 7. U-Channel
- 8. I-Beam, H-Beam

Enter number of your choice. We select number 1. The next page verifies our selection showing that our Part Shape is a Round Solid Diameter (5 inches) and the length is 2 inches long.

The following is as follows:

### Production Power Sawing Analysis

- 1. Material Group . . . . . Carbon Steels
- 2. Material Type . . . . . 1015–1035 160–175 Bhn
- 3. Material Shape . . . . . Round Solid
- 4. Material Size . . . . . 5" Diameter
- 5. Cut Surface Area . . . . . 19.635 Sq. In.
- 6. Cut Off Length . . . . . 2 in.
- 7. Machine Type . . . . . C1216A
- 8. Machine Cost . . . . . \$ 38,711.00
- 9. Blade Type . . . . . Super-Silencer
- 10. Blade Pitch . . . . . 3.5–5
- 11. Blade Length . . . . . 162 in.
- 12. Blade Width . . . . . 1.25 in.
- 13. Blade Cost . . . . . \$ 37.16
- 14. Band Speed . . . . . 295 SFM
- 15. Cutting Rate . . . . . 24.94 Sq. In./min.
- 16. Blade Life . . . . . 15152 Sq. In., 771 Pieces
- 17. Fluid Type . . . . . 600
- 18. Fluid Mix . . . . . 5:1
- 19. Fluid Cost . . . . . \$ 9.55 gal.
- 20. Operating Cost/Hr. . . . . \$ 40.00

Would you like to change any item (Y/N)

At this point anything whether cost, operation, machine can be changed with new figures put in if needed.

Finally the following comes up. The cost per cut

### Cutting Cost Analysis

Machine Cost = 0.0862

Blade Cost = 0.0482

Fluid Cost = 0.0163

Operating Cost = 0.6584

Total Cutting Cost/Piece = 0.8091

Producing 60.75 Pieces/Hour

### Cost per Cut for the Lathe or Milling Machine

The cost per cut for these machines is a little more complex, however the same principles apply. Each removal is figured in cubic inches, the amount removed based on the horsepower of the machine, the type of cutting tool used and the final total of material removed when the finished part is achieved. The other factors are basically the same. The cost of the machine used, cutting fluid cost, overhead, etc. The rule of thumb for material removal is for each horsepower generated by the machine one cubic inch of metal can be removed in one minute. Therefore a ten horsepower machine the amount of removal can be as much as ten cubic inches per minute. The following summary would look like this:

- Material Group . . . . .
- Material Type . . . . .
- Shape to be cut . . . . .
- Amount to be removed . . . . .
- Machine Type (Horsepower) . . . . .
- Cutting Tool Type . . . . .
- Cutting Tool Cost . . . . .
- Cutting Speed . . . . .
- Cutting Rate (cubic inches/min) . . . . .
- Tool Life . . . . .
- Cutting Fluid . . . . .
- Fluid Cost . . . . .
- Operating Cost (overhead) . . . . .

The Cost Analysis would be similar to the one above for each piece

- Machine Cost . . . . .
- Tool Cost . . . . .
- Fluid Cost . . . . .
- Operating Cost . . . . .
- Finally the Cost per Part . . . . .
- Also the number of Parts per Hour . . . . .

In most cases, the most cost-effective operation is obtained using the recommended speeds and feeds listed in tables provided by the cut-off machine or

machining system manufacturer. That is *always* the case when sawing or machining hard-to-cut materials, and, of course, any recommended blade or cutting tool break-in sequences should be implemented for maximum tool life. When working with easily machined materials, it may be prudent to do the cut-off or machining at somewhat less than optimal feeds and speeds to ensure longer blade life and thus lower blade cost in periods when relatively few pieces are needed and no real savings in machine, operator, or other costs can be realized. Conversely, when parts are badly needed and the need cannot be met any other way, it may be cost-effective to exceed recommended feeds and/or speeds to achieve higher output in readily machined materials at the cost of shorter blade or cutting tool life and thus higher tool cost. That, of course, must be done incrementally and cautiously, since setting these at excessively high rates could offset any advantage in higher output by sharply increased blade cost, particularly if that results in blade or cutting tool failure rather than merely reduced cutting life.

