DOCUMENT RESUME

ED 203 130 CE 029 243

AUTHOR	Bynum, Michael H.; Taylor, Edward A.
TITLE	Machinery Repairman 3 & 2. Rate Training Manual and Nonresident Career Course. Revised.
INSTITUTION	Naval Education and Training Command, Washington, D.C.
REPORT NO	NAVEDTRA-10530-E
PUB DATE	81
NOTE	671p.: Photographs and some diagrams will not reproduce well.
EDRS PRICE	MF03/PC27 Plus Postage.
DESCRIPTORS	Behavioral Objectives; Correspondence Study; *Equipment Maintenance; Hand Tools; Independent Study; Instructional Materials; Learning Activities; Machine Repairers; *Machine Tools; *Mechanics (Process); Military Training; Postsecondary Education; *Repair; Textbooks; *Trade and Industrial Education
IDENTIFIERS	Navy

ABSTRACT

This Rate Training Manual (textbook) and Nonresident Career Course form a correspondence self-study package to teach the theoretical knowledge and mental skills needed by the Machinery Repairman Third Class and Second Class. The 15 chapters in the textbook are (1) Scope of the Machinery Repairman Rating; (2) Toolrooms and Tools; (3) Layout and Benchwork; (4) Metals and Plastics; (5) Power Saws and Drilling Machines; (6) Offhand Grinding of Tools; (7) Lathes and Attachments; (8) Basic Engine Lathe Operations; (9) Advanced Engine Lathe Operations; (10) Turret Lathes and Turret Lathe Operations; (11) Milling Machines and Milling Operations; (12) Shapers, Planers, and Engravers; (13) Precision Grinding Machines; (14) Metal Buildup; and (15) The (Shipboard) Repair Department and Repair Work. Appendixes include Tabular Information of Benefit to Machinery Repairman (23 tables), Formulas for Spur Gearing, Formulas for Diametral Pitch System, and Glossary. The Nonresident Career Course follows the index. It contains 11 assignments, which are organized into the following format: textbook assignment and learning objectives with related sets of teaching items to be answered. (YLB)

MACHINERY REPAIRMAN 3 & 2

NAVAL EDUCATION AND TRAINING COMMAND

RATE TRAINING MANUAL AND NONRESIDENT CAREER COURSE

NAVEDTRA 1[illegible]530 E

Although the words "he," "his," "him" appear in this manual for economy in communication, they are not intended to indicate gender nor to affront nor to discriminate against anyone reading *Machinery Repairman 3 & 2*, NAVEDTRA 10530-E.

Naval Education And Training Program Development Center

MACHINERY REPAIRMAN 3 & 2

NAVEDTRA 10530-E

DEPARTMENT OF THE NAVY
CHIEF OF NAVAL EDUC AND TNG

1981 Edition Prepared by
MRCM Michael H. Bynum and MRC Edward A. Taylor

PREFACE

This Rate Training Manual and Nonresident Career Course (RTM/NRCC) form a self-study package to teach the theoretical knowledge and mental skills needed by the Machinery Repairman Third Class and Machinery Repairman Second Class. To most effectively train Machinery Repairmen, this package may be combined with on-the-job training to provide the necessary elements of practical experience and observation of techniques demonstrated by more senior Machinery Repairmen.

Completion of the NRCC provides the usual way of satisfying the requirements for completing the RTM. The set of assignments in the NRCC includes learning objectives and supporting questions designed to help the student learn the materials in the RTM.

Revised 1981

Stock Ordering No.
0502-LP-052-6510

Published by
NAVAL EDUCATION AND TRAINING PROGRAM
DEVELOPMENT CENTER

UNITED STATES
GOVERNMENT PRINTING OFFICE
WASHINGTON, D.C. 1981

THE UNITED STATES NAVY

GUARDIAN OF OUR COUNTRY

The United States Navy is responsible for maintaining control of the sea and is a ready force on watch at home and overseas, capable of strong action to preserve the peace or of instant offensive action to win in war.

It is upon the maintenance of this control that our country's glorious future depends; the United States Navy exists to make it so.

WE SERVE WITH HONOR

Tradition, valor, and victory are the Navy's heritage from the past. To these may be added dedication, discipline, and vigilance as the watchwords of the present and the future.

At home or on distant stations we serve with pride, confident in the respect of our country, our shipmates, and our families.

Our responsibilities sober us; our adversities strengthen us.

Service to God and Country is our special privilege. We serve with honor.

THE FUTURE OF THE NAVY

The Navy will always employ new weapons, new techniques, and greater power to protect and defend the United States on the sea, under the sea, and in the air.

Now and in the future, control of the sea gives the United States her greatest advantage for the maintenance of peace and for victory in war.

Mobility, surprise, dispersal, and offensive power are the keynotes of the new Navy. The roots of the Navy lie in a strong belief in the future, in continued dedication to our tasks, and in reflection on our heritage from the past.

Never have our opportunities and our responsibilities been greater.

CONTENTS

CREDITS

The illustrations indicated below are included in this edition of *Machinery Repairman 3 & 2*, through the courtesy of the designated companies, publishers, and associations. Permission to use these illustrations is gratefully acknowledged. Permission to reproduce these illustrations and other materials in this publication should be obtained from the source.

Source	Figures
American Technical Society	5-30, 5-31
Armstrong-Blum Manufacturing Company	5-1, 5-2
Armstrong Brothers Tool Company	7-24
Brown & Sharpe Manufacturing Company	2-7, 2-11, 2-12, 2-14, 2-17, 2-20, 2-21, 2-23, 9-12, 11-5, 11-7, 11-8, 11-9, 11-10, 11-16, 11-17, 11-20, 13-27, 13-29
Bullard Co.	10-42, 10-43
Cincinnati-Bickfort Tool Company	5-25, 5-27
Cincinnati Milacron	11-1, 11-2, 11-4, 11-11, 11-12, 11-13, 11-15, 11-18, 11-19, 11-79, 11-82, 13-11, 13-15, 13-20, 13-21, 13-22, 13-23, 13-25, 13-27, 13-28
Cincinnati Shaper Company	12-1, 12-3
Clausing Corporation	11-3
Cleveland Twist Drill Company	5-28, 5-37
DoAll Company	5-3, 5-5, 5-6, 5-7, 5-8, 5-9, 5-10, 5-11, 5-12, 5-13, 5-14, 5-15, 5-16, 5-17, 5-18, 5-19, 5-20, 5-21, 5-22, 5-23, 5-24, and table 4-1

Source	Figures
Delta Manufacturing Company	5-26
Giddings & Lewis Machine Tool Co., Division of Giddings & Lewis, Inc.	11-86
Groton Machine Company, Division of Kearney & Trecker Corporation	12-19, 12-20, 12-21, 12-22, 12-23, 12-24, 12-25, 12-26, 12-27, 12-28, 12-29, 12-30, 12-31, 12-32
Grinding Wheel Institute	13-4
Herbert D. Hall Foundation	5-33, 12-2
Kearney & Trecker Corporation	11-10
Lucas Machine Division, The New Britian Machine Company	11-87, 11-88, 11-89, 11-90
MacMillan Company*	4-2, 4-3
Monarch Machine Tool Company	7-1
Norton Company	13-13, 13-17, 13-18, 13-19
Precision Tool Co., Inc.	11-77
Reed-Prentice Corporation	7-3, 7-4
R.K. LeBlond Machine Tool Company	7-41
Rockford Machine Tool Company	12-13
SIFCO Metachemical Ltd.	14-11, 14-12, 14-13, 14-14, 14-15, 14-16, 14-17, 14-18, 14-19, and tables 14-1, 14-2, 14-3, 14-4, 14-5, 14-6, 14-7, 14-8, 14-9, 14-10, 14-11, 14-12
South Bend Lathe Works	5-32, 5-35, 7-2, 7-5, 7-9, 7-10, 7-11, 7-12, 7-13, 7-14, 7-15, 7-16, 7-17, 7-21, 7-27, 7-29, 7-32, 7-33, 7-34, 7-35, 7-36, 7-37, 7-38, 7-39, 7-40, 8-1, 8-4, 8-6, 8-9, 8-10, 8-11, 8-16, 8-18, 8-19, 8-20, 8-21, 8-22, 8-23, 8-24, 8-25, 8-26, 8-27, 8-28, 8-29, 9-2, 9-3, 9-4, 9-5, 9-6, 9-7, 9-8, 9-10, 9-11, 9-13, 9-19, 9-20, 9-21, 9-22, 9-23, 9-24, 9-25, 9-30

*From R. H. Drake: *Aircraft Sheet Metals* Copyright 1947

Source	Figures
L.S. Starrett Company	2-4, 2-5, 2-10, 2-13
Warner & Swasey Co.	10-3, 10-4, 10-5, 10-6, 10-8, 10-9, 10-10, 10-11, 10-12, 10-13, 10-14, 10-15, 10-16, 10-17, 10-18, 10-19, 10-21, 10-24, 10-25, 10-26, 10-27, 10-30, 10-31, 10-34, 10-35, 10-36, 10-37, 10-38, 10-39, 10-40, 10-41
Wilson Mechanical Instrument Company	4-4, 4-5

CHAPTER 1

SCOPE OF THE MACHINERY REPAIRMAN RATING

The official description of the scope of the Machinery Repairman rating is to "perform organizational and intermediate maintenance on assigned equipment and in support of other ships, requiring the skillful use of lathes, milling machines, boring mills, grinders, power hacksaws, drill presses, and other machine tools; portable machinery; and handtools and measuring instruments found in a machine shop." That is a very general statement, not meant to define completely the types of skills and supporting knowledge that an MR is expected to have in the different paygrades. The Occupational Standards for Machinery Repairman contain the requirements which are essential for all aspiring Machinery Repairmen to read and use as a guide in planning for advancement.

The job of restoring machinery to good working order, ranging as it does from the fabrication of a simple pin or bushing to the complete rebuilding of an intricate gear system, requires skill of the highest order at each task level. Often, in the absence of dimensional drawings or other design information, a Machinery Repairman must depend upon ingenuity and know-how to successfully fabricate a rep. r part.

One of the important products you will realize from becoming a well trained and skilled Machinery Repairman is versatility. As you gain knowledge and skill in the operation of the many different types of machines found in Navy machine shops, you will realize that even though a particular machine is used mostly for certain types of jobs, it may be capable of accepting many others. Your imagination will probably be your limiting factor and if you keep your eyes, ears, and mind open, you will discover that there are many things going on around you that can broaden your base of knowledge. You will find a certain pleasure and a source of pride in developing new and more efficient ways to do something that has become so routine that everyone else simply accepts the procedure currently being used as the only one that will work.

The skill acquired by a Machinery Repairman in the Navy is easily translated into several skills found in the machine shops of private industry. As a matter of fact, you would be surprised at the depth and range of your knowledge and skill compared to your civilian counterpart when based on a somewhat equal length of experience. The machinist trade in private industry tends to break up the job descriptions into many different titles and skill levels. The beginning skill level and one in which you will surely become qualified is "Machine Tool Operator," a job often done by semiskilled workers. The primary requirement of the job is to observe the operation, disengage the machine in case of problems and possibly maintain manual control over certain functions. Workers who do these jobs usually have the ability to operate a limited number of different types of machines. Another job description found in private industry is "Layout Man." The requirement of this job is to layout work that is to be machined by someone else. An understanding of the operation and capabilities of the different machines would be required as well as the ability to read blueprints. As you progress in your training in the Machinery Repairman rating you will become proficient in interpreting blueprints and planning the machining operations required. You will find that laying out intricate parts is not so difficult with this knowledge. A third job description is "Set-up Man," a job which requires considerable knowledge and skill, all within what you can expect to gain as a Machinery Repairman. A set-up man is responsible for placing each machine accessory and cutting tool in the exact position required to permit accurate production of work by a machine tool operator. An "All Around Machinist" in private industry is the job for which the average Machinery Repairman would qualify as far as knowledge and skill is concerned. This person would be able to operate all machines in the shop and manufacture parts from blueprints. Some Machinery Repairmen will advance their knowledge and skills throughout their Navy career to the point that they could move into a job as a "Tool and Die Maker" with little trouble. They also acquire a thorough knowledge of engineering data relative to design limitations, shop math and metallurgy. There are many other related fields where an experienced Machinery Repairman could perform—instrument maker, research and development machinist, toolroom operator, quality assurance inspector, and of course the supervisory jobs such as foreman or superintendent.

The obvious key to holding down a position of higher skill, responsibility, and pay is the same both in the Navy and in private industry. You must work hard, take advantage of the skills and knowledge of those around you, and take pride in what you do regardless of how unimportant it may seem to you. You have a great opportunity ahead of you as a Machinery Repairman in the Navy; a chance to make your future more secure than it might have been.

TYPICAL ASSIGNMENTS AND DUTIES

As a Machinery Repairman you can be assigned to a tour of duty aboard almost any type of surface ship, from a small fleet tug, which has a small 10- or 12-inch lathe, a drill press and a grinder, to a large aircraft carrier that is almost as well equipped in the machine shop as a tender or repair ship. You will find that although a ship's workspace is relatively small the machine shop will have more equipment than you might imagine. A lathe, drill press and grinder can almost be assured, but in many cases a milling machine and a second lathe are also available. A tender or repair ship is similar to a factory in the types of equipment that are installed. You will find the capabilities of such a ship to be very extensive in all areas required to maintain the complex ships of today's Navy. A Machinery Repairman is not destined to spend an entire career on sea duty. There are many shore establishments where you may be assigned. The Navy has shore-based repair activities located at various places throughout the United States and overseas. Most of these have wide-ranging capabilities for performing the required maintenance. There are general billets or assignments ashore that will not necessarily be associated with the Machinery Repairman rating, but which add to an individual's overall experience in other ways.

It would be difficult to attempt to go into detail on the duties that you may perform at each of your assignments. You will find that on small ships you may be the only Machinery Repairman aboard. This requires that you be self-motivating toward learning all you can to increase your ability as a Machinery Repairman and that you seek advice from sources off your ship when you have an opportunity. You will be surprised at how good you really are when you make an honest effort to do your best. Regardless of your assignment, you will have an opportunity to work with personnel from other ratings. This can be an experience in itself. There are many interesting skills to be found in the Navy. None of them are easy, but many will offer you some amount of knowledge that will increase your effectiveness as a Machinery Repairman or a home do-it-your-selfer.

TRAINING

Training is the method by which everyone becomes knowledgeable of and skilled in any activity, whether it's a job, a sport or something as routine as eating the proper foods. Training can take many forms and can be a conscientious or unconscientious effort on your part. However, you will make the most progress when you recognize the need to increase your level of knowledge, take the required action to obtain the training and fully apply all your efforts and resources to realize the maximum benefit from the training. In the following paragraphs, we will present a brief description of each of the types of training available to a Machinery Repairman. Keep in mind that the information listed is peculiar to your rating and that the Navy has many programs available which will allow you to increase your general education. You can obtain information concerning these programs from your career counselor or education officer.

FORMAL SCHOOLS

The Navy has available several schools which will provide an excellent background in the Machinery Repairman rating. You may have an opportunity to attend one or more of them during your career in the Navy.

The fundamentals of machine shop practice are taught in Machinery Repairman "A" school. Classroom instructions provide the theory of basic operating procedures, safety precautions and certain project procedures, while time spent in the ship provides a hands-on experience, supervised by a trained and skilled instructor. Some of the equipment that you can expect to work with in this course are lathes, milling machines, drill presses, band saws, cutoff saws, pedestal grinders and engraving machines. The length of the course and specific content may vary from time to time to accommodate the needs of the fleet. You will have no difficulty at all in performing the work in a Navy machine shop if you apply yourself in MR "A" school.

Advanced machine shop practice and the heat treatment of metals are taught in Navy schools also. These courses are usually attended by personnel in their second and subsequent enlistments at "C" school. Course content generally covers the information and associated equipment required for advancement to MR1 and MRC, although the schools are not required to establish eligibility for advancement.

You should consult with your leading petty officer or career counselor to obtain the most current information regarding school availability and your eligibility to request attendance.

RATE TRAINING MANUALS AND NONRESIDENT CAREER COURSES

Navy rate training manuals and nonresident career courses are designed as a self-study method to provide instruction to personnel in a variety of subjects. You can choose your own pace in working the courses, and you are allowed to refer to the book when trying to decide on the best or correct answer. If you are to learn anything, you must work the course yourself and not take the answers from someone else. Some rate training manuals and nonresident career courses are mandatory for you to complete to meet advancement requirements. These courses are listed in the *Manual For Advancement*, BUPERINST 1430.16 (series) and in the current (revised annually) issue of the *Bibliography For Advancement Study*, NAVEDTRA 10052 (series), where they are indicated by asterisks (*). Remember that as you advance you are responsible for the information in the rate training manuals for the paygrades below you, in addition to the courses for the next higher paygrade. A course offers an excellent opportunity to become familiar with a subject when you cannot be personally involved with the equipment. There are many small but important points that will be covered in a course that you otherwise may not learn.

ON-THE-JOB TRAINING

On-the-job training is probably the most valuable of all the training methods available to you. This is where you put the textbook theories and general procedures into specific job practice in personal contact with the problem at hand. All those unfamiliar terms that you read about in a course now begin to fit into a plan that makes sense to you. The one very important thing for you to remember is that when you are unsure about something, ask questions. An unusual job experience is of little value to you if you have to wing your way through it tooth and nail, guessing at each new step. The people that you work with and for had to learn what they know by asking questions, so they won't think you any less effi· · or valuable when you ask. There will be opportunities to tackle jobs which are difficult and seldom done, jobs which offer a great deal of experience and knowledge. These are the jobs that you should be really aggressive in pursuing and eager to accept. Regardless of the profession or the employer, the person who gets ahead is usually the one who is highly motivated toward increasing personal capacity, thereby, becoming more valuable to his employer. The Navy is no different than any other employer in this sense.

OTHER TRAINING MANUALS

Some of the publications you will use are subject to change or revision from time to time—some at regular intervals, others as the need arises. When using any publication that is subject to change or revision, be sure that you have the latest edition. When using any publication that is kept current by means of changes, be sure you have a copy in which all official changes have been made. Studying canceled or obsolete information will not help you to do your work or to advance; it is likely to be a waste of time, and may even be seriously misleading.

The training manuals you must use in conjunction with this one to attain your required professional qualifications are:

1. *Mathematics, Vol 1*, NAVEDTRA 10069-D and *Mathematics, Vol 2*, NAVPERS 10071-B. These two volumes provide a review of the mathematics you will need in shop work.
2. *Blueprint Reading and Sketching*, NAVEDTRA 10077-E, provides information on blueprint reading and layout work.
3. *Tools and Their Uses*, NAVPERS 10085-B, provides specific and practical information in the use of almost any handtool you are likely to use.

It is important that you keep abreast of required training manuals. To ensure that the most current manual is available, you should check *Bibliography For Advancement Study*, NAVEDTRA 10052 (series), and *List of Training Manuals and Correspondence Courses*, NAVEDTRA 10061 (series). Both of these references are revised annually, so be sure you have the latest one.

In addition, there are three sources of technical information that are ordinarily available on-board your ship: (1) *NAVSHIPS' Technical Manual*, which contains the official word on all shipboard machinery, (2) technical manuals provided by the manufacturers of machinery and equipment used by the Navy, and (3) machinist's handbooks. Most of these books should be readily available. However, if they are not, your leading petty officer or division officer can request them through proper channels.

SAFETY

As a Machinery Repairman, you will be exposed to many different health and safety hazards every day. A great many of these are common to all personnel who work and live aboard a Navy ship or station, and some are peculiar only to personnel who are involved with jobs within machinery spaces. Information concerning these can be found in both the *Fireman* and the *Basic Military Requirements* rate training manuals as well as instructions prepared by your command. In this section we shall look at some of the more common safety hazards that you will find in a machine shop and some of the precautions that you can take to prevent an injury to either yourself or someone else. You will find that safety is stressed throughout this manual as well as the importance of an individual's responsibility to not only be familiar with and observe all safe working standards personally, but also to encourage others to do so. Safety is a subject where the "learn by doing" method does not provide the greatest advantage.

Your eyes are one of your most priceless possessions. When you think about this fact and try to imagine how you would get along without them, you will agree that the slight inconvenience caused by wearing safety glasses, goggles or a face shield is a small price to pay for eye protection. Safety glasses or goggles should be worn any time you are around machinery in operation, including handtools, whether powered or nonpowered. Safety glasses that have side guards are the most effective for keeping out small metal chips or particles from grinding wheels. You should wear a face shield and safety glasses at all times whenever you are around any grinding operation.

Another item or protection is safety-toe shoes. Granted, the additional weight of the steel reinforced toe does not make them the most comfortable shoes that you can wear, but they do offer outstanding foot protection and they are much more comfortable than a cast. Look around your shop at the dents left in the deck from objects being dropped. Do you think your unprotected foot would fare any better?

Some of the objects you will be handling in the shop will have sharp or ragged edges on them that can cut easily. You should remove as many of these "burrs" as possible with a file. In spite of your filing efforts, heavy objects will still cut easily where there is a corner. A pair of leather or heavy cotton work gloves will protect your hands in these cases. You should NOT wear gloves when operating machinery. The chances of their being caught are too great.

Loose fitting clothing worn around moving machinery will test your strength if caught up in the rotating equipment. You would be amazed at the strength a shirt has when being wound up on a machine. Rings, bracelets and other jewelry can snag on projections of a rotating part and take a finger or other part of your body off before you know you have a problem.

How many times have you seen people bend over and pick up a heavy object by using their back? Chances are this same person will eventually injure himself or herself. The correct way to lift any heavy object is to get as close to the object as you can, spread your feet about a foot apart and squat down by bending your knees. You should keep your back straight during the lift. When you grasp the object, lift by using the muscles in your legs and hold the object close to your body. Walk slowly to your destination and lower the part exactly as you lifted it. If you have to lift something higher than your waist, you should seek assistance. Of course, there is a limit to how much weight anyone can safely pick up and this should not be exceeded.

Good housekeeping practices may demand a little more of your time than you are willing to give on some occasions, but this is just as important to a safe shop as any other measure that you can take. Small chips made during a machining operation can become very slippery when allowed to collect on a steel deck. Long, unbroken chips can trip or cut someone walking past them. Lubricating oil that has seeped from a machine or a cutting oil thrown out by the machine can be an extreme hazard on a steel deck. All liquid spillage should be cleaned up right away. If your job is causing a hazard to other personnel by throwing chips or coolant into a passageway, speak with your supervisor about isolating the immediate area by stretching tape across the area. Unused metal stock, small and large parts of equipment being worked on, toolboxes and countless other objects should not be left laying around the shop where traffic can be expected to go or where a machine operator may have to be positioned. Most well organized shops have a place for storing all movable objects and this is the place for them. It will save you time when daily cleanup or field day comes along, and it may save a serious injury.

To protect yourself from injury while operating ship machinery, there are several things you can do. The first thing is to make sure that you know how the machine operates, what each control lever does, the capability of the machine and especially where the stop button or clutch lever is in case an emergency stop is required. All guards that cover gears, drive belts, pulleys or deflect chips should be in place at all times. Use the correct tool for the job you are doing. This means more than using a scraper to remove paint instead of a 6-inch ruler. Every machine or handtool has a safe working limit that was determined by considering the stresses subjected to it under its intended use. Excessive pressures could cause failure in use with an injury following.

Whenever you are operating a machine, give it your total concentration. Daydreaming should be saved until a more relaxed time. If you must talk with someone, shut your machine off.

Electrical safety is not the private responsibility of the electricians. They can keep the equipment operating safely if they are notified when a problem exists. They cannot make everyone observe safety precautions when working around electrically powered equipment. This is a responsibility that each individual must accept and carry out.

The electrical systems used on-board ships are not like those found in your home, so however efficient you may feel you are as a handyman, do not attempt to make any repairs or adjustments on any faulty equipment on-board ship. Notify the electric shop and let the job be done by the trained electricians.

There are some basic safety precautions you can observe while using electrical equipment:

- Use only authorized portable electric equipment which has been tested by the electric shop within the prescribed time period and is properly tagged to indicate such a test.

- Report all jury-rigged portable electrical equipment to the electric shop.

- When a plastic cased or double-insulated electrically powered tool is available, use it in preference to an older metal cased tool.

- Ensure that all metal cased electrically powered tools have a three-conductor cable, a three-prong grounded plug and that they are plugged into the proper type receptacle.

- Wear rubber gloves when setting up and using the metal cased tools or when working under particularly hazardous conditions and in environments such as wet decks.

- Notify the electric shop when you feel even a slight tingle while operating electrical equipment.

- Follow the safety precautions exactly as prescribed by your maintenance requirement cards when performing maintenance on your equipment and be alert to other personnel who have taken similar precautions.