## COMPUTERS

The importance of the computer can be roughly gauged by the fact it has become a cliché to say computers have changed our lives. In addition to drastically altering such institutions as banks, stock and futures exchanges, and other financial markets; the computer increasingly and ever most strongly influences the way we shop, study, and communicate; and even marry. We can only guess the ways in which computer-related innovations and converging technologies will impact our schools government, industry, and every citizen. All are being subjected to accelerating change that promises to both complicate and ease our lives by letting embedded microchips put us in contact with our appliances, wake us and have the coffee and breakfast ready with the car warming up, the news we pre-selected by type posted on our converged computer-television-telephone-entertainment center-security system that will keep us in constant contact with everyone we choose.

In the shop will be machines driven by computers, designed by computers, parts designed, formed, inspected, packaged, shipped, billed, and entered into books for tax purposes, all handled automatically once properly set up and programmed.

It all sounds overwhelming.

But really, the computer and its microchip “brain” as well as embedded microchips that do not rise to the level of a functional computer are not that difficult to understand—conceiving, building, programming, and refining the computer was the difficult problem. Development of the microchip was an engineering marvel rivaled only by the astounding improvements in it. Nevertheless, our part, using the computer, is not that difficult to understand. As Franklin Roosevelt once noted in connection with another troubling situation, we have nothing to fear but fear itself.

For shop and plant purposes, we merely have to learn how to use available programs, operate the computer-controlled equipment, and provide some simple services the system requires. It obviously is best to be expert in computer technology,

but not essential. It does, however, help to understand how computers work. In this book, it is only necessary to know what steps are necessary to machine a part and to input the necessary parameters into the controller/computer, which then automatically uses canned routines to develop the machining sequence utilizing the codes needed to produce the part.

The following is designed to illustrate the process of developing a machining program should be helpful in understanding what is taking place.

All computers use the **ASCII** system and while the software is produce in languages such as Cobol, Fortran, Basic or what have you ASCII should be understood. It is an acronym for **American Standards Code for Information Interchange**. This code allows all digital computers to communicate with each other. The ASCII code is made up of seven bits. A bit is the smallest unit of measure in a computer word; several bits make up a byte, or a computer word. The seven bits when activated as a one and deactivated as a zero can then be shown as in letters or numbers when used in combination. An additional bit is added when parity is a requirement.

To understand the numbering system let's go to the decimal based system of 10. We are all acquainted with the ones, tens, hundredths thousandths, etc. This is called the base 10 system. ASCII does not use a base 10 binary system, instead it uses a base 2 binary system, and this requires only two symbols, 0 and 1 or on and off. Numbers larger than 1 are indicated by position. In this system each column is two times larger than the one to its right. Therefore the seven bits would look like this:

1	2	4	8	16	32	64
0	0	0	0	0	0	0

in the first column number one is represented, the second is two, the third, the third is four, the fourth is eight, the fifth is sixteen, the sixth is thirty two, and the seventh is 64. In positioning the numbers, and combining them along with the zeros as bits not needed are deactivated the combinations can represent letters and numbers in virtually an endless variety of combinations.

The zeros and ones or on and off provide the means for a computer to store, manipulate, and display information.

In addition, there are a few basic concepts underlying the microcomputer system that must be understood. First is memory. Memory is similar to the file systems and note pads we use manually to store information for later use or reference. Numbers, words, characters, instructions used in performing tasks the computer will handle are stored in its memory. Hence the bits, the Ones and the Zeros. This is digital communication.

In a computer two types of memory exists, stored memory that we use such as the hard disk that may be held there for future reference and temporary or "dynamic" storage memory that is necessary to allow entering of data that can be changed at any time. The computer can operate at any time with or without the stored data known as ROM, Read Only Memory. The computer cannot operate at all without the dynamic or temporary memory known as RAM, Random Access Memory.

Devices that permit input and output, including display or printing or disk storage are necessary to permit the computer to function. Sensory input elements keep track of conditions such as temperature, pressure, lights etc. The output elements enable the microcomputer to turn the power for motors, relays, lights, etc., on and off. The inputs allow the operator to send information to the computer using the keyboard, controller, voice-controlled devices, scanners, mouse, telephones and modems, cable, etc., while output devices send information through character display, printers, disks, and on-line and wireless transmission.

Digital signals can mean different things to different subsystems functional designs. Generally a given pattern or digital signals is assigned to given numbers, alphabetical characters or a command or instruction. When the pattern 0 or 1 represent numbers, further definition is required to understand what the numbers mean.

A given number can designate a subsystem or memory address in the computer or represent a specific quantity. It can also represent a time, distance, quantity or other limit. In all cases the pattern of 0s and 1s is a digital code that means a specific thing to a particular functional subsystem.

Terms it is useful to know include:

Circuit Board	—A special board on which specific circuits have been etched or printed.
Motherboard	—A printed circuit board with a specific bus structure that many different kinds of printed circuit boards can be plugged into.
Bus	—A wire that carries an electrical signal to two or more printed circuit boards.
CPU	—The central processing unit, the brain of the computer, the part that “thinks.”
RAM	—Random Access Memory. Temporary Storage used by the computer to allow changing of data put into it which is then transferred to a mass storage device.
ROM	—Read only memory. This is memory but it never forgets and cannot be changed.
EPROM	—Erasable programmable read only memory. Does not forget but can be erased by using an ultra violet light and reprogrammed with different information.
Interface Input Output	—A connector that translates between two parts of a system.
Serial Input Output	—Transmitting and receiving data to and from CPU through a single stream of electrical signals. Used with low quality long distance channels.
Parallel Input Output	—Used when several signals interchange at the same time. Happens fast for short distances.
Buffer	—Holding area for information to equalize or balance two different operating speeds.

Peripherals	—Displays, keyboards, scanners, modems, printers, etc.
Mass Storage	—Tapes, disks, etc.
Terminal	—Unit for connecting input, output, keyboard, display or printout.
CRT	—Monitor for displaying all printing, graphics, etc.
Bug	—The cause of malfunction, typically hard to find.

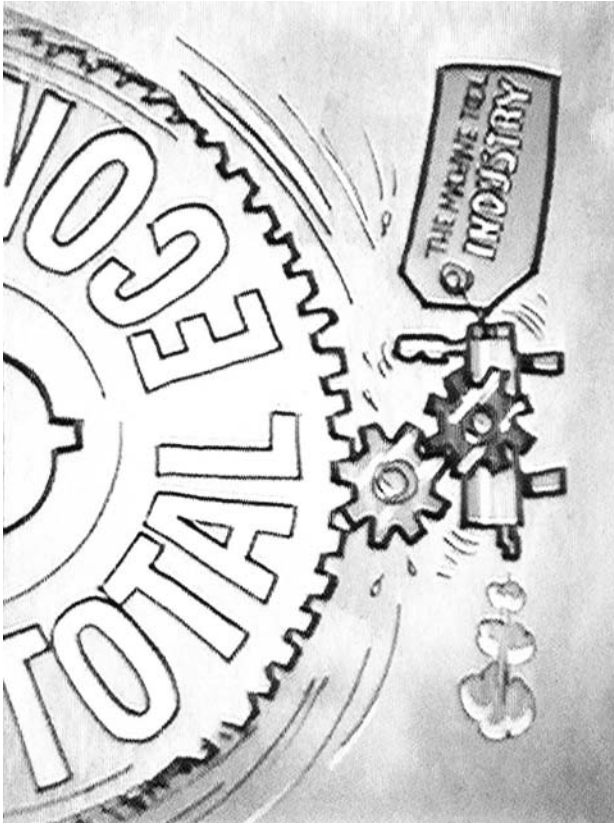


## SIMPLICITY AND COMPLEXITY

Those seeking scientific answers to “How did life begin?”; “What is the nature of reality?”; and “How do we know what we think we know?”, etc., delve into such esoteric subjects as Fractals, Chaos Theory, String Theory, Quantum Theory, Nanotechnology, and “Complexity: The Emerging Science at the Edge of Order and Chaos.” Yet despite widely divergent starting points, all seem to come to a set of reasonably similar observations: regardless of how complex a matter under scrutiny may be, there typically are a relatively few basic patterns or principles that interact to create dynamic and thus ever-changing complex systems.