Always remember that electricity strikes without warning and, unfortunately, we cannot always sit around and discuss what went wrong after an accident has happened. It is to your advantage to ask when you are not sure of something. NEVER take unnecessary chances by hurrying or being inattentive. ALWAYS THINK about what you are going to do before you do it.

PURPOSES, BENEFITS, AND LIMITATIONS OF THE PLANNED MAINTENANCE SYSTEM

The following material contains a brief discussion on the purposes, benefits, and limitations of the Planned Maintenance System.

PURPOSES

The Planned Maintenance System (PMS) was established for several purposes:

1. To reduce complex maintenance to simplified procedures that are easily identified and managed at all levels.
2. To define the minimum planned maintenance required to schedule and control PMS performance.
3. To describe the methods and tools to be used.
4. To provide for the detection and prevention of impending casualties.
5. To forecast and plan manpower and material requirements.
6. To plan and schedule maintenance tasks.
7. To estimate and evaluate material readiness.

8. To detect areas that require additional or improved personnel training and/or improved maintenance techniques or attention.

9. To provide increased readiness of the ship.

BENEFITS

PMS is a tool of command. By using PMS, the commanding officer can readily determine whether his ship is being properly maintained. Reliability is intensified. Preventive maintenance reduces the need for major corrective maintenance, increases economy, and saves the cost of repairs.

PMS assures better records, containing more data that can be useful to the shipboard maintenance manager. The flexibility of the system allows for programming of inevitable changes in employment schedules, thereby helping to better plan preventive maintenance.

Better leadership and management can be realized by reducing frustrating breakdowns and irregular hours of work. PMS offers a means of improving morale and thus enhances the effectiveness of both enlisted personnel and officers.

LIMITATIONS OF PMS

The Planned Maintenance System is not self-starting; it will not automatically produce good results. Considerable professional guidance is required. Continuous direction at each echelon must be maintained, and one individual must be assigned both the authority and the responsibility at each level of the system's operation.

Training in the maintenance steps as well as in the system will be necessary. No system is a substitute for the actual technical ability required of the officers and enlisted personnel who direct and perform the upkeep of the equipment.

SOURCES OF INFORMATION

One of the most useful things you can learn about a subject is how to find out more about it. No single publication can give you all the information you need to perform the duties of your rating. You should learn where to look for accurate, authoritative, up-to-date information on all subjects related to the naval requirements for advancement and the occupational standards of your rating.

NAVSEA PUBLICATIONS

The publications issued by the Naval Sea Systems Command are of particular importance to engineering department personnel. Although you do not need to know everything in these publications, you should have a general idea of where to find the information contained therein.

Naval Ships' Technical Manual

The Naval Ships' Technical Manual is the basic engineering doctrine publication of the Naval Sea Systems Command. The manual is kept up-to-date by means of quarterly changes. As new chapters are issued they are being designated by a new chapter numbering system.

NAVSEA Deckplate

The NAVSEA *Deckplate* is a bimonthly technical periodical published by the Naval Sea Systems Command for the information of personnel in the naval establishment on the design, construction, conversion, operation, maintenance, and repair of naval vessels and their equipment, and on other technical equipment and on programs under the cognizance of the command. This magazine is particularly useful because it presents information that supplements and clarifies information contained in the *Naval Ships' Technical Manual*. It is also of considerable interest because it presents information on new developments in naval engineering. The NAVSEA *Deckplate* was formerly known as the NAVSEA Journal.

MANUFACTURERS' TECHNICAL MANUALS

The manufacturers' technical manuals furnished with most machinery units and many items of equipment are valuable sources of information on construction, operation, maintenance, and repair. The manufacturers' technical manuals that are furnished with most shipboard engineering equipment are given NAVSHIPS numbers.

DRAWINGS

Some of your work as a Machinery Repairman requires an ability to read and work from mechanical drawings. You will find information on how to read and interpret drawings in *Blueprint Reading and Sketching*, NAVEDTRA 10077 (series).

In addition to knowing how to read drawings, you must know how to locate applicable drawings. For some purposes, the drawings included in the manufacturers' technical manuals for the machinery or equipment may give you the information you need. In many cases, however, you will find it necessary to consult the on-board drawings. The on-board drawings, which are sometimes referred to as ship's plans or ship's blueprints, are listed in an index called the ship drawing index (SDI).

The SDI lists all working drawings that have a NAVSHIPS drawing number, all manufacturers' drawings designated as certification data sheets, equipment drawing lists, and assembly drawings that list detail drawings. The on-board drawings are identified in the SDI by an asterisk(*).

Drawings are listed in numerical order in the SDI. On-board drawings are filed according to numerical sequence. There are two types of numbering systems in use for drawings that have NAVSHIPS numbers. The older system is an S-group numbering system. The newer system, used on all NAVSHIPS drawings since 1 January 1967, is a consolidated index numbering system. A cross-reference list of S-group numbers and consolidated index numbers is given in *Ship Work Breakdown Structure*.

ENGINEERING HANDBOOKS

For certain types of information, you may need to consult various kinds of engineering handbooks—mechanical engineering handbooks, marine engineering handbooks, piping handbooks, machinery handbooks, and other handbooks that provide detailed, specialized technical data. Most engineering handbooks contain a great deal of technical information, much of it arranged in charts or tables. To make the best use of engineering handbooks, use the table of contents and the index to locate the information you need.

ADDENDUM

In addition to a comprehensive index that is printed in the back of this manual, you will find the following:

1. Appendix I contains 23 tables, such as decimal equivalents of fractions; division of the circumference of a circle: formulas for length, area, and volume; tapers, etc. You will find this information helpful in your everyday shop work.
2. Appendix II is formulas for spur gearing.
3. Appendix III shows the derivation of formulas for the diametral pitch system.
4. Appendix IV is a glossary of terms peculiar to the Machinery Repairman rating.

CHAPTER 2

TOOLROOMS AND TOOLS

Your proficiency as a Machinery Repairman is greatly influenced by your knowledge of tools and your skill in using them. The information that you will need to become familiar with the correct use and care of the many powered and nonpowered handtools, measuring instruments, and gages is available from various sources to which you will have access.

This rate training manual will provide information which applies to the tools and instruments used primarily by a Machinery Repairman. Additional information on tools that are commonly used by the many different naval ratings can be found in *Tools and Their Uses*, NAVPERS 10085-B.

TOOL ISSUE ROOM

One of your responsibilities as a Machinery Repairman is the operation of the tool crib or tool issuing room. You should ensure that the necessary tools are available and in good condition and that an adequate supply of consumable items (oil, wiping rags, bolts, nuts, and screws) are available.

Operating and maintaining a toolroom is simple if the correct procedures and methods are used to set up a system. Some of the basic considerations in operating a toolroom are (1) the issue and custody of tools; (2) replacement of broken, worn, or lost tools; and (3) proper storage and maintenance of tools.

ORGANIZATION OF THE TOOLROOM

Shipboard toolrooms are limited in size by the design characteristics of the ship. Therefore, the space set aside for this purpose must be used as efficiently as possible. Since the number of tools required aboard ship is extensive, toolrooms usually tend to be overcrowded. Certain peculiarities in shipboard toolrooms also require consideration. For example: The motion of the ship at sea requires that tools be made secure to prevent movement. The moisture content of the air requires that the tools be protected from corrosion.

Permanent bins, shelves, and drawers cannot easily be changed in the toolroom. However, existing storage spaces can be reorganized by dividing larger bins and relocating tools to provide better utilization of space.

Hammers, wrenches, and other tools that do not have cutting edges may normally be stored in bins. They also may be segregated by size or other designation. Tools with cutting edges require more space to prevent damage to the cutting edges. Usually these tools are stored on shelves lined with wood, on pegboards, or on hanging racks. Pegboards are especially adaptable for tools such as milling cutters. Some provision must be made to keep these tools from falling off the boards when the ship is rolling. Store precision tools (micrometers, dial indicators and so forth) in felt-lined wooden boxes in a cabinet to reduce the effects of vibration. This arrangement allows a quick daily inventory. It also prevents the instruments from being damaged by contact with other tools. Rotating bins can be used to store large supplies of small parts, such as nuts and bolts. Rotating bins provide rapid selection from a wide range of sizes. Figures 2-1, 2-2, and 2-3 show some of the common methods of tool storage.

28.333.1

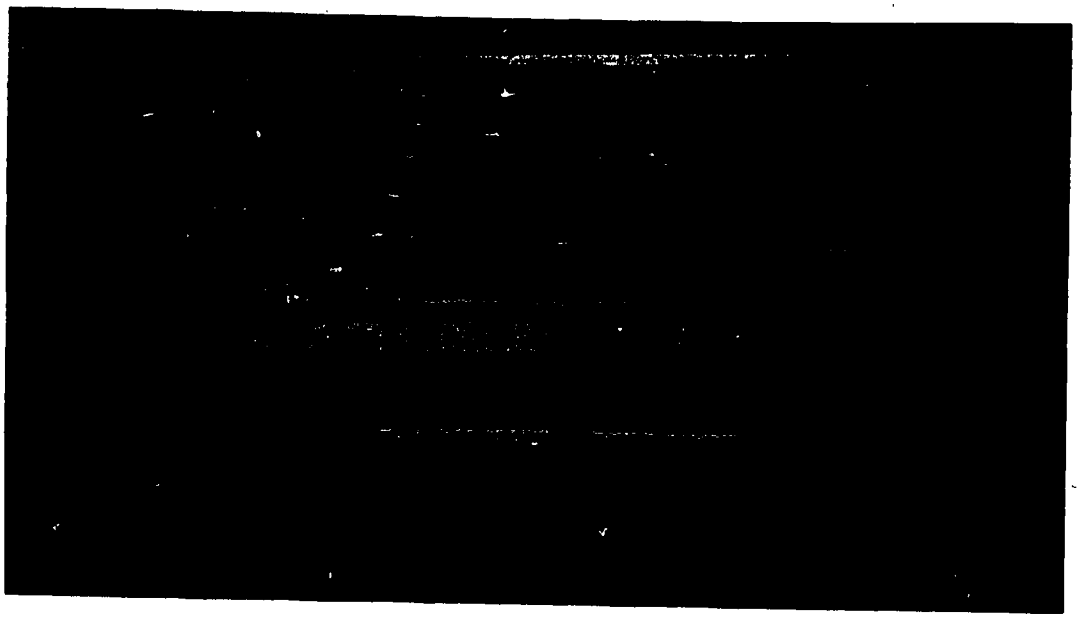
Figure 2-1.—Method of tool storage.

Frequently used tools should be located near the issuing door so that they are readily available. Seldom used tools should be placed in out of the way areas such as on top of bins or in spaces that cannot be used efficiently because of size and shape. Heavy tools should be placed in spaces or areas where a minimum of lifting is required. Portable power tools should be stored in racks. Provisions should be made for electrical extension cords and the cords of electric power tools.

All storage areas such as bins, drawers, and lockers should be clearly marked for ease in locating tools. These markings should be permanent; either stenciled with paint or on stamped metal tags.

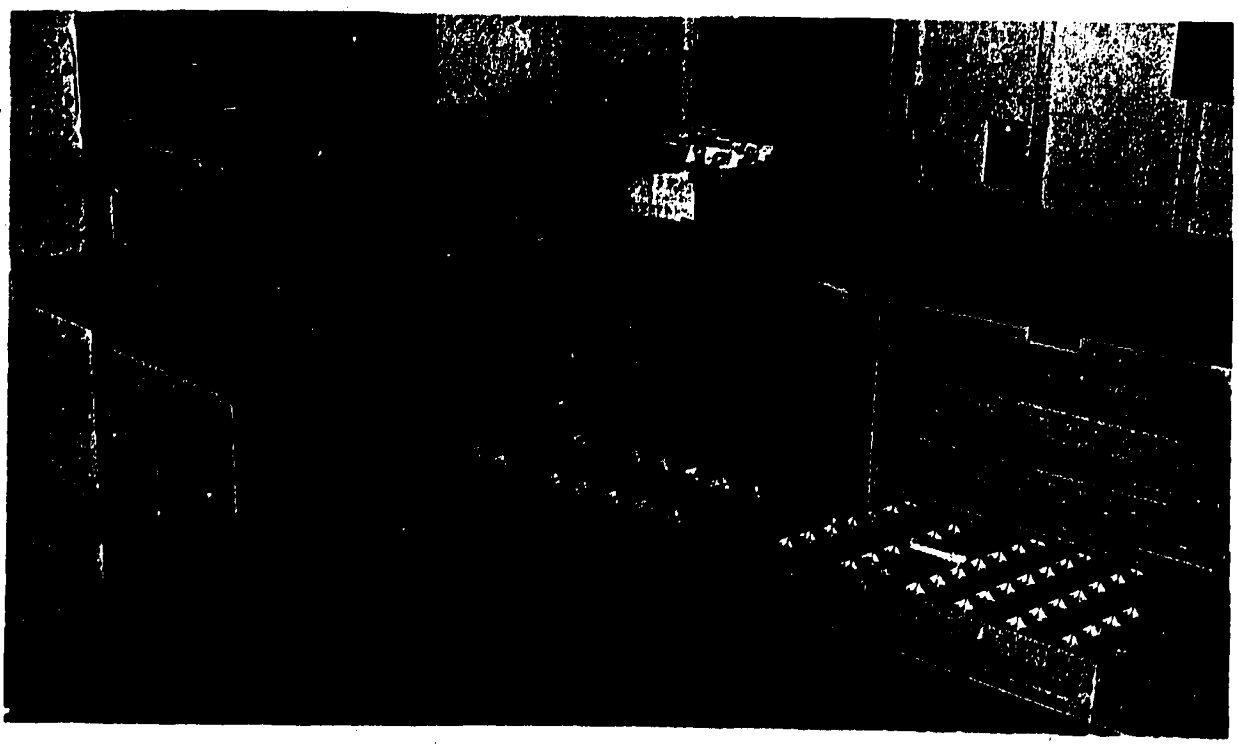
You will be responsible for the condition of all the tools and equipment in the toolroom. You should inspect all tools as they are returned to determine if they need repairs or adjustment. Set aside a space for damaged tools to prevent issue of these tools until they have been repaired.

You should wipe clean all returned tools and give a light coat of oil to the metal surfaces. Precision tools should be checked upon issue and return to determine if they are accurate.



28.334

Figure 2-2.—Method of tool storage.



28.335

Figure 2-3.—Method of tool storage.

Keep all spaces clean and free of dust to prevent foreign matter from getting into the working parts of tools.

Plan to spend a portion of each day in reconditioning damaged tools. This is important in keeping the tools available for issue and will prevent an accumulation of damaged tools.

CONTROL OF TOOLS

You will issue and receive tools and maintain control of custody of the tools. A method of identifying a borrower with the tool must be established, and provisions must be made for periodic inventory of available tools.

There are two common methods of tool issue control: the tool check system and the mimeographed form or tool chit system. Some toolrooms may use a combination of both of these systems. For example: Tool checks may be used for machine shop personnel, and mimeographed forms may be used for personnel outside the shop.

Tool checks are either metal or plastic disks stamped with numbers that identify the borrower. In this system the borrower presents a check for each tool, and the disk is placed on a peg near the space from which the tool was taken. The advantage of this system is that very little time is spent in completing the process.

If the tools are loaned to all departments in the ship, mimeographed forms generally are used. The form has a space for listing the tools, the borrower's name, the division or department, and the date. This system has the advantage of allowing anyone in the ship's crew to borrow tools and of keeping the toolroom keeper informed as to who has the tools, and how long they have been out.

You must know the location of tools and equipment out on loan, how long tools have been out, and the amount of equipment and consumable supplies you have on hand. To know this, you will have to make periodic inventories. The inventory consists of a count of all tools, by types, in the toolroom and those out on loan. In addition to control of custody, inventories will help you decide whether more strict control of equipment is required and whether you need to procure more tools and equipment for use.

Some selected items, called controlled equipage, will require an increased level of management and control due to their high cost, vulnerability to pilferage, or their importance to the ship's mission. The number of tools and instruments in this category under the control of a Machinery Repairman is generally small. However, it is important that you be aware of controlled equipage items. You can get detailed information about the designation of controlled equipage from the supply department of your activity. When these tools are received from the supply department, your department head will be required to sign a custody card, indicating a definite responsibility for management of the item. The department head will then require signed custody cards from personnel assigned to the division or shop where the item will be stored and used. As a toolroom keeper, you may be responsible for controlling the issue of these tools and ensuring their good condition. If these special tools are lost or broken beyond repair, replacement cannot be made until the correct survey procedures have been completed. Formal inventories of these items are conducted periodically as directed by your division officer or department head.

As a toolroom keeper, you may have additional duties as a supply representative for your department or division. Information on procurement of tools and supplies can be found in *Military Requirements for Petty Officer 3 & 2*, NAVEDTRA 10056-D.

SAFETY IN THE TOOLROOM AND THE SHOP

The toolroom, because of its relatively small size and the large quantity of different tools which are stored in it, can become very dangerous if all items are not kept stored in their proper places. A ship at sea can be hazardous especially in a crowded toolroom if the proper precautions are not followed for securing all drawers, bins, pegboards, and other storage facilities. Fire hazards are sometimes overlooked in the toolroom. When you consider the flammable liquids and wiping rags stored or issued from the toolroom, there is a real danger present.

As a toolroom keeper, you will play a very important part in ensuring that your shipmates are working in as safe an environment as possible. Several of your jobs are directly connected to the good working order and safe use of tools in the shop. If you were to issue an improperly ground twist drill to someone who did not have the experience to recognize the defect, the chances of the person being injured by the drill "digging in" or throwing the workpiece out of the drill press would be very real. A wrench which has been sprung or worn oversize can become a real "knucklebuster" to any unsuspecting user. An outside micrometer out of calibration can cause trouble if someone is trying to press fit two parts together using a hydraulic press. An electric-powered handtool that was properly inspected and tagged last week but has had the plug crushed since then can kill the user. The list of potential disasters that you as an individual have some influence in preventing is endless. The important thing to remember is that you as a toolroom keeper contribute more to the mission of the Navy than first meets the eye.

SHOP MEASURING GAGES

Practically all shop jobs require measuring or gaging. You will most likely measure or gage flat or round stock; the outside diameters of rods, shafts, or bolts; slots, grooves, and other openings; thread pitch and angle; spaces between surfaces; or angles and circles.

For some of these operations, you will have a choice of which instrument to use, but in other instances you will need a specific instrument. For example, when precision is not important, a simple rule or tape will be suitable, but in other instances, when precision is of prime importance, you will need a micrometer to obtain measurement of desired accuracy.

The term "gage," as used in this chapter, identifies any device which can be used to determine the size or shape of an object. There is no significant difference between gages and measuring instruments. They are both used to compare the size or shape of an object against a scale or fixed dimension. However, there is a distinction between measuring and gaging which is easily explained by an example. Suppose that you are turning work in a lathe and want to know the diameter of the work. Take a micrometer, or perhaps an outside caliper, adjust its opening to the exact diameter of the workpiece, and determine that dimension numerically. On the other hand, if you want to turn a piece of work down to a certain size without frequently taking time to measure it, set the caliper at a reading slightly greater than the final dimension desired; then, at intervals during turning operations, gage, or "size," the workpiece with the locked instrument. After the workpiece dimension has been reduced to the dimension set on the instrument, you will, of course, need to measure the work while finishing it to the exact dimension desired.

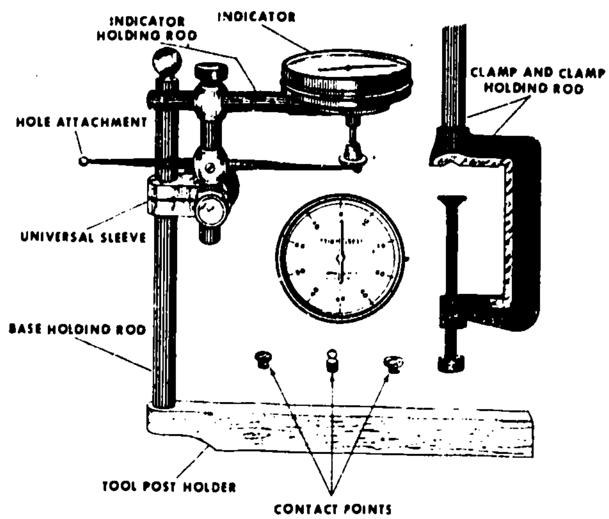
ADJUSTABLE GAGES

You can adjust adjustable gages by moving the scale or moving the gaging surface to the dimensions of the object being measured or gaged. For example, on the dial indicator, you can adjust the face to align the indicating hand with the zero point on the dial. On verniers, however, you would move the measuring surface to the dimensions of the object being measured.

Dial Indicators

Dial indicators are used by Machinery Repairmen in setting up work in machines and in checking the alignment of machinery. Proficiency in the use of the dial indicator will require a lot of practice, and you should use the indicator as often as possible to aid you in doing more accurate work.

Dial indicator sets (fig. 2-4) usually have several components that permit a wide variation of uses. The contact points allow use on different types of surfaces, the universal sleeve permits flexibility of setup, the clamp and holding rods permit setting the indicator to the work, the hole attachment indicates variation or run out of inside surfaces of holes, and the tool



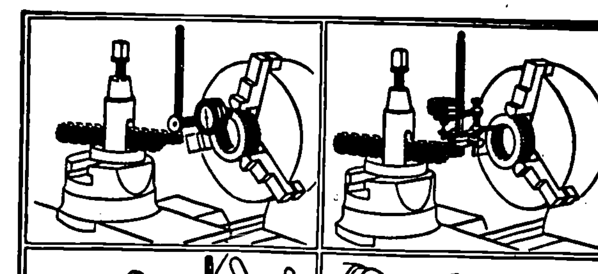
28.2X
Figure 2-4.—Universal dial indicator.

post holder can be used in lathe setups. Figure 2-5 shows some practical applications of dial indicators.

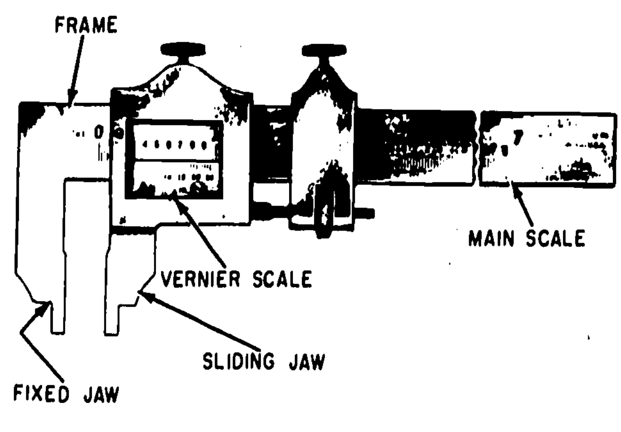
When you are preparing to use a dial indicator, there are several things that you should check. Dial indicators come in different degrees of accuracy. Some will give readings to one ten thousandths (0.0001) inch, while others will indicate to only five thousandths (0.005) inch. Dial indicators also differ in the total range or amount that they will indicate. If a dial indicator has a total of one hundred thousandths (0.100) inch in graduations on its face and has a total range of two hundred thousandths (0.200) inch, the needle will only make two revolutions before it begins to exceed its limit and jams up. The degree of accuracy and range of a dial indicator is usually shown on its face. Before you use a dial indicator, carefully depress the contact point and release it slowly; rotate the movable dial face so that the dial needle is on zero. Depress and release the contact point again and check to ensure that the dial pointer returns to zero; if it does not, have the dial indicator checked for accuracy.

Vernier Caliper

A vernier caliper (fig. 2-6) can be used to measure both inside and outside dimensions.



28.3X
Figure 2-5.—Applications of a dial indicator.