Machining presents a case in point: from the earliest days of metal cutting and forming to today’s complex sophisticated hi-tech machining systems, such underlying basic factors such as material composition and hardness, cutting tool shape and deployment, speeds and feeds, power necessary for any given cutting or forming operation, etc., apply. Whether turning out turbine components on a giant FMS [Flexible Machining System] or manually machining a part on a basic lathe or milling machine, the cutting tool must be harder than the material to be cut and the speeds and feeds must be geared to the material and shape being cut. Hi-tech Computer-Aided-Design-and-Manufacture may seem galaxies apart from Wilkinson’s 1775 boring mill, the first true machine tool, but while control, tolerances, power, and metals and materials, and similar factors have improved by quantum factors, the same basic considerations nevertheless still govern machining.

Chaos Theory holds that minor changes can unleash major consequences—A butterfly flapping its wings in Tokyo can change the weather in Chicago or New York is an oft-cited analogy—and the evolution of modern machinery fits that theory’s implications to a “T.” The innovations John Wilkinson made to the basic design of centuries old boring devices not only made possible Watson’s steam engine, the driving power of the early Industrial Revolution, but triggered an unending series of most-often “minor” machining innovations that ultimately



*In ancient Greek mythology, Zeus appointed one of his sons, Haphaistos, as blacksmith and goldsmith to the heavenly court and Haphaistos created everything the other gods needed—the role played by machining and machine tools in modern society. The drawing portrays the machine tool industry as the gear that drives the global economy. This graphically makes the point that while machine tools constitute only a tiny fraction of man’s vast arsenal of manufacturing equipment, they create themselves and all of the rest of man’s wondrously productive modern hi-tech “workshop.”*

*There’s an ironic twist to the Haphaistos-machining comparison. The role machine tools play not only is little known and given scant credit for its prodigious contributions, but when it elicits any interest at all, it often tends to conjure up images of the hades-like setting and fiery, smoke-belching machinery of the early industrial revolution despite today’s typical spanking clean shops and operations.*

*Machining is new but the attitude is not. The ancient Greeks honored art but rarely the artist., the product but not the producer, what Haphaistos created but not Haphaistos himself. All of the Greek gods with the exception of Haphaistos were portrayed as handsome or beautiful and in perfect physical condition—only he was crippled and somewhat ugly, a pretty fair indication of the scant regard the ancients held for the Blacksmith to the gods and by extension, his human counterparts.*

*Nor is the machine tool “Blacksmith” of today’s world held in any higher esteem.*

created the ultra-sophisticated modern hi-tech world. Only machining systems capable of holding tolerances to millionths of an inch could have produced the components of the computers and semi-conductors that daily shrink and revolutionize our world. As the only machines capable of reproducing themselves as well as producing all other complex machinery, machine tools are the “breeding stock” of the industrial world.

Whether the subject is medicine, genetic engineering, Human Genomics, space travel, communications, travel, or large-scale production of toasters, microwave ovens, and virtually every other common product, research, study, progress, and production are governed by the quality and quantity of the nation’s and of the world’s machine tools and of the machines and devices that only they can create.

Much as the symbolic “butterfly wings” rarely register on our consciousness as a causative factor, the role machine tools play [or even their existence] is largely unknown to the man in the street while moreover, they comprise but a miniscule portion of today’s vast global repertory of machinery and devices.

Yet as the accompanying illustration graphically portrays, they produce and thus make possible ALL of the others as well as themselves.

# INDEX

---

## Index Terms

## Links

### A

abrasive saw	28	
use of	30	
wet-type	29	
abrasive wheel	29	
rubber-bonded wheel	31	
air-bearing spindle	208	
air gaging	5	
AISI [ <i>see</i> American Iron and Steel Institute (AISI)]		
aligning irregular parts for turning		
chalk method	61	
indicator method	62	
alloying elements	239	253
aluminum alloys	250	
American Iron and Steel Institute (AISI)	239	
angle plates	37	
angular milling cutter	108	
annealing	256	
arithmetical average,		
in surface finish	196	
recommended temperatures	257	
arbor		
for milling cutters	109	117

## Index Terms

## Links

arbor ( <i>Cont.</i> )		
of milling machine	104	
support	110	
axes, in numerical control	268	
<b>B</b>		
back-up bar	35	
band machining	33	
air-powered tables	37	
angle cuts	34	
cutting fluid	24	
disk-cutting attachment	37	
fixturing in	34	
locating stops in	34	
machining accuracy	35	
servo-feed attachment	36	
squaring bar	35	
use of	36	
welding a saw blade	40	
procedure	40	
band saw, cut-off	17	
band tool	19	
coolant/cutting fluids for sawing	24	
operating	22	
sample computer readout	20	25
sawing	17	
trouble shooting band problems	38	
trouble shooting machine problems	39	
bed length of lathe	45	

## Index Terms

## Links

boring			
bar	132		
chuck, dovetail offset	132		
on a drill press	166		
considerations	166		
procedure	134		
starter hole in	132		
tools	132		
bow-drill	43		
Brinell hardness number	259		
Brinell hardness test	259		
Brown & Sharpe taper reamer	154		
<b>C</b>			
calibration	1		
calipers	4		
inside	11		
outside	11		
verniers	10	12	
description of	13		
dial	13		
digital readout	13		
reading a	13		
scales of	12		
carbide cutting tools	262		
end mills	108	133	
turning	49	80	95
casehardening	257		
carbonaceous mixtures	258		
carburizing	258		

**Index Terms****Links**

casehardening ( <i>Cont.</i> )			
gas	258		
liquid	258		
hardening	258		
cyanide	258		
nitriding	258		
cast iron	4		
normalize	4		
plate	4		
stabilize	4		
centering	54		
gage	208		
tool	55		
center drilling	55	77	162
centers, for lathes	49		
dead	49	55	
live	49	55	
chucks	50	53	
air-actuated	50		
collet	50		
cradle	60		
four-jaw	51	53	
lathe	60		
mounting	60		
tapered spindle	60		
threaded spindle	60		
six-jaw	50		
three-jaw	50		
circular blade	29		

**Index Terms****Links**

circular saw	28		
use of	29		
cold-forming, in tapping	160		
counterbore	160	161	
styles of	162		
cold sawing ( <i>see</i> circular saw)			
collets	123		
holders	117		
column, of milling machine	103		
comparators	9		
dial	9		
electronic	9		
compound	48		
compound rest	48	64	
computers	294		
terms	296		
computer assist programming	267	284	285
cnc, cad-cam	286		
concave milling cutter	108		
contour saw	25		
hydraulic	38		
use of	26		
controller, in numerical control	267	284	
convex milling cutter	108		
coolant and dust collection, in grinding	192		
coordinate measuring machine	6		
digital readout display	7		
copper alloys	251		
corner-rounding cutters	108		



**Index Terms****Links**

Cost per cut	289	
cutting cost analysis		
for the lathe	293	
for the milling machine	293	
for the cut-off saw	290	
production power saw analysis	292	
counterboring on drill press	164	
countersink	161	162
adjustable	162	
chatterless	162	
machine	162	
on drill press	162	
with pilots	162	
couplings, taps for	159	
crop cut	23	
cross-feed screw	48	
cut counter	23	
cut-off	16	
cutting fluids	263	
machining selection chart	266	
cylindrical grinding attachment	191	192

**D**

depth gages	10	
depth of cut in grinding	201	
diamond dressing tools	187	
how to dress and true a		
grinding wheel	188	
diamond tools	5	
die casting alloys	252	

## Index Terms

## Links

digital readouts	117	
digital readouts, on lathe	48	51
dimensional tolerances		
( <i>see</i> tolerances, dimensional)		
dish grinding wheel	185	208
distance between lathe centers	46	
dividers	11	
dividing head	111	
cutting a helix angle with	114	
and machining circular slots	125	
dovetail		
cutter	107	
milling on rotary table	126	
offset boring chuck	122	134
drawbar, for milling machine cutter	110	123
drawing ( <i>see</i> tempering)		
drill bit cutting tools	142	
drill point angles	144	
fractional sets	143	
general-purpose	143	
grinding of	144	
helix angles	143	
letter sets	143	
lip angles	143	
Morse tapered shanks	143	
number sets	142	
twist	143	
types of	145	
web	144	
drill gage	145	

## Index Terms

## Links

deep hole drilling, gun			
drilling, spade drilling,			
trepanning	146		
deep hole drilling	146		
gun drills	151		
oil hole drilling	149		
spade drilling	150		
drilling on a drill press	162		
multiple drilling attachment	164		
preparation for	163		
drilling on a lathe	77		
boring	78		
counterbores	81		
procedure	78		
reaming	80		
safety	77		
spotfacing	82		
drill press	139		
gear-head	140	141	
operations of	149		
radial	142		
sensitive	139	140	141