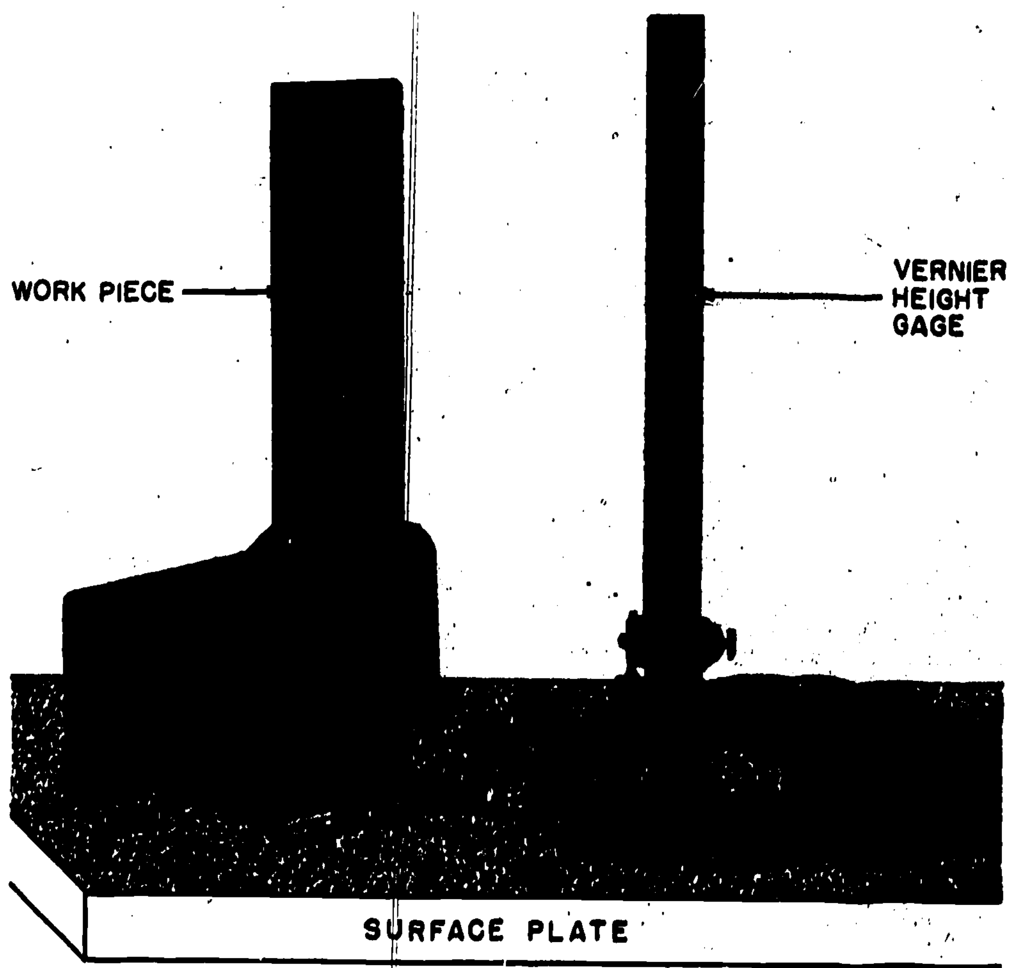


28.4B
Figure 2-6.—Vernier caliper.

Position the appropriate sides of the jaws on the surface to be measured and read the caliper from the side marked inside or outside as required. There is a difference in the zero marks on the two sides that is equal to the thickness of the tips of the two jaws, so be sure to read the correct side. Vernier calipers are available in sizes ranging from 6 inches to 6 feet and are graduated in increments of thousandths (0.001) of an inch. The scales on vernier calipers made by different manufacturers may vary slightly in length or number of divisions; however, they are all read basically the same way. Simplified instructions for interpreting the readings are covered in *Tools and Their Uses*, NAVEDTRA 10085-B.

Vernier Height Gage

A vernier height gage (fig. 2-7) is used to lay out work for machining operations or to check the dimensions on surfaces which have been machined. Attachments for the gage include the offset scriber shown attached to the gage in figure 2-7. The offset scriber lets you measure from the surface plate with readings taken directly from the scale without having to make any calculations. As you can see in figure 2-7, if



28.4(28D)X

Figure 2-7.—Vernier height gage.

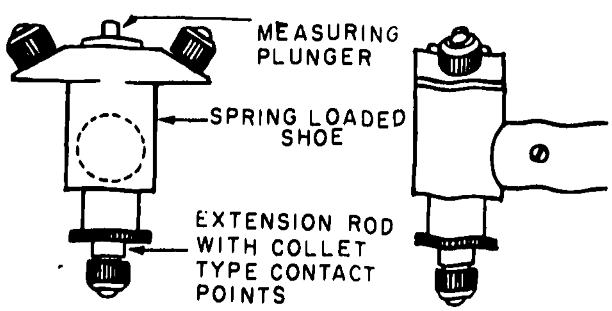
you were using a straight scriber, you would have to calculate the actual height by taking into account the distance between the surface plate and the zero mark. Some models have a slot in the base for the scriber to move down to the surface and a scale that permits direct reading. Another attachment is a rod that permits depth readings. Small dial indicators that connect to the scriber permit extremely close work in checking or laying out work. A vernier height gage is read the same way as a vernier caliper.

Dial Vernier Caliper

A dial vernier caliper looks much like a standard vernier caliper and is also graduated in one thousandths (0.001) of an inch. The main difference is that instead of a double scale, as on the vernier caliper, the dial vernier has the inches marked only along the main body of the caliper and a dial with two hands to indicate hundredths (0.100) and thousandths (0.001) of an inch. The range of the dial vernier caliper is usually 6 inches.

Dial Bore Gage

One of the most accurate tools for measuring a cylindrical bore or for checking a bore for out-of-roundness or taper is the dial bore gage. The dial bore gage (fig. 2-8) does not give a direct measurement; it gives you the amount of deviation from a preset size or the amount of deviation from one part of the bore to another. A master ring gage, outside micrometer, or vernier caliper can be used to preset the gage. A dial bore gage has two stationary spring-loaded points and an adjustable point to permit a variation in range. These three points are evenly spaced to allow accurate centering of the tool in the bore. A fourth point, the tip of the dial indicator, is located between the two stationary points. By simply rocking the tool in the bore, you can observe the amount of variation on the dial. Accuracy to one ten thousandth (0.0001) of an inch is possible with some models of the dial bore gage.



28.336

Figure 2-8.—Dial bore gage.

Internal Groove Gage

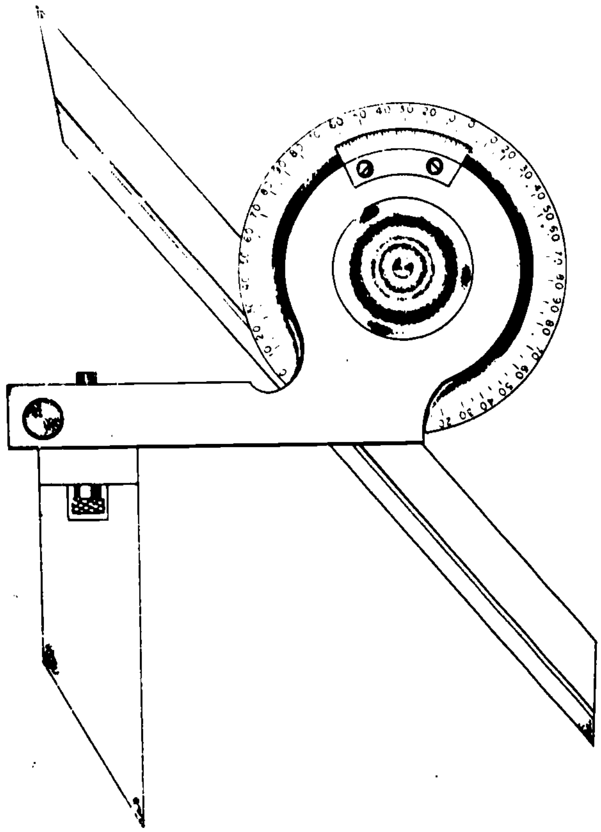
The internal groove gage is very useful for measuring the depth of an O-ring groove or other recesses inside a bore. This tool lets you measure a deeper recess and one located farther back in the bore than if you were to use an inside caliper. As with the dial bore gage, this tool must be set with gage blocks, a vernier caliper, or an outside micrometer. The reading taken from the dial indicator on the groove gage represents the difference between the desired recess or groove depth and the measured depth.

Universal Vernier Bevel Protractor

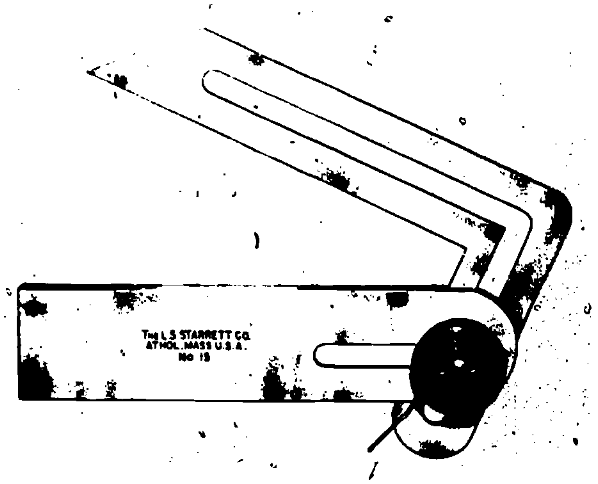
The universal vernier bevel protractor (fig. 2-9) is the tool you will use to lay out or measure angles on work to very close tolerances. The vernier scale on the tool permits measuring an angle to within 1/12° (5 minutes) and can be used completely through 360°. Interpreting the reading on the protractor is similar to the method used on the vernier caliper.

Universal Bevel

The universal bevel (fig. 2-10), because of the offset in the blade, is very useful for bevel gear work and for checking angles on lathe workpieces which cannot be reached with an ordinary bevel. The universal bevel must be set and checked with a protractor, or another suitable angle-measuring device, to obtain the desired angle.



28.337
Figure 2-9.–Universal vernier bevel protractor.



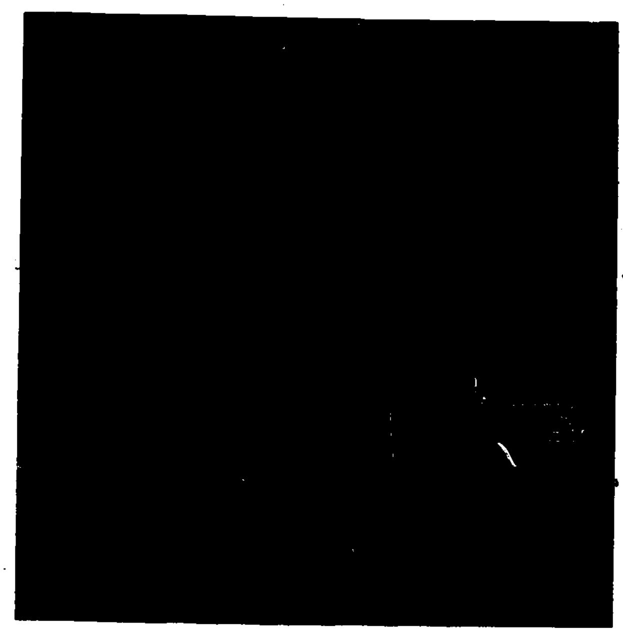
28.5X
Figure 2-10.–Universal bevel.

Gear Tooth Vernier

A gear tooth vernier (fig. 2-11) is used to measure the thickness of a gear tooth on the pitch circle and the distance from the top of the tooth to the pitch chord, at the same time. The vernier scale on this tool is read in the same way as other verniers, except that graduations on the main scale are 0.020 inch apart instead of 0.025 inch.

Cutter Clearance Gage

The cutter clearance gage (fig. 2-12) is one of the simplest to use, yet it is suitable for gaging clearance on all styles of plain milling cutters which have more than 8 teeth and a diameter range from 1/2 inch to 8 inches. To gage a tooth with this instrument, bring the surfaces of the "V" into contact with the cutter and lower the gage blade upon the tooth to be gaged. Rotate the cutter sufficiently to bring the tooth face into contact with the gage blade. If



28.4X
Figure 2-11.–Gear tooth vernier.

28.7X
Figure 2-12.—Cutter clearance gage.

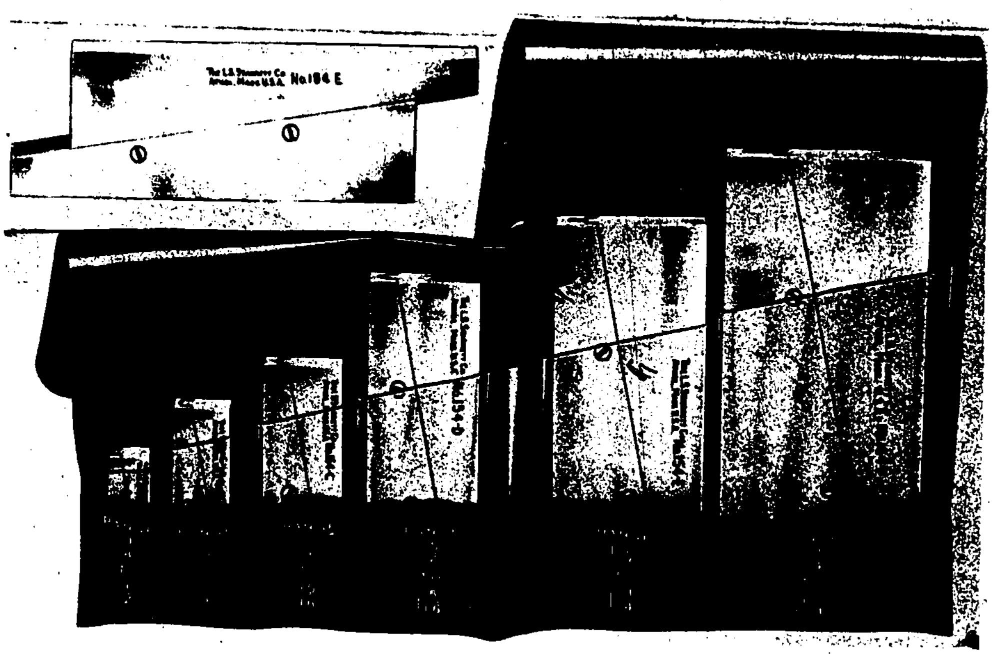
the angle of clearance on the tooth is correct, it will correspond with the angle of the gage blade. Cutter clearance gages that have an adjustable gage blade for checking clearance angles of 0°-30° on most common cutter styles are also available.

Adjustable Parallel

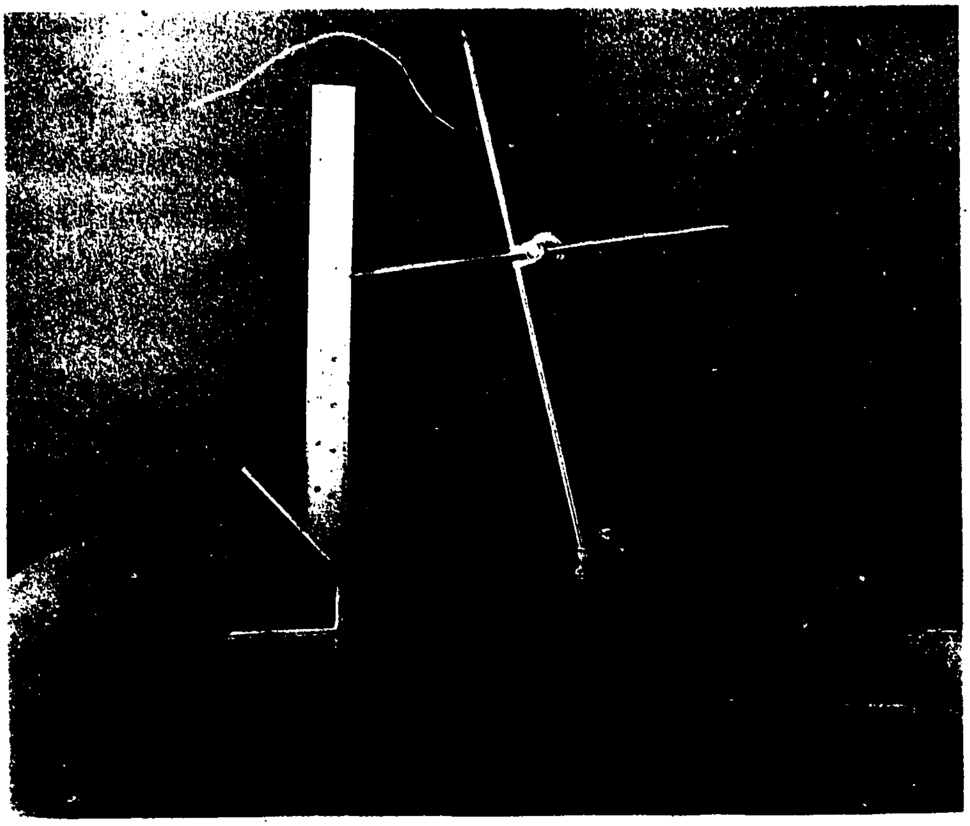
The adjustable parallel in figure 2-13 consists of two wedges connected on their inclined surfaces by a sliding dovetail. An adjustable parallel can be locked at any height between its maximum and minimum limits. This instrument, constructed to about the same accuracy of dimensions as parallel blocks, is very useful in leveling and positioning setups in a milling machine or in a shaper vise. An outside micrometer is usually used to set the adjustable parallel for height.

Surface Gage

A surface gage (fig. 2-14) is useful in gaging or measuring operations. It is used primarily in



28.6X
Figure 2-13.—Adjustable parallel.



28.9X

Figure 2-14.–Setting a dimension on a surface gage.

layout and alignment work. The surface gage is commonly used with a scriber to transfer dimensions and layout lines. In some cases a dial indicator is used with the surface gage to check trueness or alignment.

FIXED GAGES

Fixed gages cannot be adjusted. They can generally be divided into two categories, graduated and nongraduated. The accuracy of your work, when using fixed gages, will depend on your ability to determine the difference between the work and the gage. For example, a skilled machinist can take a dimension accurately to within 0.005 of an inch or less when using a common rule. Practical experience in the use of these gages will increase your ability to take accurate measurements.

Graduated Gages

Graduated gages are direct reading gages in that they have scales inscribed on them enabling you to take a reading while using the gage. The gages in this group are rules, scales, thread gages, and feeler gages.

RULES.–The steel rule with holder set (fig. 2-15A) is convenient for measuring recesses. It has a long tubular handle with a split chuck for holding the ruled blade. The chuck can be adjusted by a knurled nut at the top of the holder, allowing the rule to be set at various angles. The set has rules ranging from 1/4 to 1 inch in length.

The angle rule (fig. 2-15B) is useful in measuring small work mounted between centers on a lathe. The long side of the rule

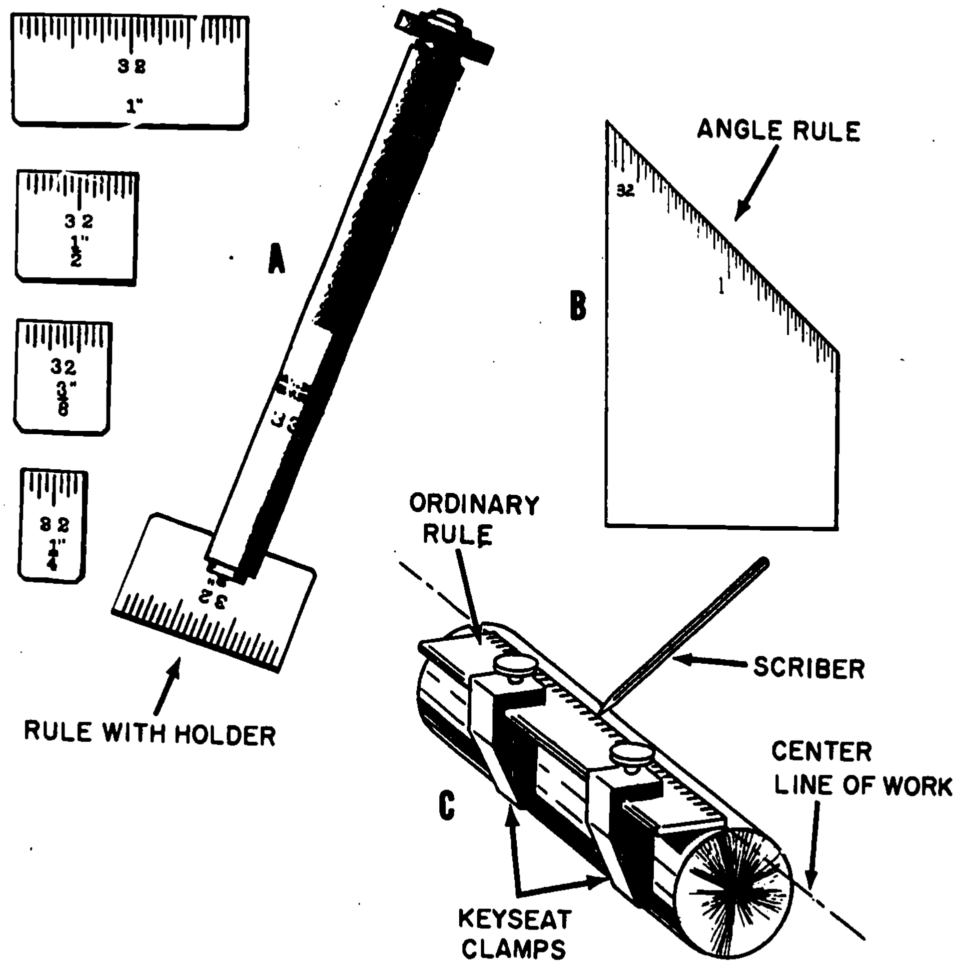


Figure 2-15.—Special rules for shop use.

28.10

(ungraduated) is placed even with one shoulder of the work. The graduated angle side of the rule can then be positioned easily over the work.

Another useful device is the keyseat rule (fig. 2-15C). It has a straightedge and a 6-inch machinist's-type rule arranged to form a right angle square. This rule and straightedge combination, when applied to the surface of a cylindrical workpiece, makes an excellent guide for drawing or scribing layout lines parallel to the axis of the work. You will find this device very convenient when making keyseat layouts on shafts.

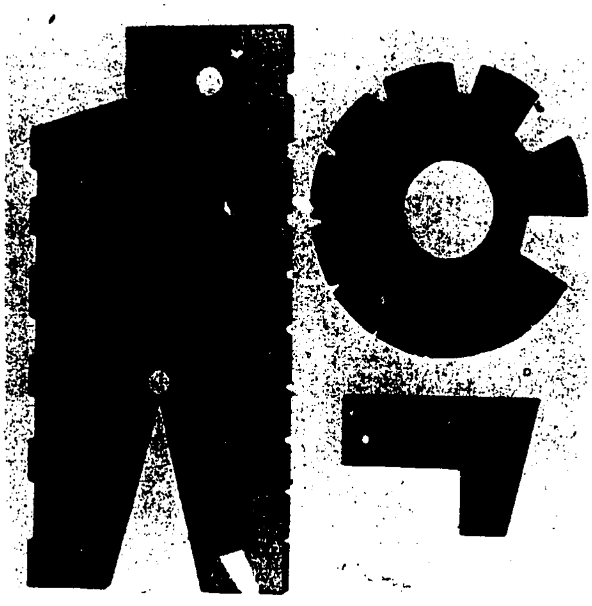
You must take care of your rules if you expect them to give accurate measurements. Do not allow them to become battered, covered with rust, or otherwise damaged in such a way that the markings cannot be read easily. Do not use them for scrapers, for once rules lose their sharp edges and square corners their general usefulness is decreased.

SCALES.—A scale is similar in appearance to a rule, since its surface is graduated into regular spaces. The graduations on a scale, however, differ from those on a rule because they are either larger or smaller than the measurements indicated. For example, a half-size scale is graduated so that 1 inch on the scale is equivalent to an actual measurement of 2 inches;

a 12-inch long scale of this type is equivalent to 24 inches. A scale, therefore, gives proportional measurements instead of the actual measurements obtained with a rule. Like rules, scales are made of wood, plastic, or metal, and they generally range from 6 to 24 inches.

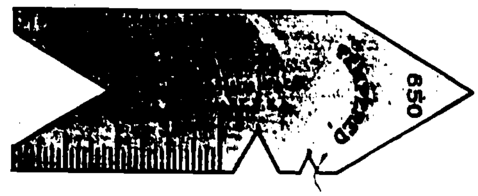
ACME THREAD TOOL GAGE.–This gage (fig. 2-16) is used to both grind the tool used to machine Acme threads and to set the tool up in the lathe. The sides of the Acme thread have an included angle of 29° (14 1/2° to each side), and this is the angle made into the gage. The width of the flat on the point of the tool varies according to the number of threads per inch. The gage provides different slots for you to use as a guide when grinding the tool. Setting the tool up in the lathe is simple. First, ensure that the tool is centered to the work as far as height is concerned. Then, with the gage edge laid parallel to the centerline of the work, adjust the side of your tool until it fits the angle on the gage very closely.