## **E**

end-feed grinding, in centerless			
grinding	180		
endless blade	26		
end mills	106		
flute length	106		
flutes	106		

**Index Terms****Links**

end mills ( <i>Cont.</i> )		
number of flutes	106	107
roughing cutters	108	
shell	130	
straight shank	123	
<b>F</b>		
facing on a lathe	53	
between centers	54	
in a chuck	53	
ferrous metals	238	
filing on a lathe	84	
procedure	85	
removing burrs	84	
spindle speed	85	
toothforms, of file	84	
flaring cup		
grinding wheel	185	208
flexible rule	10	
flood cooling	192	
follower rest	46	52
use of	53	
form dresser	191	
form grinding	198	
free-machining steels	239	241
friability, of		
grinding wheel	181	197
fringe line	7	

## Index Terms

## Links

### G

gage blocks	2		
arithmetical progression of	3		
calibrating with optical flats	7		
Hoke blocks	4		
wringing	4		
gap-bed lathes	46		
gear cutters	107		
gear-head drill press	140	141	
gear-head lathe	48		
go/no go gage	132		
thread checking with	73		
grinding	174		
abrasive belt	180		
attachments and accessories	208		
centerless	179		
coolant (fluids)	194		
crush	199		
cylindrical	178	178	200
external	200		
flatness & finish	194		
forming	198	203	
internal	179	202	203
mandrels	208		
plain	202		
plain cylindrical	202		
plunge	198		
relief	195		
setups for various operations	216		

**Index Terms****Links**

grinding ( <i>Cont.</i> )			
surface	162	163	193
maintaining			
accuracy during	197		
taper	203		
tool and cutter	176	180	204
universal grinder	201		
grinding brazed			
carbide bits	90		
grinding high-speed steel bits	90		
grinding wheels	181		
aluminum oxide	182		
bond	181		
CBN	183		
Coolant (cutting fluids)	194		
cutter	208		
diamond	183		
dressing	188	199	202
friability	181	197	
grade	182		
grain size	181		
silicon carbide	182		
standards	184		
structure	182		
superabrasives	183		
trueing & balancing of	187		
types of	181		
use and care of	186		

## Index Terms

## Links

### H

hacksaw	31	
use of	31	
hacksaw blades	31	
handcrank, spindle	122	
hardening	253	
critical point	253	
decarburization	254	
“freeze” material	253	
quench, oil or water	255	
red heat hardness	254	
hardness testing		
Brinell	259	
Knoop hardness numbers	261	
Rockwell	259	
scleroscope	260	
Vickers	261	
handwheel, of lathe	48	
heat treatment of metals	253	
high-speed attachment,		
for milling machine	115	
Hoke, W.	2	3
Hoke blocks	4	
horizontal milling attachment,		
for vertical milling		
machines	115	
horizontal milling machining	127	
climb milling	129	
conventional milling on	129	

## Index Terms

## Links

horizontal milling machining ( <i>Cont.</i> )	
feeds and speeds for	128
surface feet per minute table	135
slab mill	127
setting up	128
types of operations	127
Hultgren ball, in Brinell hardness test	259

## **I**

index crank	111	
indexing head ( <i>see</i> dividing head)		
index plate	111	
index table	19	
indicator probe, for aligning work	120	
infeed grinding, in centerless grinding	180	
inspection of newly sharpened cutter	222	
interchangeable parts	2	
interference patterns	2	
lightwave	7	
interferometer	2	
interferometry	2	3
internal cut	25	40
international meter	3	

## **J**

jig	131	
bore	131	
grinder	131	
Johansson, C.E.	2	3



## Index Terms

## Links

### **K**

Kerf	33
Knoop hardness numbers	261
knurling on lathe	82
procedure	81

### **L**

lapping machine	3		
lathe dog	50	55	56
lathes			
accessories	49		
axes of	47		
basic lathe	44		
Ezpath	94		
model designations	45		
nomenclature	45	46	
operations of	43	47	
types of	44		
turning centers	93		
layout ink	26		
layout line	25	26	
length, of lathe	46		