CENTER GAGE.–The center gage (fig. 2-17) is used like the Acme thread gage. Each



5.16.1X

Figure 2-16.–Acme thread gage

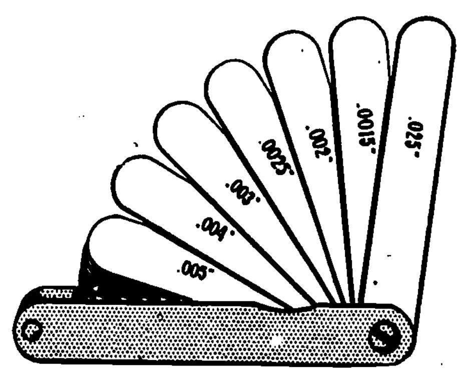


5.16.2X

Figure 2-17.–Center gage.

notch and the point of the gage has an included angle of 60°. The gage is used primarily to check and to set the angle of the V-sharp and other 60° standard threading tools. The center gage is also used to check lathe centers. The edges are graduated into 1/4, 1/24, 1/32, and 1/64 inch for ease in determining the pitch of threads on screws.

FEELER GAGE.–A feeler (thickness) gage, like the one shown in figure 2-18, is used to determine distances between two closely mating surfaces. This gage is made like a jackknife with blades of various thicknesses. When you use a combination of blades to get a desired gage thickness, try to place the thinner blades between the heavier ones to protect the thinner blades and to prevent their kinking. Do not force blades into openings which are too small; the blades may bend and kink. A good way to



4.19

Figure 2-18.–Feeler (thickness) gage.

get the "feel" of using a feeler gage correctly is to practice with the gage on openings of known dimensions.

RADIUS GAGE.–The radius gage (fig. 2-19) is often underrated in its usefulness to the machininst. Whenever possible, the design of most parts includes a radius located at the shoulder formed when a change is made in the diameter. This gives the part an added margin of strength at that particular place. When a square shoulder is machined in a place where a radius should have been, the possibility that the part will fail by bending or cracking is increased. The blades of most radius gages have both concave (inside curve) and convex (outside curve) radii in the common sizes.

Nongraduated Gages

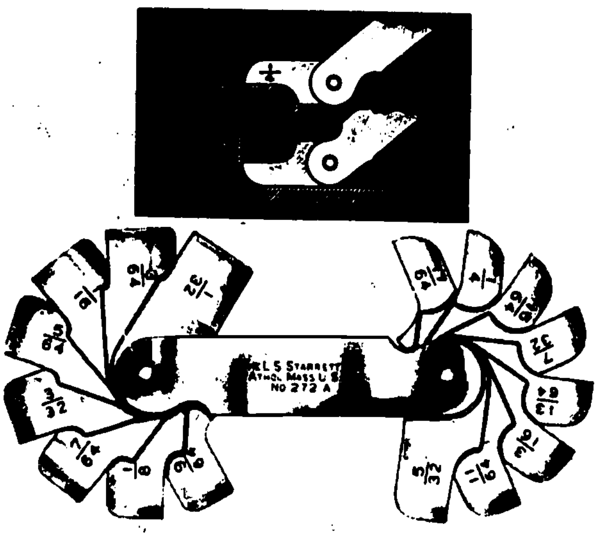
Nongraduated gages are used primarily as standards, or to determine the accuracy of form or shape.

STRAIGHTEDGES.–Straightedges look very much like rules, except that they are not graduated. They are used primarily for checking surfaces for straightness; however, they can also be used as guides for drawing or scribing straight lines. Two types of straightedges are shown in figure 2-20. Part A shows a straightedge made of steel which is hardened on the edges to prevent wear; it is the one you will probably use most often. The straightedge shown in Part B has a knife edge and is used for work requiring extreme accuracy.

Always keep a straightedge in a box when it is not in use. Some straightedges are marked with two arrows, one near each end, which indicate balance points. When a box is not provided, place resting pads on a flat surface in a storage area where no damage to the straightedge will occur from other tools. Then, place the straightedge so that the two balance points set on the resting pads.

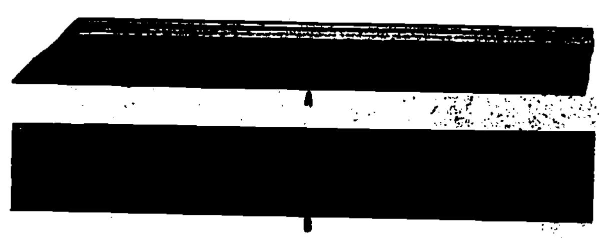
MACHINIST'S SQUARE.–The most common type of machinist's square is a hardened steel blade securely attached to a beam. The steel blade in NOT graduated. (See fig. 2-21.) This instrument is very useful in checking right angles and in setting up work on shapers, milling machines, and drilling machines. The size of machinist's squares ranges from 1 1/2 to 36 inches in blade length. The same care should be taken in storage, and use of as is taken with a micrometer.

SINE BAR.–A sine bar (fig. 2-22) is a precision tool used to establish angles which required extremely close accuracy. When used in



28.338

Figure 2-19.—Fillet or radius gages.



28.11X

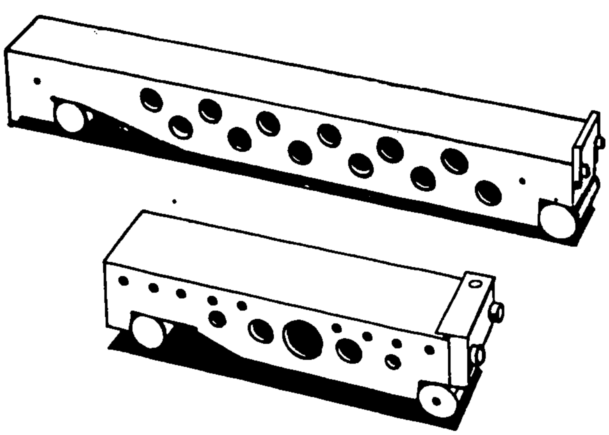
Figure 2-20.—Straightedges.

28.12X
Figure 2-21.—Machinist's square.

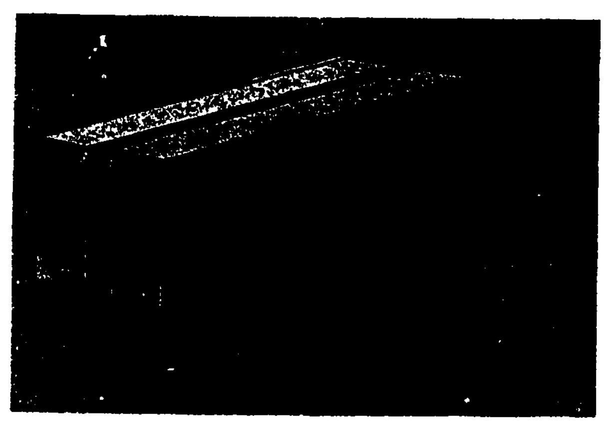
conjunction with a surface plate and gage blocks, angles are accurate to 1 minute (1/60°). The sine bar may be used to measure angles on work and to lay out an angle on work to be machined, or work may be mounted directly to the sine bar for machining. The cylindrical rolls and the parallel bar, which make up the sine bar, are all precision ground and accurately positioned to permit such close measurements. Any scratches, nicks, or other damage should be repaired before the sine bar is used, and care must be exercised in using and storing the sine bar. Instructions on using the sine bar are included in chapter 3.

PARALLEL BLOCKS.—Parallel blocks (fig. 2-23) are hardened, ground steel bars that are used in laying out work or setting up work for machining. The surfaces of the parallel block are all either parallel or perpendicular, as appropriate, and can be used to position work in a variety of setups with accuracy. They generally come in matched pairs and standard fractional dimensions. Care should be used in storing and handling to prevent damage. If it becomes necessary to regrind the parallel blocks, be sure to change the size stamped on the ends of the blocks.

GAGE BLOCKS.—Gage blocks are used as master gages to set and check other gages and instruments. Their accuracy is from eight millionths (0.000008) of an inch to two millionths (0.000002) of an inch, depending on the grade of the set. To visualize this minute amount, consider that the thickness of a human hair divided 1,500 times equals 0.000002 inch. This degree of accuracy applies to the thickness of the gage block, the parallelism of the sides, and the flatness of the surfaces. To attain this accuracy, a fine grade of hardenable alloy steel is ground and then lapped until the gage blocks are so smooth and flat that when they are "wrung" or placed one atop the other in the proper manner, you cannot separate them by pulling straight out. A set of gage blocks has enough different size blocks so that you can establish



28.339
Figure 2-22.—Sine bars.



28.13X
Figure 2-23.—Parallel blocks.

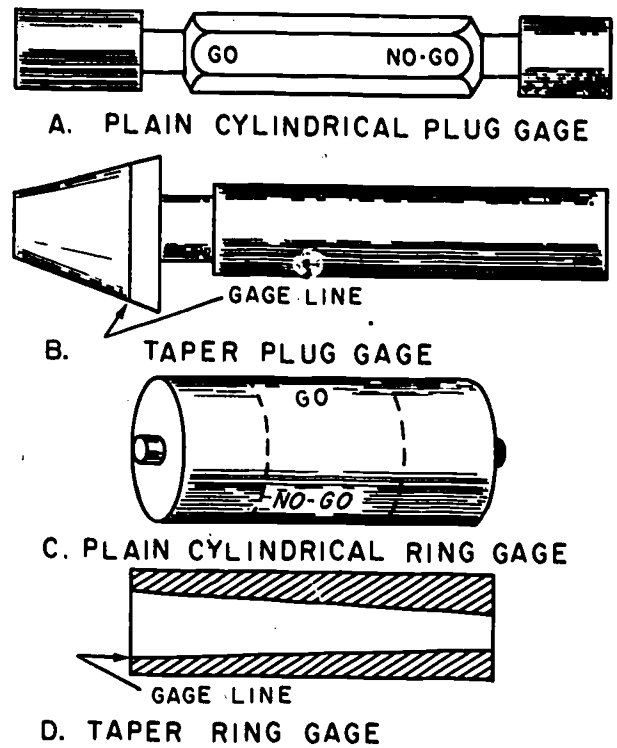
any measurement within the accuracy and range of the set. As you might expect, anything so accurate requires exceptional care to prevent damage and to ensure continued accuracy. A dust-free temperature-controlled atmosphere is preferred. After use, wipe each block clean of all marks and fingerprints and coat it with a thin layer of white petrolatum to prevent rust.

MICROMETER STANDARDS.—Micrometer standards are either disk or tubular shaped gages that are used to check outside micrometers for accuracy. Standards are made in sizes so that any size micrometer can be checked. They should be used on a micrometer on a regular basis to ensure continued accuracy. Additional information for the use of the standards are given later in this chapter.

RING AND PLUG GAGES.—A ring gage (fig. 2-24) is a cylindrical-shaped disk that has a precisely ground bore. Ring gages are used to check machined diameters by sliding the gage over the surface. Straight, tapered, and threaded diameters can be checked by using the appropriate gage. The ring gage is also used to set other measuring instruments to the basic dimension required for their operation. Normally, ring gages are available with a "GO" and a "NOT GO" size that represents the tolerance allowed for the particular size or job.

A plug gage (fig. 2-24) is used for the same types of jobs as a ring gage except that it is a solid shaft-shaped bar that has a precisely ground diameter for checking inside diameters or bores.

THREAD MEASURING WIRES.—The most accurate method of measuring the fit or pitch diameter of threads, without going into the expensive and sophisticated optical and comparator equipment, is thread measuring wires. These wires are accurately sized, depending on the number of threads per inch, so that when they are laid over the threads in a position that allows an outside micrometer to measure the distance between them, the pitch diameter of the threads can be determined. Sets are available that contain all the more common sizes. Detailed information on computing and using the wire method for measuring is covered in chapter 9.

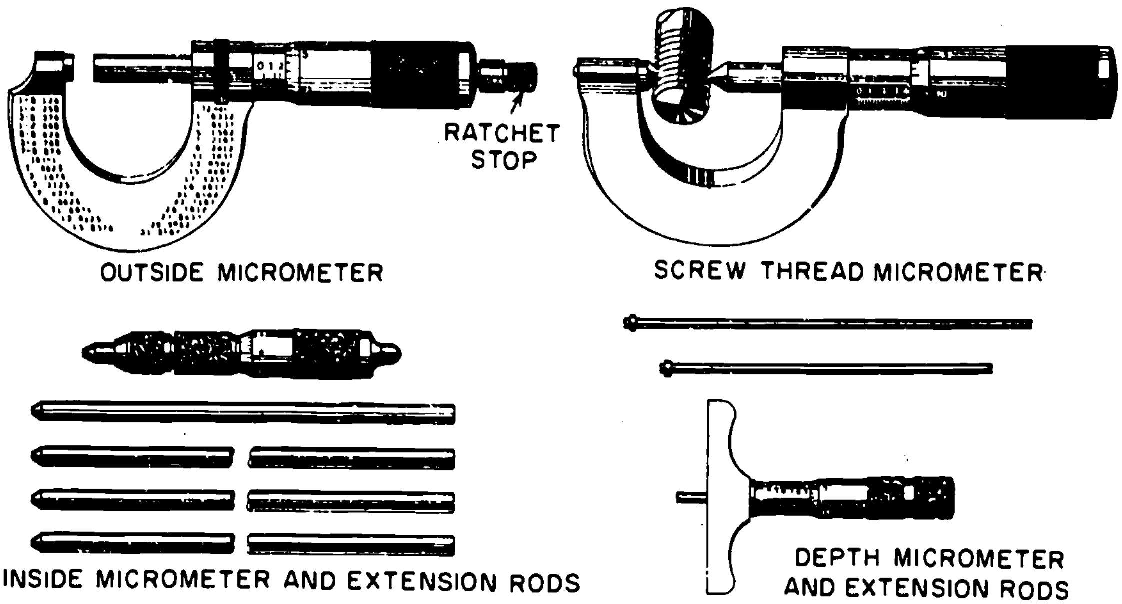


28.340

Figure 2-24.—Ring gage and plug gage.

MICROMETERS

Micrometers are probably the most often used precision measuring instruments in a machine shop. There are many different types, each having been designed to permit measurement of surfaces for various applications and configurations of workpieces. The degree of accuracy obtainable from a micrometer also varies, with the most common graduations being from one thousandth (0.001) inch to one ten thousandths (0.0001) inch. Information on the correct procedure for interpreting the reading on micrometers can be found in *Tools and Their Uses*, NAVEDTRA 10085-B. A brief description of the more common types of micrometers is provided in the following paragraphs.



4.20.1B

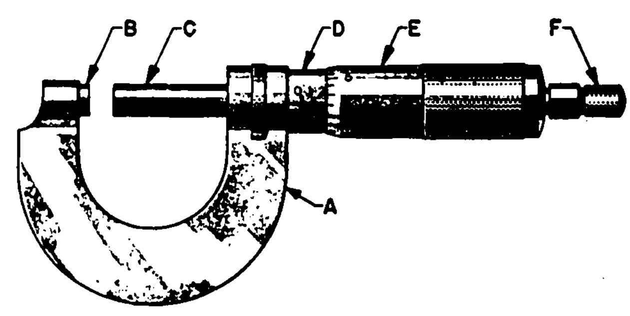
Figure 2-25.—Common types of micrometers.

Outside Micrometer

An outside micrometer (fig. 2-25 and 2-26) often called a micrometer caliper, or mike, is used to measure the thickness or the outside diameter of parts. They are available in sizes ranging from 1 inch to about 96 inches in steps of 1 inch. The larger sizes normally come as a set with interchangeable anvils which provide a range of several inches. The anvils have an adjusting nut and a locking nut to permit setting the micrometer with a micrometer standard. Regardless of the degree of accuracy designed into the micrometer, the skill applied by each individual is the primary factor in determining accuracy and reliability in measurements. Training and practice will result in a proficiency in using this tool that will benefit you greatly.

Inside Micrometer

An insi[illegible] micrometer (fig. 2-25) is used to [illegible] diameters or between parallel [illegible]; are available in sizes ranging from [illegible] to about 107 inches. The individual interchangeable extension rods that are assembled to the micrometer head vary in size by 1 inch. A small sleeve or bushing, which is 0.500 inch long, is used with these rods in most inside micrometer sets to provide the complete range of sizes. Using the inside micrometer is slightly more difficult than using the outside



A. Frame
B. Anvil
C. Spindle
D. Sleeve
E. Thimble
F. Ratchet stop

4.20.1A

Figure 2-26.—Nomenclature of an outside micrometer caliper.

micrometer, primarily because there is more chance of your not getting the same "feel" or measurement each time you check the same surface.

The correct way to measure an inside diameter is to hold the micrometer in place with one hand as you "feel" for the maximum possible setting of the micrometer by rocking the extension rod from left to right and in and out of the hole. Adjust the micrometer to a slightly larger measurement after each series of rocking movements until no rocking from left to right is possible and a very slight drag is felt on the in and out rocking movement. There are no specific guidelines on the number of positions within a hole that should be measured. If you are checking for taper, then you should take measurements as far apart as possible within the hole. If you are checking for roundness or concentricity of a hole, then you should take several measurements at different angular positions in the same area of the hole. You may take the reading directly from the inside micrometer head, or you may use an outside micrometer to measure the inside micrometer.

Depth Micrometer

A depth micrometer (fig. 2-25) is used to measure the depth of holes, slots, counterbores, recesses, and the distance from a surface to some recessed part. This type of micrometer is read exactly opposite from the method used to read an outside micrometer. The zero is located toward the closed end of the thimble. The measurement is read in reverse and increases in amount (depth) as the thimble moves toward the base of the instrument. The extension rods come either round or flat (blade-like) to permit measuring a narrow, deep recess or groove.

Thread Micrometer

The thread micrometer (fig. 2-25) is used to measure the depth of threads that have an included angle of 60°. The measurement obtained represents the pitch diameter of the thread. They are available in sizes that measure pitch diameters up to 2 inches. Each micrometer has a given range of number of threads per inch that can be measured correctly. Additional information on using this micrometer can be found in chapter 9.

Miscellaneous Micrometers

The machine tool industry has been very responsive to the needs of the machinist by designing and manufacturing measuring instruments for practically every imaginable application. If you find that you are devising measuring techniques for a particularly odd application with the resulting measurements being of questionable value and that you do it on a routine basis, maybe a special micrometer will make your work easier and more reliable. Some of the special micrometers that you may have a need for are described in later discussions.

BALL MICROMETER.—This type micrometer has a rounded anvil and a flat spindle. It can be used to check the wall thickness of cylinders, sleeves, rings, and other parts that have a hole bored in a piece of material. The rounded anvil is placed inside the hole and the spindle is brought into contact with the outside diameter. Ball attachments that fit over the anvil of regular outside micrometers are also available. When using the attachments, you must compensate for the diameter of the ball as you read the micrometer.

BLADE MICROMETER.—A blade micrometer has an anvil and a spindle that are thin and flat. The spindle does not rotate. This micrometer is especially useful in measuring the depth of narrow grooves such as an O-ring seat on outside diameters.

GROOVE MICROMETERS.—A groove micrometer looks like an inside micrometer with two flat disks. The distance between the disks increases as you turn the micrometer. It is used to measure the width of grooves or recesses on either the outside or the inside diameter. The width of an internal O-ring groove is an excellent example.

CARE AND MAINTENANCE OF GAGES

The proper care and maintenance of precision instruments is very important to a conscientious Machinery Repairman. To help you maintain your instruments in the most accurate and reliable condition possible, the Navy has established a calibration program that provides calibration technicians, the required standards and procedures, and a schedule of how often an instrument must be calibrated to be reliable. When an instrument is calibrated, a sticker is affixed to the instrument showing the date the calibration was done and the date the next calibration is due. Whenever possible, you should use the Navy calibration program to verify the accuracy of your instruments. Some repair jobs, due to their sensitive nature, demand the reliability provided by the program. Information concerning the procedures that you can use in the shop to check the accuracy of an instrument is contained in the upcoming paragraphs.

Micrometers

The micrometer is one of the most used, and often one of the most abused, precision measuring instruments in the shop. Careful observation of the do's and don'ts listed below will enable you to take proper care of the micrometer you use.

1. Always stop the work before taking a measurement. Do NOT measure moving parts because the micrometer may get caught in the rotating work and be severely damaged.

2. Always open a micrometer by holding the frame with one hand and turning the knurled sleeve with the other hand. Never open a micrometer by twirling the frame, because such practice will put unnecessary strain on the instrument and cause excessive wear of the threads.

3. Apply only moderate force to the knurled thimble when taking a measurement. Always use the friction slip ratchet if there is one on the instrument. Too much pressure on the knurled sleeve will not only result in an inaccurate reading, but may also cause the frame to spring, forcing the measuring surfaces out of line.

4. When a micrometer is not in actual use, place it where it is not likely to be dropped. Dropping a micrometer can cause the frame to spring; if dropped, the instrument should be checked for accuracy before any further readings are taken.

5. Before a micrometer is returned to stowage, back the spindle away from the anvil, wipe all exterior surfaces with a clean, soft cloth, and coat the surfaces with a light oil. Do not reset the measuring surfaces to close contact because the protecting film of oil in these surfaces will be squeezed out.

MAINTENANCE OF MICROMETERS.–A micrometer caliper should be checked for zero setting (and adjusted when necessary) as a matter of routine to ensure that reliable readings are being obtained. To do this, proceed as follows:

1. Wipe the measuring faces, making sure that they are perfectly clean, and then bring the spindle into contact with the anvil. Use the same moderate force that you ordinarily use when taking a measurement. The reading should be zero; if it is not, the micrometer needs further checking.

2. If the reading is more than zero, examine the edges of the measuring faces for burrs. Should burrs be present, remove them with a small slip of oilstone; clean the measuring surfaces again, and then recheck the micrometer for zero setting.

3. If the reading is less than zero, or if a zero reading is not obtained after making the correction described above, you will need to adjust the spindle-thimble relation. The method for setting zero differs considerably between makes of micrometers. Some makes have a thimble cap which locks the thimble to the spindle; some have a special rotatable sleeve on the barrel that can be unlocked; and some have an adjustable anvil.

Methods For Setting Zero.–To adjust the THIMBLE-CAP TYPE, back the spindle away from the anvil, release the thimble cap with the small spanner wrench provided for that purpose, and bring the spindle into contact with the anvil. Hold the spindle firmly with one hand and rotate the thimble to zero with the other; after zero relation has been established, rotate the spindle counterclockwise to open the micrometer, and then tighten the thimble cap. After tightening the cap, check the zero setting again to be sure that the thimble-spindle relation was not disturbed while the cap was being tightened.

To adjust the ROTATABLE SLEEVE TYPE, unlock the barrel sleeve with the small spanner wrench provided for that purpose, bring the spindle into contact with the anvil, and rotate the sleeve into alignment with the zero mark on the thimble. After alignment, back the spindle away from the anvil, and retighten the barrel sleeve locking nut. Recheck for zero setting, to be sure that you did not disturb the thimble-sleeve relation while tightening the lock nut.

To set zero on the ADJUSTABLE ANVIL TYPE, bring the thimble to zero reading, lock the spindle if a spindle lock is provided, and loosen the anvil lock screw. After the lock screw has been loosened, bring the anvil into contact with the spindle, making sure that the thimble is still set on zero. Tighten the anvil setscrew lock nut slightly, unlock the spindle, and back the spindle away from the anvil; then lock the anvil setscrew firmly. After locking the setscrew, check the micrometer for zero setting to make sure that the anvil was not moved out of position while the setscrew was being tightened.

The zero check and methods of adjustment of course apply directly to micrometers that will measure to zero; the PROCEDURE FOR LARGER MICROMETERS is essentially the same except that a standard must be placed between the anvil and the spindle in order to get a zero measuring reference. For example, a 2-inch micrometer is furnished with a 1-inch standard. To check for zero setting, place the standard between the spindle and the anvil and measure the standard. If zero is not indicated, the micrometer needs adjusting.

Testing For And Correcting Errors By Use Of Standards.—A micrometer must be tested from time to time for uneven wear of measuring threads and for concave wear of the measuring faces because these defects are not detectable by zero-setting checks. The test for uneven internal wear can be made by measuring a flat-surfaced standard; the test for concavity of measuring faces, by measuring a cylindrical disk-shaped standard.

The procedure for making these tests and correcting the defects which are found is as follows: First, check the micrometer for zero setting and adjust as necessary. Then take measurements of several different size gage blocks or other accurate standards. If the micrometer readings do not agree with the gage dimensions, wear in the internal parts of the micrometer is indicated, and the micrometer must be adjusted. Adjustment is made with the thread wear compensating nut, located inside the thimble assembly. After the gage block test has been completed, measure several cylindrical standards of different sizes. Discrepancies between micrometer readings and the marked (actual) sizes of the standards indicate that the measuring surfaces are concave. This condition can be corrected by lapping the measuring faces on a true flat surface. After lapping the faces of the micrometer, reset the instrument for zero reading and measure the cylindrical standards again.