### **M**

machine speed clamps	37		
mandrels	51		
master standard	1	2	
black granite	1	5	

**Index Terms****Links**

magnetic chuck	177	189	
Ceramax	190		
electromagnetic	189	191	
magna-vise	189	190	
mounting & dressing in parallels	199		
permanent magnet	189		
sine	191		
V-block	191		
vise	190		
Maudslay, H.	44		
and screw-cutting lathe	44		
measurement			
precision	11		
reference	11		
measurement			
standards	1		
Michelson, A.	2		
micrometers	4	9	13
digital	14		
features of	14		
reading a	14		
milling cutters	108		
half-side	109		
holders for	109		
plain	109	110	129
side	129		
staggered-tooth	109	129	
straight-tooth	129		

## Index Terms

## Links

milling machine				
accessories	111			
axes of	103			
basic functions of	101			
boring on	131			
cutting tools	105			
carbide	108			
digital readouts for	117			
head	102			
headstock	111			
helical milling	114			
horizontal	101	104	119	127
universal	111			
vertical	101	102	113	
components of	102			
machining centers	136			
muti-tool tool changer	138			
milling speeds and feeds	133			
Moiré fringe patterns	6			
monolite	7			
Morse taper reamer	154			
moveable workstop	36			
multiple drilling attachment	164			
multiple tapping attachment	164			
<b>N</b>				
nonferrous metals	238	250		
normalize, cast iron	4			
normalizing	257			

## Index Terms

## Links

numerical control programming	257			
absolute	273			
address characters	272			
home	274			
incremental	273			
interpolation	273			
machine codes (M functions)	270			
manual program	274			
preparator functions (G codes)	271	272	274	277
sequence numbers	274	277		
subroutines	273			
zero-zero point	274			

## **O**

optical flats	2	7		
optical form dresser	192			

## **P**

parallels	5			
pedestal grinder	88			
finishing wheel	88			
roughing wheel	88			
safety precautions	89			
photoelectric cell	6			
pilot hole	161			
in boring	134			
pipe threads, turning of	74			
planes of reference	4			

## Index Terms

## Links

plastics	238	253	
composite materials	253		
glass-reinforced	253		
plates			
drive	50	55	56
face	50	56	
plunger arm	111		
pneumatic measurement			
( <i>see</i> air gaging)			
polishing on a lathe	84		
aluminum oxide cloth	85		
procedure for filing	85		
procedure for polishing	86		
positive stop	23		
silicon carbide cloth	85		
spindle speeds	85		
precision surface grinding			
( <i>see</i> grinding, surface)			
prickpunch	26		
projects			
#1—Tap Wrench	97		
#2—Wheelpuller	168		
#3—Toolmaker's Vise	227		

## **Q**

quenching	253		
in air	253		
in oil	253		
in water	253		
quick-change gear box, on lathe	48		

**Index Terms****Links**

quick-change holders	117	
<b>R</b>		
radial drill press	140	142
drill head for	142	
radial arm	142	
radius and angle wheel dresser	191	
ram, of milling machine	102	
reamer		
adjustable	154	
chamfer angle	152	
chucking	153	
cutting edges	152	
dowel pin size	154	
feeds	152	
finishing	152	
flutes	152	
hand	153	
rose	152	
sharpening of	154	
tapered	154	
types of	153	
reaming	151	
on a drill press	162	165
relief angles	210	
indicator-drop method	210	
primary	210	
secondary	210	
resinoid bond	181	
reverse-engineering	286	

## Index Terms

## Links

ring gage	203		
Rockwell hardness test	259		
C-scale	260		
hardness number	260		
rotary chucks	191		
rotary table, on milling machine	114		
machining circular slots on	125		
milling dovetails on	126		
rough cut	16		
roughness width, in surface finish	197		
royal cubit	1		
rubber bond	181		
 <b>S</b>			
SAE [ <i>see</i> Society of Automotive Engineers (SAE)]			
saucer grinding wheel ( <i>see</i> dish grinding wheel)			
saw-guide post	37		
sector arm	111		
sensitive drill press	139	140	141
bench type	141		
cutting tools for	141		
floor-standing type	141		
setting gage	4		
shellac bond	182		
slotting attachment, for			
milling machine	115	116	
slug	16		