Inside Micrometers.—These instruments can be checked for zero setting adjusted in about the same way as a micrometer caliper; the main difference in the method of testing is that an accurate micrometer caliper is required for transferring readings to and from the standard when an inside micrometer is being checked.

Micrometers of all types should be disassembled periodically for cleaning and lubrication of internal parts. When this is done, each part should be cleaned in noncorrosive solvent, completely dried, and then given a lubricating coat of watchmaker's oil or a similar light oil.

Vernier Gages

Vernier gages also require careful handling and proper maintenance if they are to remain accurate. The following instructions apply to vernier gages in general:

1. Always loosen the binding screws before attempting to move sliding arms.

2. Never force a gage into position. Forcing, besides causing an inaccurate reading, is likely to force the arms out of alignment.

3. When taking a measurement, use only gentle pressure on the fine adjustment screw. Heavy pressure will force the two scales out of parallel.

4. Prior to putting a vernier gage away, wipe it clean and give it a light coating of oil. (Perspiration from hands will cause the instrument to corrode rapidly.)

Dials

Dial indicators and other instruments that have a mechanically operated dial as part of their measurement features are easily damaged by misuse and lack of proper maintenance. The following instructions apply to dials in general:

1. As previously mentioned, be sure that the dial you have selected to use has the range capability required. When a dial is extended beyond its design limit, some lever, small gear or rack must give to the pressure. The dial will be rendered useless if this happens.

2. Never leave a dial in contact with any surface that is being subjected to a shock (such as hammering a part when dialing it in) or an erratic and uncontrolled movement that could cause the dial to be overtraveled.

3. Protect the dial when it is not being used. Provide a storage area where the dial will not receive accidental blows and where dust, oil, and chips will not contact it.

4. When a dial becomes sticky or sluggish in operating, it may be either damaged or dirty. You may find that the pointer is rubbing the dial crystal or that it is bent and rubbing the dial face. A sluggish dial should never be oiled. Oil will compound the problems. A suitable cleaning solvent should be used to remove all dirt and residue.

CHAPTER 3

LAYOUT AND BENCHWORK

As an MR 3 or MR 2 you will repair or assist in repairing a great many types of equipment used on ships. In addition to making replacement parts, you will disassemble and assemble equipment, make layouts of parts to be machined, and do precision work in fitting mating parts of equipment. This is known as benchwork and includes practically all repair work other than actual machining.

This chapter contains information that you should know to enable you to make effective repairs to equipment. A brief discussion on blueprints and mechanical drawings is included because in many repair jobs you must rely heavily on information acquired from these sources. Other sources on information that you should study for details on specific equipment include the *NAVSHIPS' Technical Manual*, manufacturers' technical manuals and Rate Training Manuals that have information related to the equipment on which you are working.

MECHANICAL DRAWINGS AND BLUEPRINTS

A mechanical drawing, made with special instruments and tools, gives a true representation of an object to be made, including its shape, size, description, specifications as to material to be used, and method of manufacture. A blueprint is an exact duplicate of a mechanical drawing. For reference purposes, every ship is furnished blueprint copies of all important mechanical drawings used in the construction of its hull and machinery. These blueprints are usually stowed in an indexed file in the log room, damage control office, technical library, or other central location, where they will be readily available for reference.

The following paragraphs cover briefly some important points in connection with working from sketches and blueprints. They do not contain definitions of all drafting terms, or information regarding the mechanics of blueprint reading, both of which are covered in detail in the Rate Training Manual, *Blueprint Reading and Sketching*, NAVPERS 10077-E.

Of the many types of blueprints you will use aboard ship, the simplest is the PLAN VIEW. This blueprint shows the position, location, and use of the various parts of the ship. You will use plan views to find your duty and battle stations, the sickbay, the barber shop, and other parts of the ship.

In addition to plan views, you will find aboard ship other blueprints called assembly prints, unit or subassembly prints, and detail prints. These prints show various kinds of machinery and mechanical equipment.

ASSEMBLY PRINTS show the various parts of the mechanism and how the parts fit together. Individual mechanisms, such as motors and pumps, will be shown on SUBASSEMBLY PRINTS. These show location, shape, size, and relationships of the parts of the subassembly unit. Assembly and subassembly prints are used to learn operation and maintenance of machines and equipment.

Machinery Repairmen are most interested in DETAIL PRINTS; these will give you the information required to make a new part. They show size, shape, kind of material, and method of finishing. You will find them indispensable in your work.

WORKING FROM DRAWINGS

Detail prints usually show only the individual part that you must produce. They show two or more orthographic views of the object, and in special cases they may show an isometric projection without dimension lines. An isometric projection shows how the part will look when made.

Each drawing or blueprint has a number in the title box in the lower right-hand corner of the print. The title box also shows the part name, the scale used, the pattern number, the material required, the assembly or subassembly print number to which the part belongs, and the names or initials of the persons who drew, checked, and approved the drawings. (See fig. 3-1.)

Accurate and satisfactory fabrication of a part described on a drawing depends upon how well the MR does the following:

- Correctly reads the drawing and closely observes all data thereon.
- Selects the correct material.
- Selects the correct tools and instruments for laying out the job.
- Uses the baseline or reference line method of locating the dimensional points during layout, thereby avoiding cumulative errors (described later in this chapter).
- Strictly observes tolerances and allowances.
- Accurately gauges and measures the work throughout the fabricating process.
- Gives due consideration, when measuring, for expansion of the workpiece by heat generated by the cutting operations. This is especially important in checking dimensions during finishing operations, if work is being machined to close tolerance.

COMMON BLUEPRINT SYMBOLS

In learning to read machine drawings you must first become familiar with the common terms, symbols and conventions (general practice) that are normally used. The information in figures 3-2, 3-3, and 3-4 will provide the basic data that you will need to interpret blueprints. As you study each of the various lines and symbols refer back to figure 3-1 to see how each is used in blueprints.

Surface Texture

Control over the finished dimensions of a part is no longer the only factor you must consider when deciding how you will do a job. The degree of smoothness, or surface roughness, has become very important in the efficiency and life of a machine part.

A finished surface may appear to be perfectly flat; however, upon close examination with surface finish measuring instruments, the surface is found to be formed of irregular waves. On top of the waves are other smaller waves which we shall refer to as peaks and valleys. These peaks and valleys are used to determine the surface roughness measurements of height and width. The larger waves are measured to give the waviness height and width measurements. Figure 3-5 illustrates the general location of the various areas for surface finish measurements and the relation of the symbols to the surface characteristics.

Surface roughness is the measurement of the finely spaced surface irregularities, the height, width, direction, and shape of which establishes the predominant surface pattern. These irregularities are caused by the cutting or abrading action of the machine tools that have been used to obtain the surface. One method of measuring the irregularities is by using special measuring instruments equipped with a tracer arm. The tracer arm has either a diamond or a sapphire contact point with a 0.0005-inch radius. The tracer arm travels across the surface

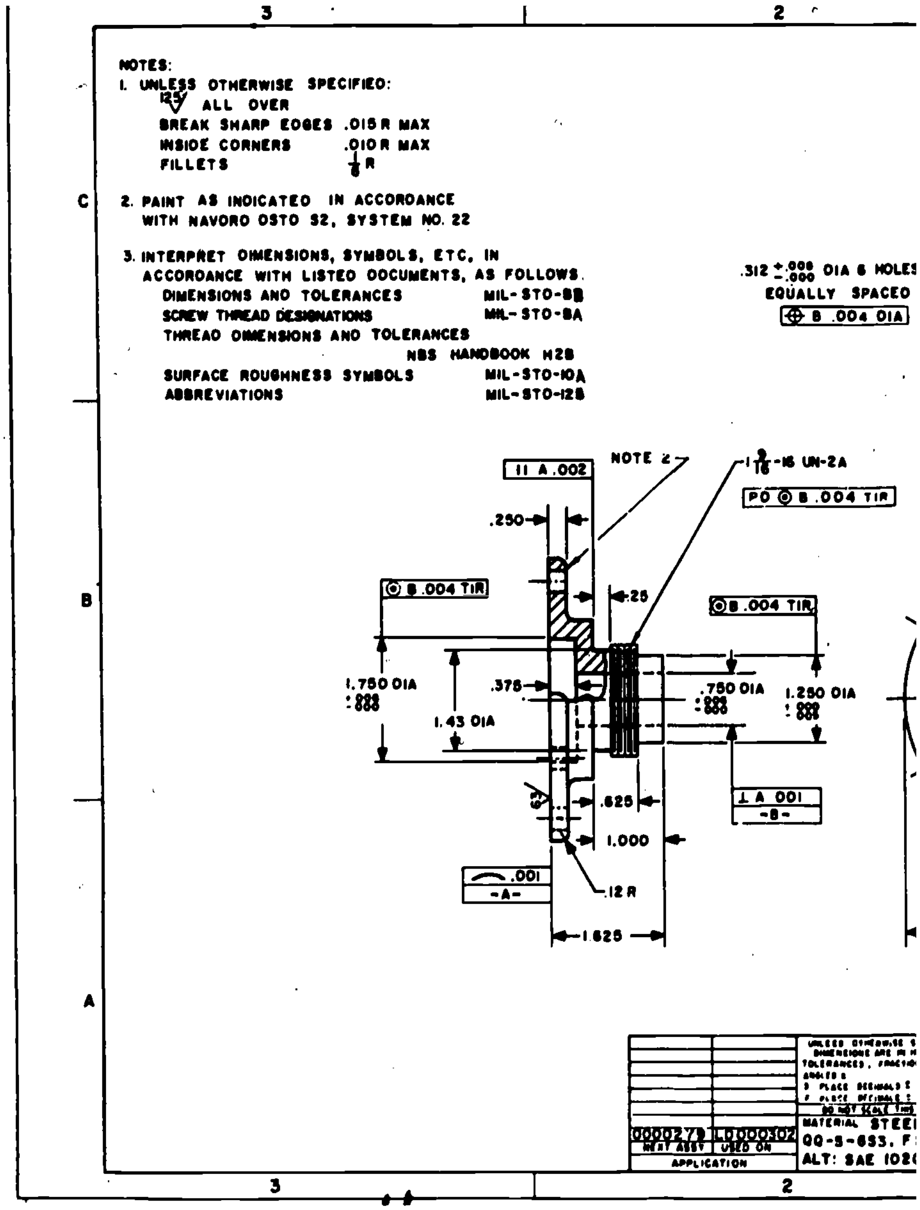


Figure 3-1.—Detail drawing of ba

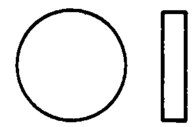
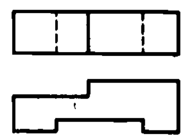
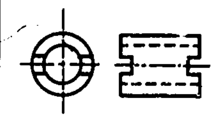
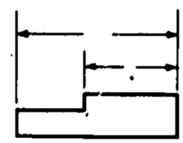
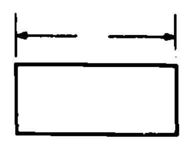
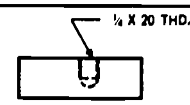
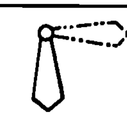
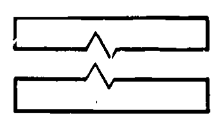
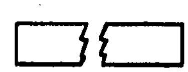
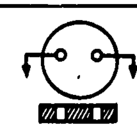
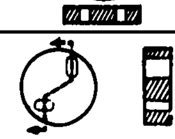
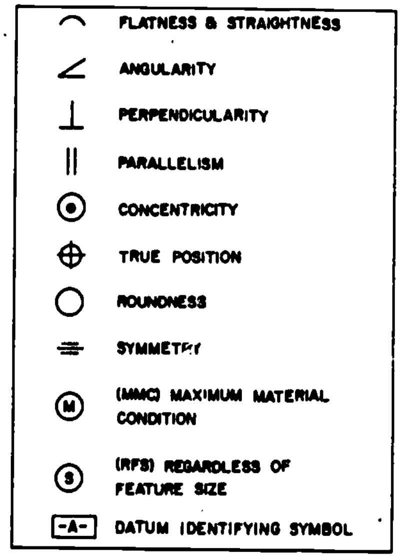
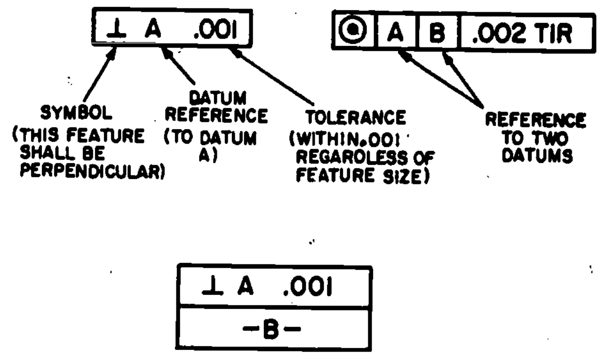
LINE STANDARDS			
NAME	CONVENTION	DESCRIPTION AND APPLICATION	EXAMPLE
VISIBLE LINES		HEAVY UNBROKEN LINES USED TO INDICATE VISIBLE EDGES OF AN OBJECT	
HIDDEN LINES		MEDIUM LINES WITH SHORT EVENLY SPACED DASHES USED TO INDICATE CONCEALED EDGES	
CENTER LINES		THIN LINES MADE UP OF LONG AND SHORT DASHES ALTERNATELY SPACED AND CONSISTENT IN LENGTH USED TO INDICATE SYMMETRY ABOUT AN AXIS AND LOCATION OF CENTERS	
DIMENSION LINES		THIN LINES TERMINATED WITH ARROW HEADS AT EACH END USED TO INDICATE DISTANCE MEASURED	
EXTENSION LINES		THIN UNBROKEN LINES USED TO INDICATE EXTENT OF DIMENSIONS	
LEADER		THIN LINE TERMINATED WITH ARROWHEAD OR DOT AT ONE END USED TO INDICATE A PART, DIMENSION OR OTHER REFERENCE	
PHANTOM OR DATUM LINE		MEDIUM SERIES OF ONE LONG DASH AND TWO SHORT DASHES EVENLY SPACED ENDING WITH LONG DASH USED TO INDICATE ALTERNATE POSITION OF PARTS, REPEATED DETAIL OR TO INDICATE A DATUM PLANE	
BREAK (LONG)		THIN SOLID RULED LINES WITH FREEHAND ZIG-ZAGS USED TO REDUCE SIZE OF DRAWING REQUIRED TO DELINEATE OBJECT AND REDUCE DETAIL	
BREAK (SHORT)		THICK SOLID FREE HAND LINES USED TO INDICATE A SHORT BREAK	
CUTTING OR VIEWING PLANE / VIEWING PLANE OPTIONAL		THICK SOLID LINES WITH ARROWHEAD TO INDICATE DIRECTION IN WHICH SECTION OR PLANE IS VIEWED OR TAKEN	
CUTTING PLANE FOR COMPLEX OR OFFSET VIEWS		THICK SHORT DASHES USED TO SHOW OFFSET WITH ARROWHEADS TO SHOW DIRECTION VIEWED	

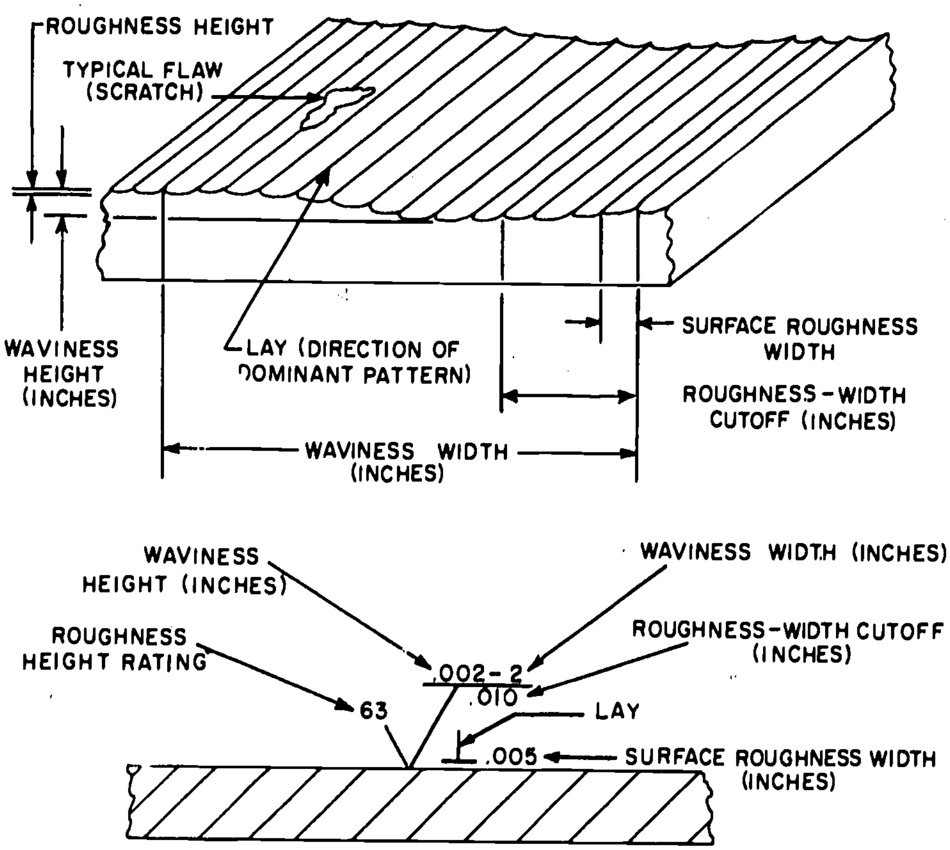
Figure 3-2.—Line characteristics and conventions for MIL-STD drawings. 142.46



65.82A
Figure 3-3.—Geometric characteristic sybmols.

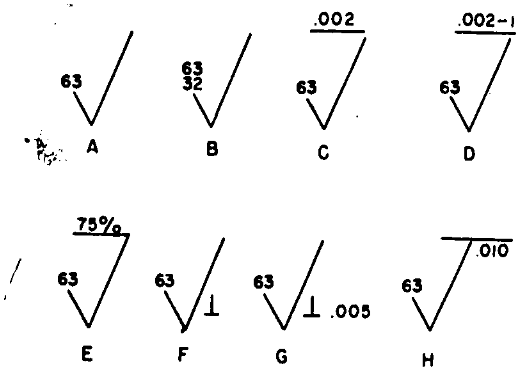


65.82B
Figure 3-4.—Feature control symbol incorporating datum reference.



126.43
Figure 3-5.—Relation of symbols to surface characteristics.

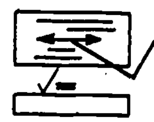
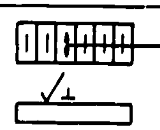
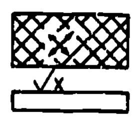
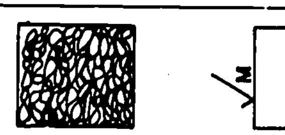
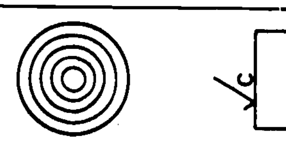
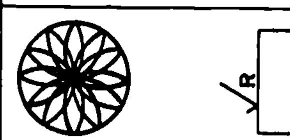
45



126.44
Figure 3-6.—Symbols used to indicate surface roughness, waviness, and lay.

and the contact point moves up and down the peaks and valleys. The movement of the contact point is amplified electrically and recorded graphically on a graduated tape. From this tape the various measurements are determined.

The basic roughness symbol is a check mark. This symbol is supplemented with a horizontal extension line above it when requirements such as waviness height, waviness width, or contact area must be specified in the symbol. A drawing that shows only the basic symbol indicates that the surface finish requirements are detailed in the Notes block. The roughness height rating is placed at the top of the short leg of the check (part A, fig. 3-6). If only one number is shown for roughness height, it is the maximum permissible roughness height rating; if two are

LAY SYMBOL	DESIGNATION	EXAMPLE
=	LAY PARALLEL TO THE BOUNDARY LINE REPRESENTING THE SURFACE TO WHICH THE SYMBOL APPLIES.	 DIRECTION OF TOOL MARKS
⊥	LAY PERPENDICULAR TO THE BOUNDARY LINE REPRESENTING THE SURFACE TO WHICH THE SYMBOL APPLIES.	 DIRECTION OF TOOL MARKS
X	LAY ANGULAR IN BOTH DIRECTIONS TO BOUNDARY LINE REPRESENTING THE SURFACE TO WHICH SYMBOL APPLIES.	 DIRECTION OF TOOL MARKS
M	LAY MULTIDIRECTIONAL	
C	LAY APPROXIMATELY CIRCULAR RELATIVE TO THE CENTER OF THE SURFACE TO WHICH THE SYMBOL APPLIES.	
R	LAY APPROXIMATELY RADIAL RELATIVE TO THE CENTER OF THE SURFACE TO WHICH THE SYMBOL APPLIES.	

126.45
Figure 3-7.—Symbols indicating the direction of lay.

shown, the top number is the maximum (part B, fig. 3-6). A point to remember is that the smaller the number in the roughness height rating, the smoother the surface.

Waviness height values are shown directly above the extension line at the top of the long leg of the basic check (part C, fig. 3-6). Waviness width values are placed just to the right of the waviness height values (part D, fig. 3-6). Where minimum requirements for contact or bearing surfaces must be shown, the percentage is placed at the top of the long leg of the basic check (part E, fig. 3-6). Any further surface finish requirements that would have been shown in that location, such as waviness width or height, will be shown in the Notes block of the drawing.

Lay is the direction of the predominant surface pattern produced by the tool marks. The symbol indicating lay is placed to the right and slightly above the point of the surface roughness symbol as shown in part F of figure 3-6. (Figure 3-7 shows the six symbols that indicate the direction of lay.)

The roughness width value is shown just to the right and parallel to the lay symbol. The roughness width cutoff is placed immediately below the extension line and to the right of the long leg of the basic check mark. These symbols for roughness width are shown in parts G and H of figure 3-6.

In the past, an alpha-numeric symbol was used to indicate the degree of smoothness required on a part. This system was not very effective because no specific or measurable value was assigned to each classification of finish. A fine tool finish can mean different things to different people. Some of the more common symbols that may be found on older blueprints are shown in table 3-1.

Table 3-1.—Former Finish Designations

Preferred Symbols	Meaning	Alternate Symbols			
F_1	Rough Tool Finish	V_1	Fr.	FIN.	TF.
F_2	Fine Tool Finish	V_2	F.	Fs.	SF.
F_3	Grind Finish	V_3	Fg.	Gr.	
F_4	Polish	V_4	Bf.	Buff	
F_5	Drill	V_5	Dr.		
F_6	Ream	V_6	Rm.		
F_7	File Finish	V_7	ff.	Ff.	
F_8	Scrape	V_8	scr.		
F_9	Spot Face	V_9			
Finish All Over			F.A.O.		f.a.o.

68.187

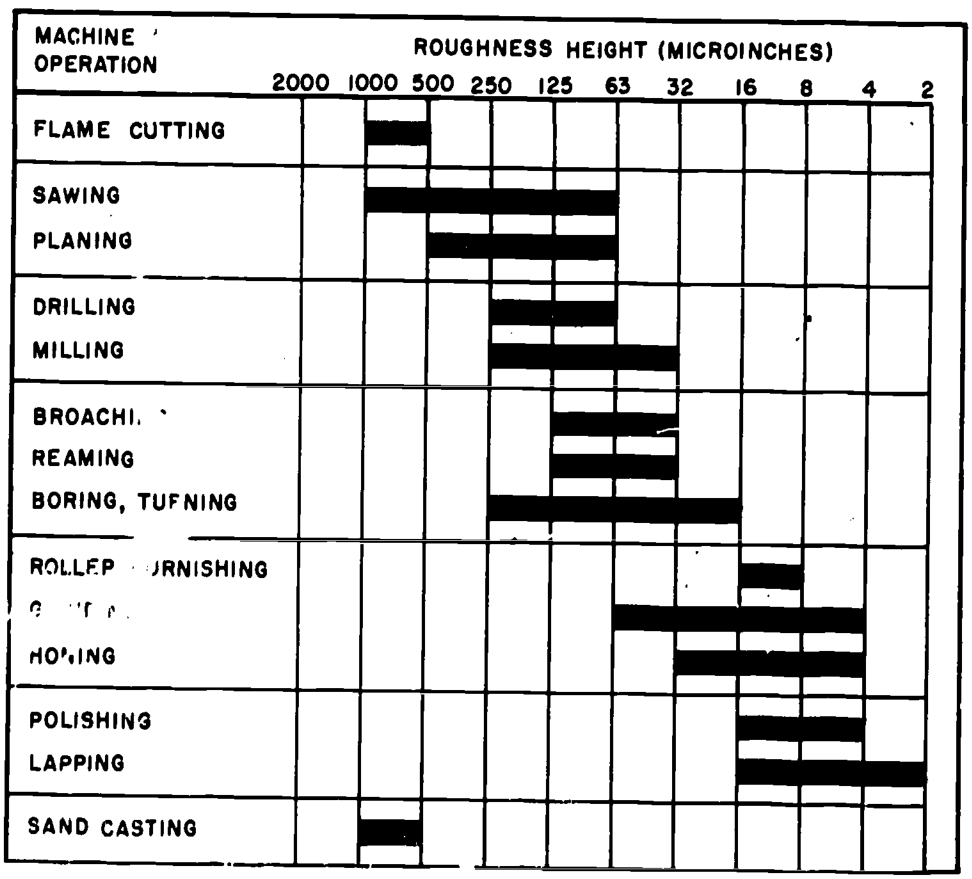
Your shop may not have the delicate and expensive instruments used to measure the irregularities of a surface although some of the larger and more fully equipped repair facilities will have them. There are roughness comparison specimens available today that will serve all but the most critical applications. These can be small plastic or metal samples, representing various roughness heights in several lay patterns. Figure 3-8 gives a sampling of some roughness height values that can be obtained by the different machine operations that you will encounter. Use it as an estimating tool only, as it has the same shortcomings as the "F" values in table 3-1.