## Index Terms

## Links

Society of Automotive		
Engineers (SAE)	239	
speeds and feeds, for materials	261	
cutting speeds	261	
depth of cut	263	
feed rate	263	
revolutions per minute	262	
surface feet per minute	262	
spiral-fluted taper reamer	154	
spotface	160	162
squares	5	10
combination, as layout tool	11	
right-angle reference	11	
stabilize cast iron	4	
standards		
master	1	2
measurement	1	
temperature	3	
starter hole, in boring	132	
steady rest	52	58
use of	58	
steel rules	10	
scribed scales	10	
six-inch-rule scales	10	
steels, alloy	240	241
chromium in	241	
designation of	241	
machinability of	240	
nickel in	240	
silicon in	242	



## Index Terms

## Links

steels, high-speed tool	248
carburized, (mold steels)	249
case hardened, (mold steels)	249
hardened substrate, (mold steels)	249
“L” series, (high speed)	240
mold series	249
molybdenum	249
tungsten in	248
steels, stainless	244
steels, standard carbon	241
designation of	241
machinability of	240
phosphorus in	241
spread	239
sulfur in	241
tolerance	239
trace elements in	239
steels, tool	245
chromium in	248
cold work	247
hot heading dies	248
hot work	248
molybdenum in	248
oil hardening of	246
shock resisting	246
tungsten in	248
water hardening of	246
straight grinding wheel	208
superalloys	252

**Index Terms****Links**

surface				
finish	194			
specifying	194			
flatness	194			
surface plate	4			
three-plate method	4			
swaging, in tapping	160			
swing, of lathe	45			
<b>T</b>				
table of milling machine	97			
tailstock, of lathe	50	55	58	63
taper gage	203			
tapers	63			
American Standard machine	63			
attachments	65			
British Standard	63			
Brown & Sharpe	63			
Jacobs	63			
Jarno	63			
milling	132			
Morse	63			
taper micrometer	203			
taper turning attachments	50	52		
tapping	155			
attachment for	167			
hand	166			
machine	167			
on drill press	164	167	168	
tapping head	155			

## Index Terms

## Links

tapping locknuts, taps for	159	
taps	155	
Acme	159	
cut thread	155	
Dryseal	159	
forming	160	
ground thread	155	160
Hy-Hook taper pipe	159	
hand	156	
Hy-Spiral	158	
machine screw	158	
nomenclature	157	
nut	159	
pulley	159	
spiral point	156	
spiral point bottoming	156	
straight pipe	159	
tapper	158	
taper pipe	159	
tolerances	156	
types of	155	
tempering	255	
thread cutting on a lathe	68	
Acme	73	
cutting		
external	70	
internal	72	
methods	68	
tapping	75	
turning	68	

## Index Terms

## Links

thread cutting on a lathe ( <i>Cont.</i> )		
using a die	76	
worms	76	
threads	65	
characteristics of	66	
functions of	65	
measurement of	66	
nomenclature	67	
types of	65	
throughfeeding, in centerless grinding	180	
T-nuts	35	
tolerances, dimensional	4	
toolbit grinding	86	
left-hand	87	
nomenclature	87	
right-hand	87	
toolholder, on a lathe	50	53
tooth rest		
adjustable	208	208
shapes of	209	
positioning of	209	
setting	211	
trammels	11	
trouble shooting band problems	38	
trouble shooting		
machine problems	39	
T-slot		
cutter	107	127
milling	127	

## Index Terms

## Links

### turning

definition	43	
in face plate, method of	56	
irregular shapes	60	61
pipe threads	74	
round shapes	61	
taper	63	
turret, six-position	51	

## U

U.S. Bureau of Standards	3	
universal milling attachment	116	
universal spiral attachment	116	
universal workhead	208	

## V

variable-drive control lever	139	
Vblocks	5	
vertical milling attachment	116	119
vertical milling machine	94	118
aligning workhead	121	
angling the head	122	
components of	102	
facing on	130	
machining slots	123	125
machining Woodruff key	124	
milling compound angles	123	
swivel mounting	120	
tramming the head	119	

**Index Terms****Links**vertical milling machine (*Cont.*)

vitrified bond 181

**W**

waviness, in surface finish 196 197

wear land 210

welding a band saw blade 40

annealing blade 42

checking weld 42

procedure 40

removing flash 42

Whitney, E. 100

Woodruff key

cutter 107

on a vertical milling machine 124

woods 238

work hardening 262

working cubit 1

work measurement tools 5