UNITS OF MEASUREMENT

Accuracy is the trademark of the Machinery Repairman, and it is to your advantage to always strive for the greatest amount of accuracy. You can work many hours on a project and if it is not accurate, you will oftentimes have to start over. With this thought in mind, study carefully the following information about both the English and the metric systems of measurement.

English System

The inch is the basic (or smallest whole) unit of measurement in the English system. Parts of the inch must be expressed as either common fractions or decimal fractions. Examples of common fractions are 1/2, 1/4, 1/8, 1/16, 1/32, and 1/64. Decimal fractions can be expressed with a numerator and denominator (1/10, 1/100, 1/1000, etc.), but in most machine shop work and on blueprints or drawings they are expressed in decimal form such as 0.1, 0.01, and



28.322

Figure 3-8.—Roughness height values for machine operations.

0.001. Decimal fractions are expressed in the following manner:

One-tenth inch	= 0.1 in.
One-hundredth inch	= 0.01 in.
One-thousandth inch	= 0.001 in.
One ten-thousandth inch	= 0.0001 in.

You will occasionally need to convert a common fraction to a decimal. This is easily done by dividing the denominator of the fraction into the numerator. As an example, the decimal equivalent of the fraction 1/16 inch is: 1 ÷ 16 = 0.625 inch. A chart giving the decimal equivalents of the most common fractions is shown in Appendix I.

Metric System

The metric system is used by many countries including the United States, and as a Machinery Repairman you should have an understanding of this system of measurement. The basic unit of linear measurement for the metric system is the meter.

In the metric system the meter can be subdivide , into the following parts:

10 decimeters (dm)
or
100 centimeters (cm)
or
1000 millimeters (mm)

Therefore, 1 decimeter is 1/10 of a meter, 1 centimeter is 1/100 meter, and 1 millimeter is 1/1000 meter. The metric unit of measurement most often used in the machinist trade is the millimeter (mm).

If you understand the relationship of the two systems, you can convert easily from one system to the other. For example, 1 meter is equal to 39.37 inches; 1 inch is equal to 2.54 centimeters (or 25.4 millimeters). To convert from the English system to the metric system, multiply the number of inches by 2.54 (for centimeters) or 25.40 (for millimeters). As an example: 1.375 inches converted to centimeters is 1.375 inch X 2.540 = 3.4925 cm. Further, 0.0008 inch converted to millimeters is 0.0008 inch X 25.40 = 0.0203 mm.

To convert from the metric system to the English system, divide the metric units of measure by either 2.54 (for centimeters) or 25.4 (for millimeters). As an example: 0.215 mm converted to inches is 0.215 mm ÷ 25.4 = 0.0084 inch.

LIMITS OF ACCURACY

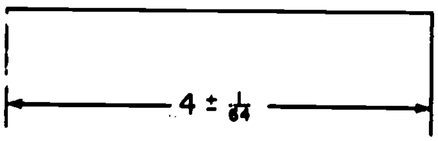
You must work within the limits of accuracy specified on the drawing. A clear understanding of TOLERANCE and ALLOWANCE will help you to avoid making small, but potentially dangerous errors. These terms may seem closely related but each has a very precise meaning and application. In the following paragraphs we will point out the meanings of these terms and the importance of observing the distinctions between them.

Tolerance

Working to the absolute or exact basic dimension is impractical and unnecessary in most instances; therefore, the designer calculates, in addition to the basic dimensions, an allowable variation. The amount of variation, or limit of error permissible is indicated on the drawing as plus or minus (±) a given amount, such as ±0.005; ±1/64. The difference between the allowable minimum and the allowance maximum dimension is tolerance. For example, in figure 3-9:

Basic dimension	= 4
Long limit	= 4 1/64
Short limit	= 3 63/64
Tolerance	= 1/32

When tolerances are not actually specified on a drawing, fairly concrete assumptions can be made concerning the accuracy expectod, by



28.14

Figure 3-9.—Basic dimension and tolerance.

using the following principles. For dimensions that end in a fraction of an inch, such as 1/8, 1/16, 1/32, 1/64, consider the expected accuracy to be to the nearest 1/64 inch. When the dimension is given in decimal form, the following applies:

If a dimension is given as 3.000 inches, the accuracy expected is ±0.0005 inch; or if the dimension is given as 3.00 inches, the accuracy expected is ±0.005 inch. The ±0.0005 is called, in shop terms, "plus or minus five ten-thousandths of an inch." The ±0.005 is called "plus or minus five thousandths of an inch."

Allowance

Allowance is an intentional difference in dimensions of mating parts to provide the desired fit. A CLEARANCE ALLOWANCE permits movement between mating parts when assembled. For example, when a hole with a 0.250-inch diameter is fitted with a shaft that has a 0.245-inch diameter, the clearance allowance is 0.005 inch. An INTERFERENCE ALLOWANCE is the opposite of a clearance allowance. The difference in dimensions in this case provides a tight fit. Force is required when assembling parts that have an interference allowance. If a shaft with a 0.251-inch diameter is fitted into the hole identified in the preceding example, the difference between the dimensions will give an interference allowance of 0.001 inch. As the shaft is larger than the hole, force is necessary to assemble the parts.

What is the relationship between tolerance and allowance? In the manufacture of mating parts, the tolerance of each part must be controlled so that the parts will have the proper allowance when assembled. For example, if a hole 0.250 inch in diameter with a tolerance of 0.005 inch (±0.0025) is prescribed for a job, and a shaft to be fitted in the hole is to have a clearance allowance of 0.001 inch, the hole must first be finished within the limits and the required size of the shaft determined exactly, before the shaft can be made. If the hole is finished to the upper limit of the basic dimension (0.2525 inch), the shaft would be machined to 0.2515 inch or 0.001 inch smaller than the hole. If the dimension of the shaft was given with the same tolerance as the hole, there would be no control over the allowance between the parts. As much as 0.005-inch allowance (either clearance or interference) could result.

To provide a method of retaining the required allowance while permitting some tolerance in the dimensions of the mating parts, the tolerance is limited to one direction on each part. This single direction (unilateral) tolerance stems from the basic hole system. If a clearance allowance is required between mating parts, the hole may be larger but not smaller than the basic dimension; the part that fits into the opening may be smaller, but not larger than the basic dimension. Thus, shafts and other parts that fit into a mating opening have a minus tolerance only, while the openings have a plus tolerance only. If an interference allowance between the mating parts is required, the situation is reversed; the opening can be smaller but not larger than the basic dimension, while the shaft can be larger but not smaller than the basic dimension. Therefore you can expect to see a tolerance such as +.005, -0, or +0, -.005, but with the required value not necessarily .005. One way to get a better understanding of a clearance allowance, or in interference allowance, is to make a rough sketch of the piece and add dimensions to the sketch where they apply.

LAYOUT

Layout is the term that describes the marking of metal surfaces to provide an outline for machining. A layout is comparable to a single view (end, top, or side) of a part which is sketched directly on the workpiece. Any difficulty in making layouts depends on the intricacies of the part to be laid out and the number of operations required to make the part. A flange layout, for example, is relatively simple as the entire layout can be made on one surface of the blank flange. However, an intricate casting may require layout lines on more than one surface. This requires careful study and concentration to ensure that the layout will have the same relationships as those shown on the drawing (or sample) that you are using.

When a part must be laid out on two or more surfaces, you may need to lay out one or two surfaces and machine them to size before using further layout lines. This prevents removal of layout lines on one surface while you are machining another. In other words, it would be useless to lay out the top surface of a part and machine off the layout lines while cutting the part to the layout lines of an end surface.

Through the process of computing and transferring dimensions, you will become familiar with the relationship of the surfaces. Understanding this relationship will be of benefit in planning the sequence of machining operations.

You should be able to hold the dimensions of a layout to within a tolerance of 1/64 inch. Sometimes you must work to a tolerance of even less than that.

A layout of a part is made when the directional movement or location of the part is controlled by hand or aligned visually without the use of precision instruments (such as when work is done on bandsaws or drill presses). In cutting irregular shapes on shapers, planers, lathes, or milling machines, layout lines are made, and the tool or work is guided by hand. In making a part with hand cutting tools, layout is essential.

Mechanical drawing and layout are closely related subjects; knowledge of one will help you to understand the other. A knowledge of general mathematics, trigonometry, and geometry, as well as the selection and use of the required tools is necessary in doing jobs related to layout and mechanical drawing. Study *Mathematics, Volume I*, NAVPERS 10069-C; *Mathematics, Volume II*, NAVPERS 10071-B; *Tools and Their Uses*, NAVPERS 10085-B, and *Blueprint Reading and Sketching*, NAVEDTRA 10077-E, for additional information.

MATERIALS AND EQUIPMENT

A scribed line on the surface of metal is usually hard to see; therefore, a layout liquid is used to provide a contrasting background. Commercially prepared layout dyes or inks are available through the Navy supply system. Chalk can be used, but it does not stick to a finished surface as well as layout dye. The layout dyes, commonly used, color the metal surface with a blue or copper tint. A line scribed on this colored surface reveals the color of the metal through the background.

The tools generally used for making layout lines are the combination square set, machinist's square, surface gage, scribe, straightedge, rule, divider, and caliper. Tools and equipment used in setting up the part to be laid out are surface plates, parallel blocks, angle plates, V-blocks, and sine bar. Surface plates have very accurately scraped flat surfaces. They provide a mounting table for the work to be laid out so that all lines in the layout can be made to one reference surface. Angle plates are used to mount the work at an angle to the surface plate. Angle plates are commonly used when the lines in the layout are at an angle to the reference surface. These plates may be fixed or adjustable; fixed angle plates are more accurate because one surface is machined to a specific angle in relation to the base. Adjustable angle plates are convenient to use because the angular mounting surface can be adjusted to meet the requirements of the job. V-blocks are used for mounting round stock on the surface plate. Parallel blocks are placed under the work to locate the work at a convenient height.

The sine bar is a precision tool used for determining angles which require accuracy within 5 minutes of arc. The sine bar may be used to check angles or establish angles for layout and inspection work. The sine bar must be used in conjunction with a surface plate and gage blocks if accuracy is to be maintained. Use of the sine bar will be covered later in this chapter.

Toolmaker's buttons (figure 3-10) are hardened and ground cylindrical pieces of steel, used to locate the centers of holes with extreme accuracy. You may use as many buttons as necessary on the same layout by spacing them the proper distance from each other with gage blocks.

Many other special tools, which may be made by you, will be useful in obtaining layouts that are accurate and easily done. Transfer screws and punches for laying out from a sample are two that you can use on many jobs and save time in doing the job.

LAYOUT METHODS

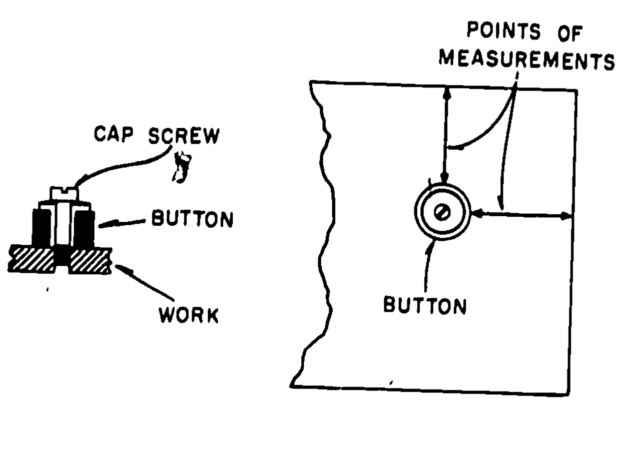
To ensure complete accuracy when making layouts, establish a reference point or line on the work. This line, called the baseline, is located so that you can use it as a base from which to measure dimensions, angles, and lines of the layout. You can use a machined edge or centerline as a reference line. Circular layouts, such as flanges, are usually laid out from a center point and a diameter line.

You can hold inaccuracy in layouts to a minimum by using the reference method because errors can be made only between the reference line and one specific line or point. Making a layout by referencing each line or point to the preceding one can cause you to compound any error, thus an inaccurate layout.

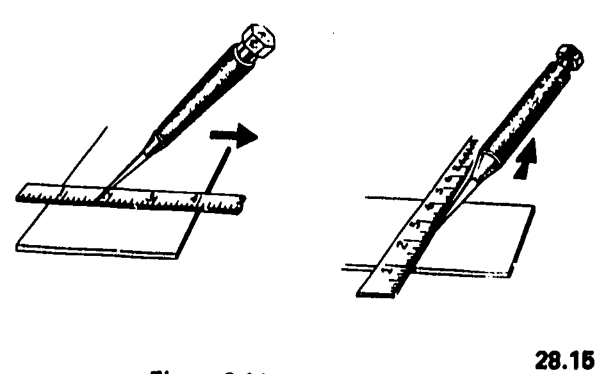
Making a layout on stock that has one or more machine finished surfaces usually is easy. Laying out a casting, however, presents special problems because the surfaces are too rough and not true enough to permit the use of squares, surface plates, or other mounting methods with any degree of accuracy. A casting usually must be machined on all surfaces. Sufficient material must be left outside the layout line for truing up the surface by machining. For example, a casting might have only 1/8-inch machining allowance on each surface (or a total of 1/4-inch oversize). It is obvious in this example that taking more than 1/8 inch off any surface would mean the loss of the casting. The layout procedure is especially important when there are irregular surfaces or cored holes in the casting. The layout lines then must be within the machining allowance on all surfaces. Do NOT attempt to make the layout so that a maximum amount of material is removed from one surface and a minimum amount from another surface.

Making Layout Lines

The following information applies to practically all layouts. Layout lines are formed by using a reference edge or point on the stock or by using the surface plate as a base. Study carefully the section on geometric construction as this will aid you in making layouts when a reference edge of the stock or a surface plate mounting of the stock cannot be used.



28.323
Figure 3-10.—Toolmaker's buttons and their application.



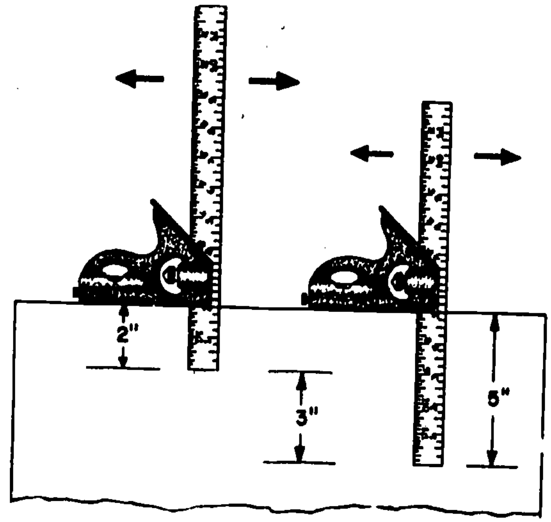
28.15
Figure 3-11.—Using a scribe.

LINES SQUARE OR PARALLEL TO EDGES.–When scribing layout lines on sheet metal, hold the scratch awl, or scribe, as shown in figure 3-11, leaning it toward the direction in which it will be moved and away from the straightedge. This will help scribe a smooth line which will follow the edge of the straightedge, template, or pattern at its point of contact with the surface of the metal.

To square a line on stock with a combination square, place the squaring head on the edge of the stock, as shown in figure 3-12. Draw the line along either edge of the blade. The line will be square with the edge of the stock against which the squaring head is held; that is, the angle between the line and the edge will be 90°.

To draw lines parallel to an edge using a combination square, extend the blade from the squaring head the required distance, such as the 2-inch setting shown in figure 3-13. Secure the blade at this position. Scribe a line parallel to the edge of the stock by holding the scratch awl, or scribe, at the end of the blade as you move the square along the edge. All lines so scribed, with different blade settings, will be parallel to the edge of the stock and parallel to each other.

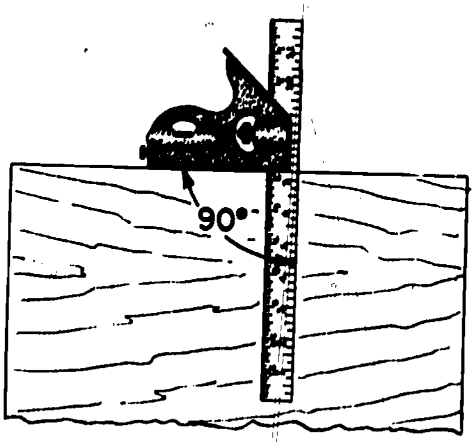
To scribe a line parallel to an edge with a hermaphrodite caliper, hold the caliper, as shown in figure 3-14, so that the curved leg



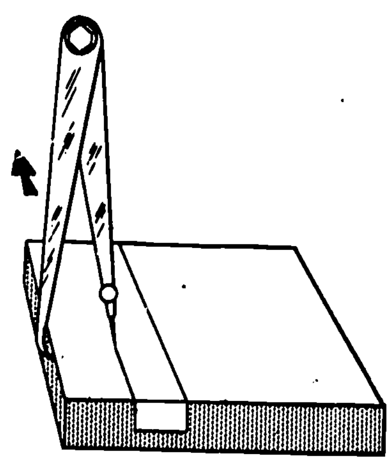
28.17
Figure 3-13.–Laying out parallel lines with a combination square.

maintains contact with the edge while the other leg scribes the line. Hold the caliper in such a way that the line will be scribed at the desired distance from the edge of the stock.

FORMING ANGULAR LINES.–To lay out a 45° angle on stock, using a combination

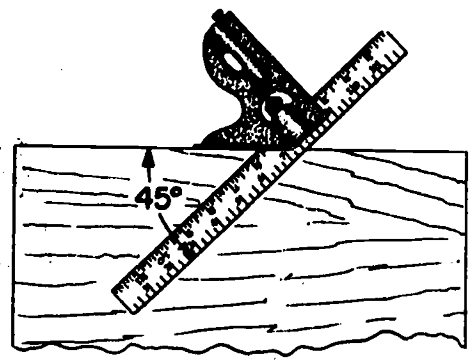


28.16
Figure 3-12.–Using the combination square.



28.18
Figure 3-14.–Laying out a parallel line with a hermaphrodite caliper.

50



28.19

Figure 3-15.—Laying out a 45° angle.

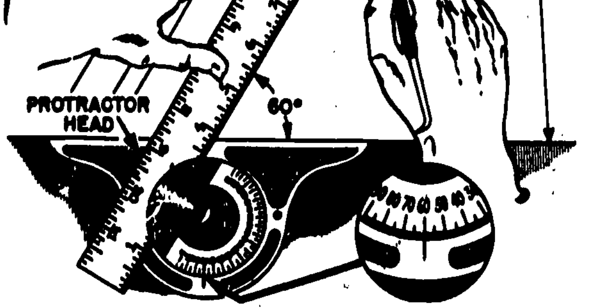
square, place the squaring head on the edge of the stock, as shown in figure 3-15, and draw the line along either edge of the blade. The line will form a 45° angle with the edge of the stock against which the squaring head is held.

To draw angular lines with the protractor head of a combination square, loosen the adjusting screw and rotate the blade so that the desired angle lines up with the index mark on the body of the protractor head. The setting shown in figure 3-16 is 60°. Tighten the screw to hold the setting.

Hold the body of the protractor head in contact with a true edge of the work with the blade resting on the surface. Scribe the lines along the edge of the blade on the surface of the work. The angle set on the scale determines the angle laid out on the work. All lines drawn with the same setting, and from the same true edge of the work, will be parallel lines.

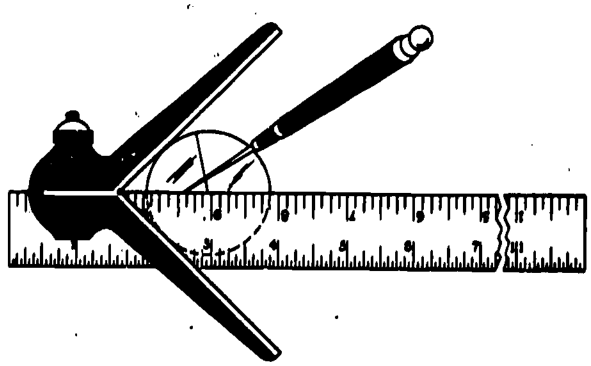
LAYING OUT CIRCLES AND IRREGULAR LINES.—Circles or segments of circles are laid out from a center point. To ensure accuracy, prick-punch the center joint to keep the point of the dividers from slipping out of position. Use the center head and rule as illustrated in figure 3-17, to locate the center of round stock. To find the center of square and rectangular shapes, scribe straight lines from opposite corners of the workpiece. The intersection of the lines locates the center.

To lay out a circle with a divider, take the setting of the desired radius from the rule, as shown in figure 3-18. Note that the 3-inch setting is being taken AWAY from the end of the rule. This reduces the chance of error as each point of the dividers can be set on a graduation. Place one leg of the divider at the center of the



28.20

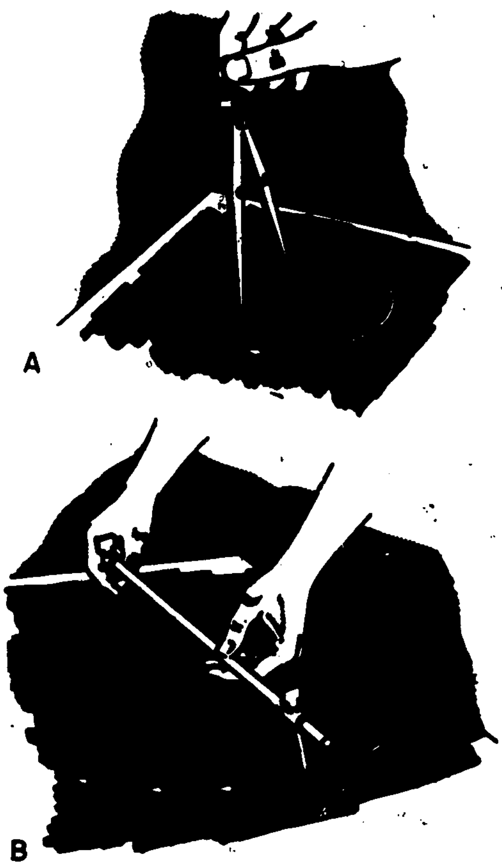
Figure 3-16.—Laying out angular lines.



28.21

Figure 3-17.—Locating the center of round stock.

4.18
Figure 3-18.—Setting a divider to a dimension.



28.22
Figure 3-19.—Laying out circles.

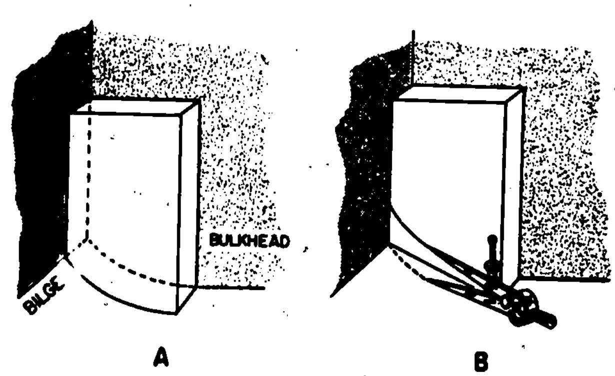
proposed circle, lean the tool in the direction it will be rotated, and rotate it by rolling the knurled handle between your thumb and index finger. (A of fig. 3-19.)

When setting trammel points, shown in B of figure 3-19, follow the same directions as for a divider; you may need a steel tape to set the trammel points.

To lay out a circle with trammel points, hold one point at the center, lean the tool in the direction you plan to move the other point, and swing the arc, or circle, as shown in B of figure 3-19.

To transfer a distance measurement with trammel points, hold one point as you would for laying out a circle and swing a small arc with the other point opened to the desired distance.

Scribing an irregular line to a surface is a skill used in fitting a piece of stock, as shown in figure 3-20, to a curved surface. In A of figure 3-20 you see the complete fit. In B of figure 3-20 the divider has scribed a line from left to right. When scribing horizontal lines, keep the legs of the divider plumb (one above the other). When scribing vertical lines, keep the legs level. To scribe a line to an irregular surface, set the divider so that one leg will follow the irregular surface and the other leg will scribe a line on the material that is being fitted to the irregular surface. (See B of fig. 3-20.)



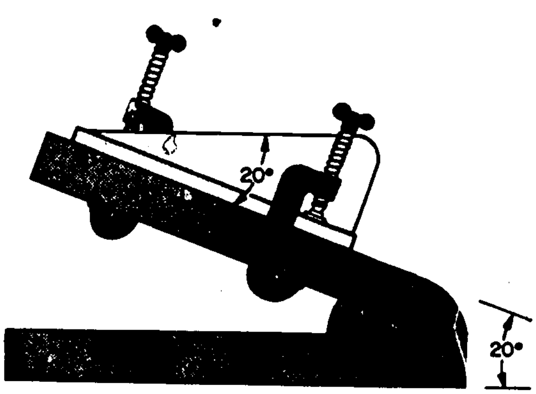
28.23
Figure 3-20.—Laying out an irregular line from a surface.

USING THE SURFACE PLATE.—The surface plate is used with such tools as parallels, squares, V-blocks, surface gages, angle plates, and sine bar in making layout lines. Angle plates similar to the one shown in figure 3-21 are used to mount work at an angle on the surface plate. To set the angle of the angle plate, use a protractor and rule of the combination square set or use a vernier protractor.

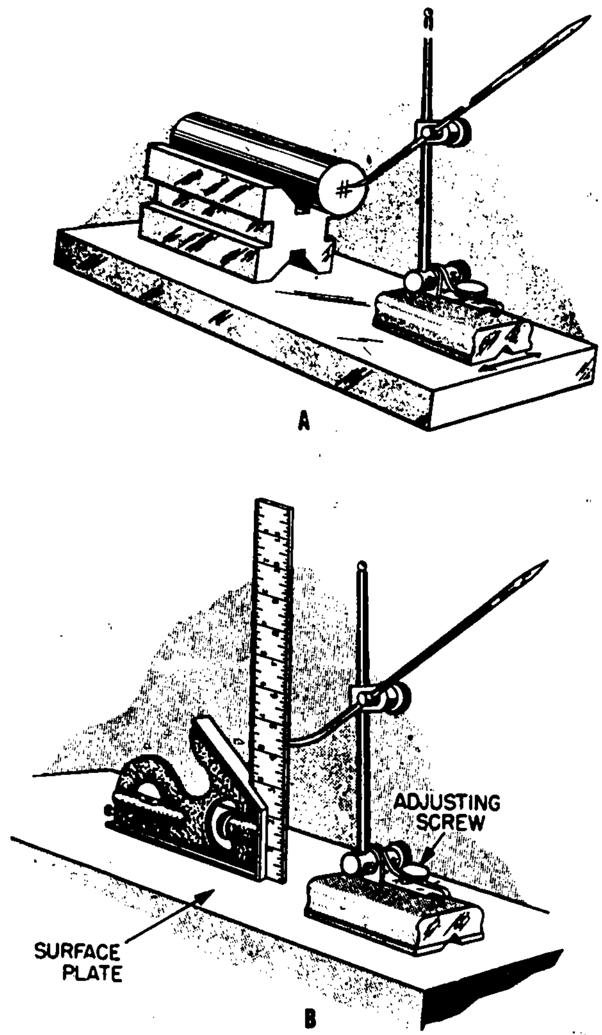
Part A of figure 3-22 shows a surface gage V-block combination used in laying out a piece of stock. To set a surface gage for height, first [illegible] the top of the surface plate and the [illegible]om of the surface gage. Then place the squaring head of a combination square, as shown in B of figure 3-22. Secure the scale so that the end is in contact with the surface of the plate. Move the surface gage into position.

USING THE SINE BAR.—A sine bar is a precisely machined tool steel bar used in conjunction with two steel cylinders. In the type shown in figure 3-23, the cylinders establish a precise distance of either 5 inches or 10 inches from the center of one to the center of the other, depending upon the model used. The bar itself has accurately machined parallel sides, and the axes of the two cylinders are parallel to the adjacent sides of the bar within a close tolerance. Equally close tolerances control the cylinder roundness and freedom from taper. The slots or holes in the bar are for convenience in clamping workpieces to the bar. Although the illustrated bars are typical, there is a wide variety of specialized shapes, widths, and thicknesses.

The sine bar itself is very easy to set up and use. You do need to have a basic knowledge of trigonometry to understand how it works. When a sine bar is set up, it always forms a right triangle. A right triangle has one 90° angle. The base of the triangle, formed by the sine bar, is the surface plate, as shown in figure 3-23. The



28.24
Figure 3-21.—Angle plate.



28.25
Figure 3-22.—Setting and using a surface gage.

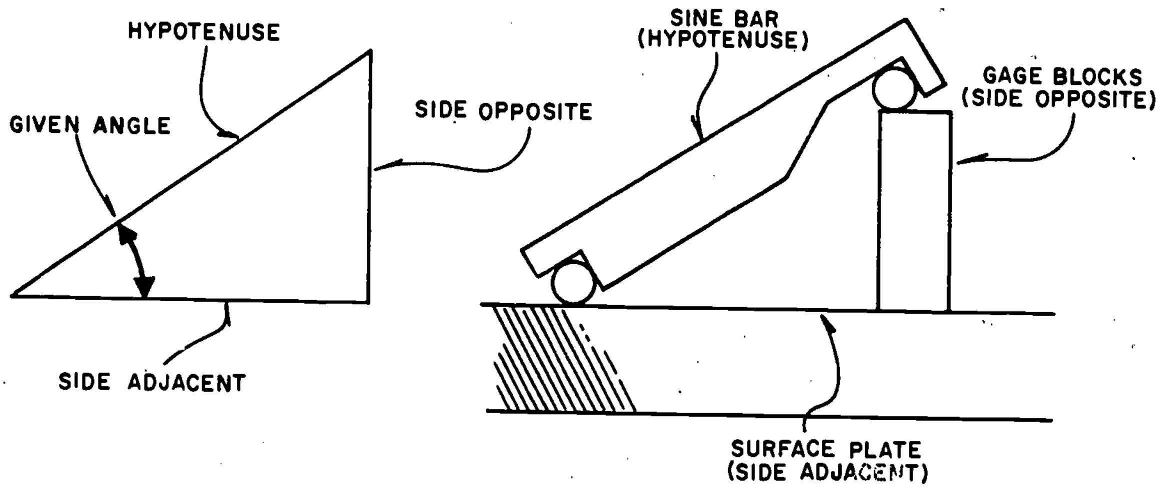


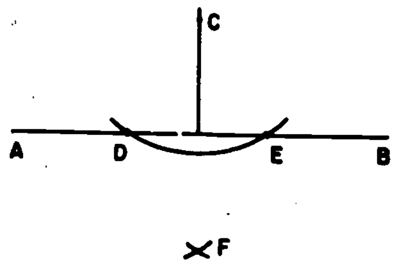
Figure 3-23.—Setup of the sine bar.

side opposite is made up of the gage blocks that raise one end of the sine bar. The hypotenuse is always formed by the sine bar, as shown in figure 3-23. The height of the gage block setting may be found in two ways. The first method is to multiply the sine of the angle needed by the length of the sine bar. The sine of the angle may be found in any table of natural trigonometric functions. For example, if you had to set a 10-inch sine bar to check a 30°5′ angle on a part, you would first go to a table of natural trigonometric functions and find the sine of 30°5′. Then multiply by 10 inches: .50126 X 10 = 5.0126, which would be the height of the gage blocks. The second method is to use a table of sine bar constants. These tables give the height setting for any given angle (to the nearest minute) for a 5-inch sine bar. Tables are not normally available for 10-inch bars because it is just as easy to use the sine of the angle and move the decimal point one place to the right.

Although sine bars have the appearance of being rugged, they should receive the same care as gage blocks. Because of the nature of their use in conjunction with other tools or parts that are heavy, they are subject to rough usage. Scratches, nicks, and burrs should be removed or repaired. They should be kept clean from abrasive dirt, sweat, and other corrosive agents. Regular inspection of the sine bar will locate such defects before they are able to affect the accuracy. When sine bars are stored for extended periods, all bare metal surfaces should be cleaned and then covered with a light film of oil. Placing a cover over the sine bar will further prevent accidental damage and discourage corrosion.

GEOMETRIC CONSTRUCTION OF LAYOUT LINES.—Sometimes you will need to scribe a layout that cannot be made using conventional layout methods. For example, you cannot readily make straight and angular layout lines on sheet metal with irregular edges by using a combination square set; neither can you mount sheet metal on angle plates in a manner that permits scribing angular lines. Geometric construction is the answer to this problem.

Use a divider to lay out a perpendicular FROM a point TO a line, as shown in figure 3-24. Lightly prick-punch point C, then swing any arc from C which will intersect the line AB, and prick-punch intersections D and E as in the figure. With D and E as centers, scribe two arcs which intersect at a point such as F. Place a straightedge on points C and F. The line drawn along this straightedge from point C to line AB will be perpendicular (90°) to line AB.

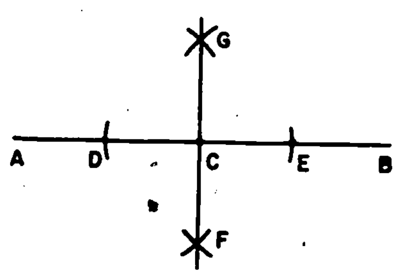


28.26
Figure 3-24.—Layout of a perpendicular from a point to a line.

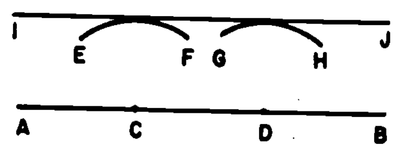
Use a divider to lay out a perpendicular FROM a point ON a line, as shown in figure 3-25. Lightly prick-punch the point identified in the figure as C on line AB. Then set the divider to any distance to scribe arcs which intersect AB at D and E with C as the center. Punch C and E lightly. With D and E used as centers and with the setting of the divider increased somewhat, scribe arcs which cross at points such as F and G. The line drawn through F and G will pass through point C and be perpendicular to line AB.

To lay out parallel lines with a divider, set the divider to the selected dimension. Then referring to figure 3-26, from any points (prick-punched) such as C and D on line AB, swing arcs EF and GH. Then draw line IJ tangent to these two arcs and it will be parallel to line AB and at the selected distance from it.

Bisecting an angle is another geometric construction with which you should be familiar.



11.216
Figure 3-25.—Layout of a perpendicular from a point on a line.



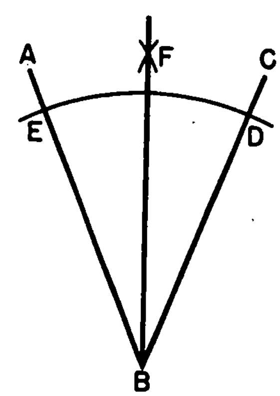
28.27
Figure 3-26.—Layout of a parallel line.

Angle ABC (fig. 3-27) is given. With B as a center, draw an arc cutting the sides of the angle at D and E. With D and E as centers, and with a radius greater than half of arc DE, draw arcs intersecting at F. A line drawn from B through point F bisects the angle ABC.

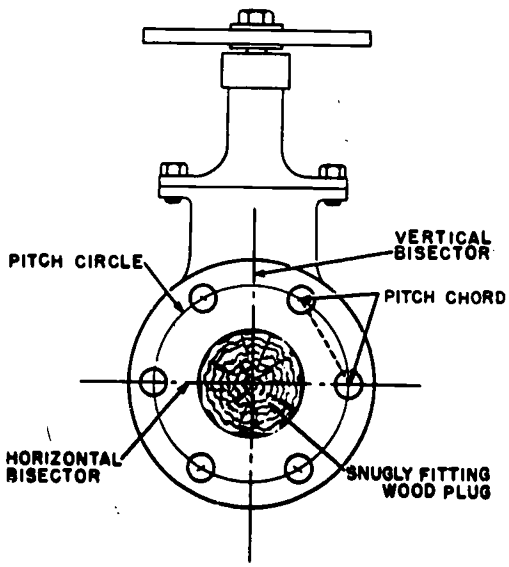
Laying Out Valve Flange Bolt Holes

Before describing the procedure for making valve flange layouts, we need to clarify the terminology used in the description. Figure 3-28 shows a valve flange with the bolt holes marked on the bolt circle. The straight-line distance between the centers of two adjacent holes is called the PITCH CHORD. The bolt hole circle itself is called the PITCH CIRCLE. The vertical line across the face of the flange is the VERTICAL BISECTOR, and the horizontal line across the face of the flange is the HORIZONTAL BISECTOR.

The bolt holes center on the pitch circle and are equidistant; the pitch chord between any



11.219
Figure 3-27.—Bisecting an angle.



28.28

Figure 3-28.—Flange layout terminology.

two adjacent holes is exactly the same as the pitch chord between any other two adjacent holes. Note that the two top holes and the two bottom holes straddle the vertical bisector; the vertical bisector cuts the pitch chord for each pair exactly in half. This is the standard method of placing the holes for a 6-hole flange. In the 4-, 8-, or 12-hole flange, the bolt holes straddle both the vertical and horizontal bisectors. This system of hole placement permits a valve to be installed in a true vertical or horizontal position, provided, of course, that the pipe flange holes are also in standard location on the pitch circle. Before proceeding with a valve flange layout job, find out definitely whether the holes are to be placed in standard position. If you are working on a "per sample" job, follow the layout of the sample.

Assuming that you have been given information relative to the size and number of holes and the radius of the pitch circle, the procedure for setting up the layout for straight globe or gate valves is as follows:

1. Fit a fine grain wood plug into the opening in each flange. (See fig. 3-28.) The plug should fit snugly and be flush with the face of the flange.

2. Apply layout dye to the flange faces, or, if dye is not available, rub chalk on the flange faces to facilitate the drawing of lines which will be clearly visible.

3. Locate the center of each flange with a surface gage, or with a center head and rule combination, if the flange diameter is relatively small. (See part A fig. 3-22 and fig. 3-17.) After you have the exact center point located on each flange, mark the center with a sharp prick-punch.

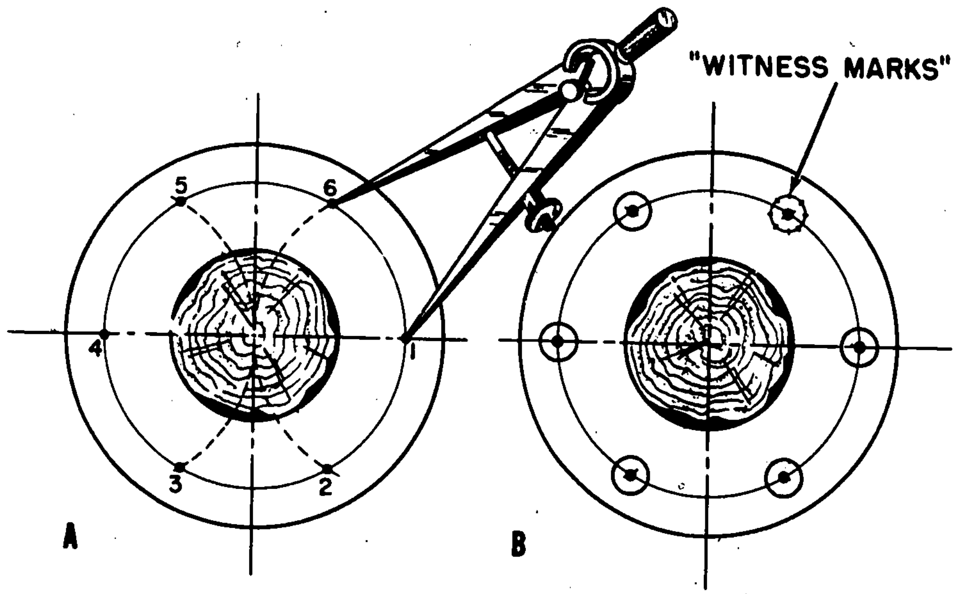
4. Scribe the pitch or bolt circle, using a pair of dividers. Check to see that the pitch circle and the outside edge of the flange are concentric.

5. Draw the vertical bisector. This line must pass through the center point of the flange and it must be visually located directly in line with the axis of the valve stem. (See fig. 3-28.)

6. Draw the horizontal bisector. This line must also pass through the center point of the flange and must be laid out at right angles to the vertical bisector. (See fig. 3-28 and fig. 3-25.)

Up to this point, the layout is the same for all flanges regardless of the number of holes. Beyond this point, however, the layout differs with the number of holes. The layout for a 6-hole flange is the simplest one and will be described first.

SIX-HOLE FLANGE.—Set your dividers exactly to the dimension of the pitch circle radius. Place one leg of the dividers on the point where the horizontal bisector crosses the pitch circle on the right-hand side of the flange, point (1) in part A of figure 3-29, and draw a small arc across the pitch circle at points (2) and (6). Next, place one leg of the dividers at the intersection of the pitch circle and horizontal bisector on the left-hand side of the flange point (4), and draw a small arc across the pitch circle line at points (3) and (5). These points, (1 to 6), are the centers for the holes. Check the accuracy



28.29

Figure 3-29.—Development of a 6-hole flange.

of the pitch chords. To do this, leave the dividers set exactly as you had them set for drawing the arcs. Starting from the located center of any hole, step around the circle with the dividers. Each pitch chord must be equal to the setting of the dividers; if it is not, you have an error in hole mark placement that must be corrected before you center punch the marks for the holes. After you are sure the layout is accurate, center punch the hole marks and draw a circle of appropriate size around each center-punched mark and prick-punch "witness marks" around the circumference as shown in part B of figure 3-29. These witness marks will be cut exactly in half by the drill to verify a correctly located hole.

FOUR-HOLE FLANGE.—Figure 3-30 shows the development for a 4-hole flange layout. Set your dividers for slightly more than half the distance of arc AB, and then scribe an intersecting arc across the pitch circle line from points A, B, C, and D, as shown in part A of figure 3-30. Next, draw a short radial line through the point of intersection of each pair of arcs as shown in part B. The points where these lines cross the pitch circle, (1), (2), (3), and (4), are the centers for the holes. To check the layout for accuracy, set your divider for the pitch between any two adjacent holes and step around the pitch circle. If the holes are not evenly spaced, find your error and correct it. When the layout is correct, follow the center-punching and witness-marking procedure described for the 6-hole flange layout.

EIGHT-HOLE FLANGE.—Figure 3-31 showns the development of an 8-hole placement. The procedure is as follows: First, locate point E by the same method as described for locating point (1) in the 4-hole layout. Then divide arc AE in half by the same method. The midpoint of arc AE is the location for the center of hole (1). (See part A of fig. 3-31.) Next, set your dividers for distance A (1), and draw an arc across the pitch circle line from A at point (8); from B at points (2) and (3); from C at (4) and (5); and from D at (6) and (7). (See part B of fig. 3-31.) Now set your calipers for distance AE and gage the pitch chord for accuracy. Then finish the layout as described in the preceding paragraphs.

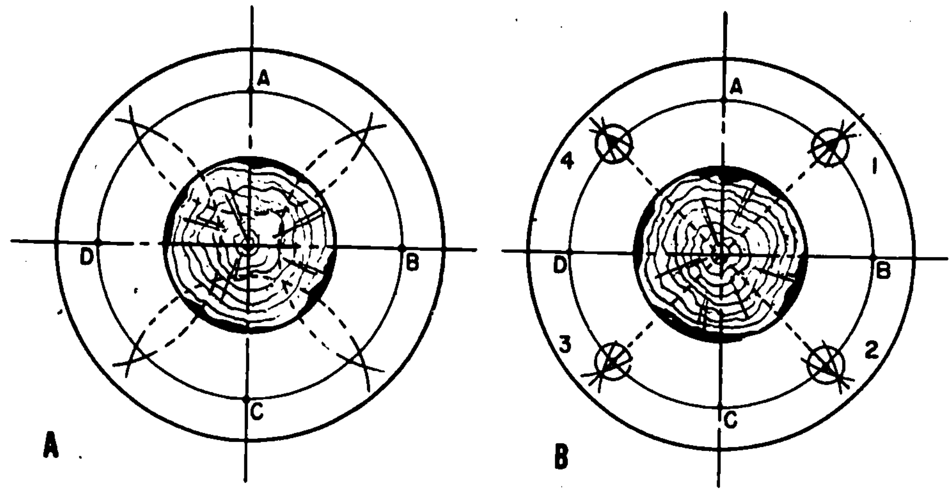


Figure 3-30.—Four-hole flange development.

28.30

MATHEMATICAL DETERMINATION OF PITCH CHORD LENGTH.–In addition to the geometric solutions given in the preceding paragraphs, the spacing of valve flange bolt hole centers can be determined by simple multiplication, provided a constant value for the desired number of bolt holes is known. The diameter of the pitch circle multiplied by the constant equals the length of the pitch chord. The constants for specified numbers of holes are given in table 3-2.

Here is an example of the use of the table. Suppose a flange is to have 9 bolt holes laid out on a pitch circle with a diameter of 10 inches. From the table, select the constant for a 9-hole flange. The pitch diameter (10 inches) multiplied by the appropriate constant (.342) equals the length of the pitch chord (3.420

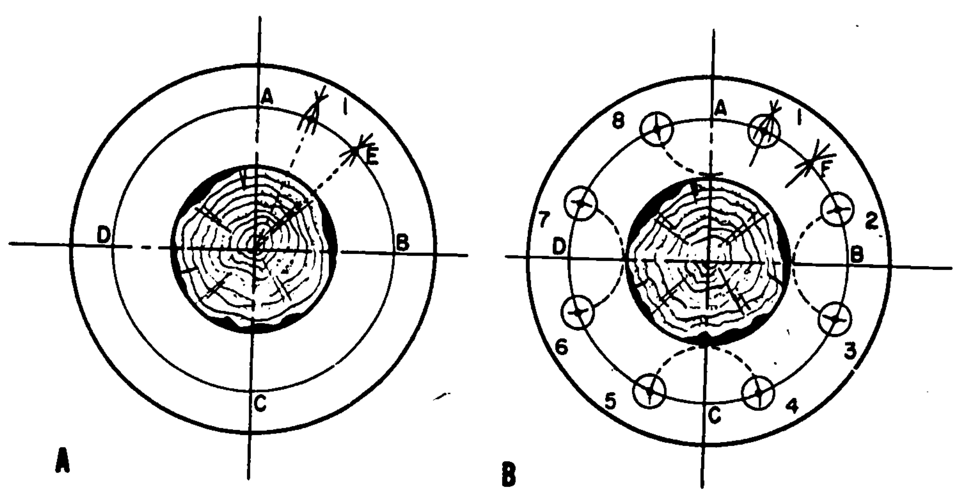


Figure 3-31.—Eight-hole flange development.

28.31

Table 3-2.—Constants for Locating Centers of Flange Bolt Holes

No. bolt holes	Constant
3	0.866
4	.7071
5	.5879
6	.5
7	.4338
8	.3827
9	.342
10	.309
11	.2817
12	.2588
13	.2394
14	.2225
15	.2079
16	.195
17	.184
18	.1736
19	.1645
20	.1564

28.294

inches). Set a pair of dividers to measure 3.420 inches, from point to point, and step off around the circumference of the pitch circle to locate the centers of the flange bolt holes. Note, however, that the actual placement of the holes in relation to the vertical and horizontal bisectors is determined separately. (This is of no concern if the layout is for an unattached pipe flange rather than for a valve flange.)

BENCHWORK

In this chapter, we will consider benchwork as related to repair work, other than machining, in restoring equipment to an operational status. In repairing equipment, benchwork progresses in several distinct steps: obtaining information, disassembly of the equipment, inspection for defects, repair of defects, reassembly, and testing.

Obtaining information about equipment is an essential step in making repairs. There are many possible sources for this information. Job orders generally give brief descriptions of the equipment and the required repair. Manufacturers' technical manuals and blueprints give detailed information on operational characteristics and physical descriptions of the equipment. Operators can provide information on specific techniques of operation and may furnish clues as to why the equipment failed. The leading petty officer of your shop can provide valuable information on repair techniques, and he will help you interpret the information. Use these sources of information to become familiar with the equipment before attempting the actual repair work. If you are thoroughly acquainted with the equipment, you will not have to rely on trial and error methods which are time consuming and sometimes questionable in effectiveness.

There are specific techniques that can be used in assembly and disassembly of equipment which will improve the effectiveness of a repair job. You should note such things as fastening devices, fits between mating parts, and the uses of gaskets and packing when repairing equipment. Noting the positions of parts in relation to mating parts or the unit as a whole is extremely helpful in ensuring that the parts are ir rrect locations and positions when the unit is 1ᴗassembled.

Inspecting the equipment before and during the repair procedure is necessary to determine causes of defects or damage. The renewal or replacement of a broken or worn part of a unit may give the equipment an operational status. Eliminating the cause of damage prevents recurrence.

Repairs are made by replacement of parts, by machining the parts to new dimensions, or by using handtools to overhaul and recondition the equipment. Handtools are used in the repair procedure in jobs such as filing and scraping to true surfaces and to remove burrs, nicks, and sharp edges.

It is often said that a repair job is incomplete until the repaired equipment has been tested for satisfactory operation. How equipment is tested depends on the characteristics of the equipment. In some cases testing facilities are available in the shop. When these facilities are not available, the unit may be placed back in operation and tested by normal use.

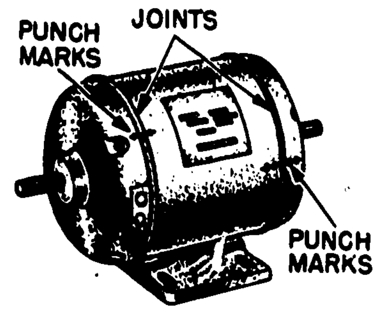
ASSEMBLY AND DISASSEMBLY

You may not be completely familiar with much of the equipment that you are required to disassemble, repair, and reassemble. You must, therefore, use techniques that will aid you in remembering the position and location of parts in relatively intricate mechanisms. The following information applies in general to assembly and disassembly of any equipment.

Equipment should be disassembled in a clean, well-lighted work area. With plenty of light, small parts are less likely to be misplaced or lost, and small but important details are more easily noted. Cleanliness of the work area as well as the proper cleaning of the parts as they are removed, decreases the possibility of damage due to foreign matter when the parts are reassembled.

Before starting any disassembly job, select the tools and parts you think you will need and take them to the work area. This procedure will permit you to concentrate on the work without unnecessary interruptions during the disassembly and reassembly processes.

Have a container at hand for holding small parts to prevent loss. Use tags or other methods of marking to identify the parts with the unit from which they are taken. Doing this prevents mixing parts of one piece of equipment with parts belonging to another similar unit, especially if several pieces of equipment are being repaired in the same area. Use a scribe or prick-punch to mark the relative positions of mating parts that are required to mate in a certain position. (See fig. 3-32.) You must pay close attention to details of the equipment you are taking apart to fix in your mind how the parts fit together. When overhauling equipment, use no more force than necessary to disassemble and reassemble the parts. Doublecheck for overlooked fastening devices if heavy pressure is required to separate parts. An overlooked pin, key, or setscrew that locks parts in place can



28.32
Figure 3-32.—Mating parts location marks.

cause extensive damage if pressure is applied to the parts. If hammers are required to disassemble parts, use a mallet or hammer with a soft face (lead, plastic, or rawhide) to prevent distortion of surfaces. If bolts or nuts or other parts are stuck together due to corrosion, use penetrating oil to free the parts.

PRECISION WORK

The majority of repair work that you perform will involve some amount of precision hand work of parts. Broadly defined, precision hand work to the Machinery Repairman can range from using a file to remove a burr or rough, sharp edge on a hatch dog to reaming a hole for accurately locating very close fitting parts. To accomplish these jobs, you must be proficient in the use of files, scrapers, precision portable grinders, thread cutting tools, reamers, broaches, presses and oxyacetylene torches.

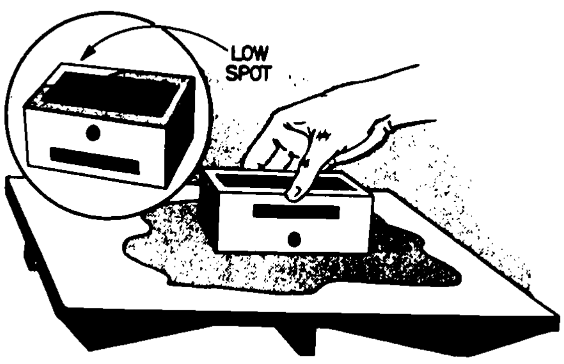
Scraping

Scraping produces a surface that is more accurate in fit and smoother in finish than a surface obtained in a machinery operation. It is a skill that requires a great deal of practice before you become proficient at it. Patience, sharp tools and a light "feel" are required to scrape a surface that is smooth and uniform in fit.

Some of the tools you will use for scraping will be similiar to files without the serrated edges. They are available either straight or with various radii or curves for scraping an internal surface at selected points. Other scraper tools may look like a paint scraper, possibly with a carbide tip attached. You may find that a scraper made by you from material in your shop will best suit the requirements of the job at hand.

A surface plate and nondrying prussian blue are required for scraping a flat surface. Lightly coat the surface plate with blue and move the workpiece over this surface. The blue will stick to the high spots on the workpiece, revealing the areas to be scraped. (See fig. 3-33.) Scrape the areas of the workpiece surface that are blue and check again. Continue this process unitl the blue coloring shows on the entire surface of the workpiece. To reduce frictional "drag" between mating finished scraped surfaces, rotate the solid surfaces so that each series of scraper cuts is made at an angle of 90° to the preceding series. This action gives the finished scraped surface a crosshatched or basket weave appearance. The crosshatched method also enables you to more easily see where you have scraped the part.

A shell-type, babbitt-lined, split bearing or a bushing often requires hand scraping to ensure a proper fit to the surface that it supports or runs on. To do this, very lightly coat the shaft (or a mandrel the same size as the shaft) with nondrying prussian blue. Turning the bearing on the shaft (or the mandrel in the bearing) just a short distance will leave thin deposits of the bluing on the high spots in the bearing babbitt. Then lightly scrape the high spots with a scraper shaped to permit selective scraping of the high spots without dragging along the other areas. Be very careful when doing this to prevent tapering the bearing excessively in either the longitudinal or radial direction. When all the high spots have been worked out, smooth out (or replace if necessary) the bluing on the shaft or mandrel and repeat the process until an acceptable seating pattern is achieved. This job cannot be rushed and done properly at the same time. A poor seating pattern on a bearing could lead to an early failure when it is placed in service.



28.33

Figure 3-33.—Checking a surface.

Removal of Burrs and Sharp Edges

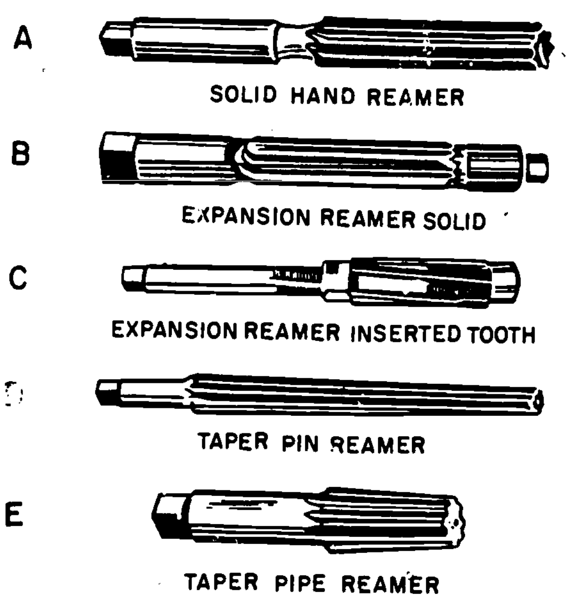
One of the most common injuries that occurs in machine shops is a cut or scratch caused by a sharp edge on a part. When a pump or other machinery that has been overhauled, binds or wipes with little or no operating time, an investigation will often reveal a sharp edge that has peeled or broken off and jammed into an area that has very little clearance. In spite of this and other instances that cause either discomfort or additional work, the removal of burrs and sharp edges is often overlooked by the machinist. Close examination of the old part or the blueprint will sometimes indicate that a machined radius is required. Regardless of the design or use of a part, a few seconds in removing these sharp edges with a file is time well spent.

Hand Reaming

When you need a round hole that is accurate in size and smooth in finish, reaming is the process that you would probably select. There are two types of reaming processes—machine reaming and hand reaming. Machine reaming requires a drill press, lathe, milling machine or other power tool to hold and drive either the reamer or the part. Machine reaming will be covered in chapter 8. Hand reaming is more accurate and the method you will probably use most in precision benchwork.

A hand reamer has a straight shank and a square machined on the end. It is driven by hand with a tap wrench placed on the square end. Several different types of hand reamers are available, as shown in figure 3-34. Each of the different types has an application for which it is best suited and a limiting range or capability. The solid hand reamer in part A of figure 3-34 is used for general purpose reaming operations where a standard or common fractional size is required. It is made with straight, helical, or spiral flutes. A helical fluted reamer is used when an interrupted cut, such as a part with a keyway through it, must be made. The helical flutes ensure a greater contact area of the cutting edges than the straight fluted reamer, preventing the reamer from hanging up on the keyway and causing chatter, oversizing and poor finishes.

The expansion reamer in B of figure 3-34 is available as either straight or helical fluted. These reamers are used when a reamed hole slightly larger than the standard size is required. Expansion reamers can be adjusted from about



28.324

Figure 3-34.—Hand reamers.

0.006 inch larger for a 1/4-inch reamer to about 0.012 inch larger for a 1 1/2-inch reamer. The adjustment is made by turning the screw on the cutting end of the reamer.

The expansion reamer in C of figure 3-34 has a much greater range for varying size. Each reamer is adjustable to provide an overlap of the smallest diameter of the next larger reamer. The cutting blades are the insert type and can be removed and replaced when they become dull. Adjustment is made by loosening and tightening the two nuts on each side of the blades.

The taper pin reamer in D of figure 3-34 has a taper of 1/4 inch per foot and is used to ream a hole to accept a standard size taper pin. This reamer is used most often when two parts require a definite alignment position. When drilling the hole for this reamer, it is often necessary to step drill through the part with several drills of different sizes to help reduce the cutting pressure put on the reamer. Charts which give the recommended drill sizes are available in several machinist reference books. In any case, the smallest drill used cannot be larger than the small diameter of the taper pin.

The taper pipe reamer in E of figure 3-34 has a taper of 3/4 inch per foot and is used to prepare a hole that is to be threaded with a tapered pipe thread.

The size of the rough drilled or bored hole to be hand reamed should be between 0.002 inch and about 0.015 inch (1/64) smaller than the reamer size. A smoother and more accurate reamed hole can be produced by keeping to a minimum the amount of material that a reamer is to remove. You must be careful to keep the rough hole from being oversized or out-of-round. This is a very common problem in drilling holes, and you can prevent it only by using a correctly sharpened drill under the most closely controlled conditions possible. Information on drilling can be found in chapter 5.

Alignment of the reamer to the rough hole is a critical factor in preventing oversized, out-of-round or bell-mouthed holes. If possible, perform the reaming operation while the part is still set up for the drilling or boring operation. Then insert a center in the spindle of the machine and place it in the center hole in the shank of the reamer to guide it.

Another method of alignment is to fabricate a fixture with guide bushings made from bronze or a hardened steel to keep the reamer straight. When a rough casting or a part that has the reamed hole at an angle to its surface must be reamed, it is best to spot face or machine the area next to the hole so that they are perpendicular. This will prevent an uneven start and possibly reamer breakage. In most reaming operations, you will find that the use of a lubricant will give a better reamed hole. The lubricant or cutting fluid helps to reduce heat and friction and washes away the chips that build up on the reamer. Soluble oil will normally serve very well; however, in some cases, a lard or sulfurized cutting oil may be required. When the reaming operation is complete, remove the reamer from the part by continuing to turn the reamer in the same direction (clockwise) and putting a slight upward pressure on it with your hand until it has cleared the hole completely. Reversing the direction of the reamer will probably result in damage to the cutting edges and the hole.

A straight hand reamer is generally tapered on the beginning of the cutting edges for a distance approximately equal to the diameter of the reamer. You will have to consider this when reaming a hole that does not go all the way through a part.

Broaching

Broaching is a machining process that cuts or shears the material by forcing a broach through the part in a single stroke. A broach is a tapered, hardened bar, into which has been cut teeth that are small at the beginning of the tool and get progressively larger toward the end of the tool. The last several teeth will usually be the correct size of the desired shape. Broaches are available to cut round, square, triangular and hexagonal holes. Internal splines and gears and keyways can also be cut using a broach. A keyway broach requires a bushing that will fit snugly in the hole of the part and has a rectangular slot in it to slide the broach through. Shims of different thicknesses are placed behind the broach to adjust the depth of the keyway cut.

A broach is a relatively expensive cutting tool and is easily rendered useless if not used and handled properly. Like all other cutting tools, it should be stored so that no cutting edge is in contact with any object that could chip or dull it. Preparation of the part to be broached is as important as the broaching operation itself. The size of the hole should be such that the beginning pilot section enters freely but does not allow the broach to freely fall past the first cutting edge or tooth. If the hole to be broached has flat sides opposite each other, you need only to measure across them and allow for some error from drilling. The broach will sometimes have the drill size printed on it. Be sure the area around the hole to be broached is perpendicular on both the entry and exit sides.

Most Navy machine shop applications involve the use of either a mechanical or a hydraulic press to force the broach through the part. A considerable amount of pressure is required to broach, so be sure that the setup is rigid and that all applicable safety precautions are strictly observed. A slow even pressure in pushing the broach through the part will produce the most accurate results with the least damage to the broach and in the safest manner. Do not bring the broach back up through the hole, push it on through and catch it with a soft cushion of some type. A lubricant is required for broaching most metals. A special broaching oil is best; however, lard oil or soluble oil will help to cool the tool, wash away chips and prevent particles from galling or sticking to the teeth.

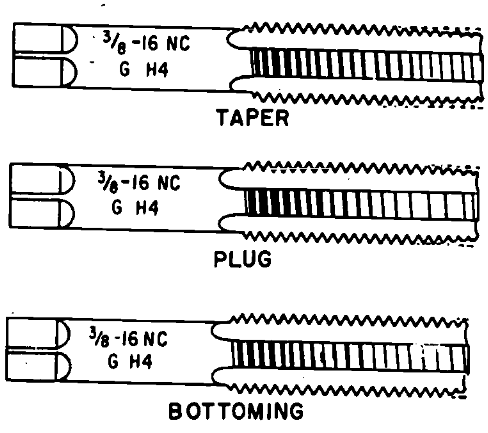
Hand Taps and Dies

Many of the benchwork projects that you do will probably have either an internally or an externally threaded part in the design specifications. The majority of the threads cut on a benchwork project are made with either hand taps, in the case of externally threaded parts, or hand dies for internally threaded parts. The use of these two cutting tools has come to be considered as a simple skill requiring little or no knowledge of the tools and no preplanning of the operation to be performed. It is true that the operations are simple, but only after several factors concerning the correct selection and operational procedures of the tools have been studied and practiced. Taps and dies are fast and accurate cutting tools that can make a job much easier and will produce an excellent end product. The information given in the following paragraphs will provide the general knowledge and operational factors to start you in the correct use of taps and dies.

TAPS.–Hand taps (figure 3-35) are precision cutting tools which usually have three or four flutes and a square on the end for placing a tap wrench to turn the tap. Taps are made from either hardened carbon steel or high-speed steel and are very hard and brittle. They are easily broken or damaged when treated rough or forced too quickly through a hole.

Taps for most of the different thread forms, described later in this manual, are available either as a standard stock item or can be special ordered from a tap manufacturer. The information in this section includes only the most commonly used thread forms, the Unified thread and the American National thread. Both of these thread systems have a 60-degree included angle or V form.

Taps usually come in a set of three for each different diameter and number of threads per inch. A taper, or starting tap (figure 3-35), has 8 to 10 of the beginning teeth that are tapered. The taper allows each cutting edge or tooth to cut slightly deeper than the one before it. This permits an easier starting for the tap and exerts a minimum amount of pressure against the tool. The next several teeth after the taper ends are at the full designed size of the tap. They remove only a small amount of material and help to leave a fine finish on the threads. The last few teeth have a very slight back taper that allows the tap to clear the final threads cut without rubbing or binding. The plug tap has 3 to 5 of the beginning teeth tapered and the remaining length has basically the same design as the taper tap. The bottoming tap has only 1 to 1 1/2 of the beginning teeth tapered. Since the greatest amount of material is removed by the tapered teeth, it is always advisable to begin the tapping operation with the taper, or starting tap. If the hole being tapped goes all the way through the material, the taper tap is usually the only one required. If the hole is a blind one, or does not go all the way through the material, all three taps will be required. The taper tap will be used first, followed by the plug tap, and the final pass will be made with the bottoming tap.



11.5

Figure 3-35.–Set of taps.

Standard Sizes and Designations.–The size of a tap is marked on the shank or the smooth area between the teeth and the square on the end. The numbers and letters always follow the same pattern and are simple to understand. As an example, the marking 3/8 - 16 NC (fig. 3-35) means that the diameter of the tap is 3/8 inch and that it has 16 threads per inch. The NC is a symbol indicating the thread series. In this case, the NC stands for the American National Coarse Thread Series.

Some of the more common thread series symbols are NE, American National Fine; NS, American National Special; NEF, American National Extra Fine; NPT, American National Standard Tapered pipe. A "U" placed in front of one of these symbols indicates the UNIFIED THREAD SYSTEM, a system that has the same basic form as the American National and is interchangeable with it, differing mainly in tolerance or clearance. These thread systems will be covered in more detail in chapter 9. If an LH appears on the marking after the thread series symbols, the tap is left-handed.

The next group of markings usually found on taps refers to the method of producing the threads on the tap and the tolerance of the tap. As an example, in the marking G H4 (fig. 3-35) the G indicates that the threads were ground on the tap. The greatest majority of the taps manufactured today are ground. The next symbol, H4, refers to the tolerance of the tap. The H means that the tap has a pitch diameter that is above (HIGH) the basic pitch diameter for that size tap. An L means that the pitch diameter is under (LOW) the basic pitch diameter for that size tap. The number following the H or L indicates the amount of tolerance in increments of 0.0005 inch. In the example H4, the pitch diameter is a maximum of 0.002 inch (4 X 0.0005) above the basic pitch diameter. In the case of an L, the amount is under the basic pitch diameter. A number of 1 through 10 can be found on taps. This tolerance limit symbol plays an important part in obtaining the correct fit for a given class of threads. Thread classes will be covered later in this manual.

The only difference in the size and designation markings for taps that will probably be found in Navy machine shops is in machine screw diameter taps, or numbered taps as they are often called in the shop. Instead of the diameter being represented by a fraction, a number of 0 through 14 is used. You can easily convert these numbers to a decimal equivalent by remembering that the number 0 tap has a diameter of 0.60 inch and each tap number after that increases in diameter by 0.013 inch. As an example:

Size 0 = 0.60 inch dia.

Size 3 = 0.99 inch dia. [0.060 + (3 X 0.013)]

Size 14 = 0.242 inch dia. [0.060 + (14 X 0.013)]

A typical marking on a tap might be 10.24 UNC, indicating a diameter of 0.190 inch, 24 threads per inch, and a Unified National Coarse thread series.

Tapping Operations.—The first step in any successful tapping operation is the selection of the correct size tap with sharp, unbroken cutting edges on the teeth. A dull tap will require excessive force to produce the threads and increases greatly the chance of the tap breaking and damaging the part being tapped. A dull tap can also produce ragged, torn and undersize threads, leading to a damaged part.

The tap drill or the size of the hole that is made for the tap is very important if the correct fit is to be obtained. If a hole were to be drilled that was equal in size to the minor, or smallest, diameter of the tap, a 100% thread height would result. To tap a hole this size would require excessive pressure and breakage could occur, especially with a small tap or a material that is hard. Unless a blueprint or other design references indicate differently, a 75% thread height is usually considered adequate and is actually only about 5% less in terms of strength or holding power than a 100% thread height. In some of the less critical jobs, it is possible to have a 60% thread height without a significant loss in strength.

There are two simple formulas that may be applied to calculate the tap drill size for any size tap. The simplest and the one most often used will produce a thread height of approximately 75%. It is as follows: DRILL SIZE = TAP DIAMETER MINUS ONE DIVIDED BY THE NUMBER OF THREADS PER INCH: ($DS = TD - \frac{1}{N}$). As an example, the drill size for a 1/4 - 20 NC tap is figured as follows:

Step 1: DS = 1/4 - 1/20

Step 2: DS = 0.250 - 0.050

Step 3: DS = 0.200 in.

The nearest standard size drill would then be selected to make the hole. In this case, a number 8 drill has a diameter of 0.199 in. and a number 7 drill has a diameter of 0.201 in. Unless the size differences are very great, it is more effective to select the larger drill size or the number 7 drill for this tap.

The second formula, although slightly more difficult, allows for a selection of the desired percentage of thread height. To use it, you must know the straight depth of the thread. You can obtain this data from various charts in handbooks for machinists or by using the formulas in chapter 9 of this manual. It is as follows: DRILL SIZE = TAP DIAMETER MINUS THE DESIRED PERCENTAGE OF THREAD HEIGHT TIMES TWICE THE STRAIGHT DEPTH. As an example, if a 60% thread height is desired for a 1/4 - 20 NC tap, the drill size is figured as follows:

Step 1: DS = 1/4 - .60 X 2(0.032)

Step 2: DS = 0.250 - .60 X 0.064

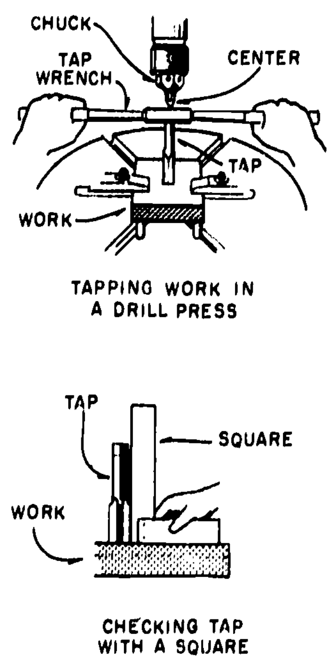
Step 3: DS = 0.250 - 0.038

Step 4: DS = 0.212 in.

The nearest standard size drill to 0.212 inch is a number 3 drill which has a diameter of 0.213 inch. A word of caution about drilling holes for tapping is important at this point. Even if the drill is ground perfectly, the part is rigidly clamped and the drilling machine has no looseness, the drilled hole can be expected to be oversized. In the case of the number 7 and the number 3 drills selected in the two examples given, the drilled holes will probably be approximately 0.003 to 0.004 inch oversize. This should be considered in planning the operation. Additional information on drilling holes is in chapter 5.

After you have selected the tap and tap drill, you must securely clamp the part to be tapped in a vise or drill press table as dictated by the parts size and shape. You MUST be sure that the part cannot vibrate loose and be thrown out of the vise or off the drill press table. When a twist drill driven by a geared motor digs in or binds in a part, a great amount of force is exerted against the part. You could lose a finger or hand, break a leg, or worse if this happens. It is best to start the drilling operation with a small drill or a center drill (described later in this manual) by aligning the drill point as close as possible to the center punch mark you made to locate the center of the hole. When this is completed, insert the tap drill into the drilling machine or drill press and drill the hole. If the hole is very large, you should use a drill several sizes below the tap drill size to prevent an out-of-round or excessively oversized hole. Do NOT move the part when making the various tool changes.

The hole is now ready to be tapped. Some taps have a center hole in the shank that will fit over the point of a center. If this is the case and the setup will allow it, place a center in the drill press without moving the part; place a tap wrench over the square shank, turn the center into the center hole on the tap (fig. 3-36) and slowly turn the tap while applying a slight downward pressure on the center to help guide the tap. If a center cannot be used, align the tap as close as possible by eye and make 2 or 3 turns with the tap handle. Remove the tap handle and place a good square on the surface of the part (if machined flat) and bring it into contact with the shank of the tap at two points 90° from each other (fig. 3-36). If the tap is not perpendicular or square with the surface at both points, back it



28.325

Figure 3-36.—Starting a tap.

out and start over. When the tap is square, begin turning the tap wrench slowly. After making two or three turns, turn the tap backwards to break the chips and help clear them from the path of the taps. Proceed with this until the tap bottoms out; then place the next tap in the set in the hole and repeat the tapping procedure. If the hole is blind, remove the taps often to clear the chips from the bottom.

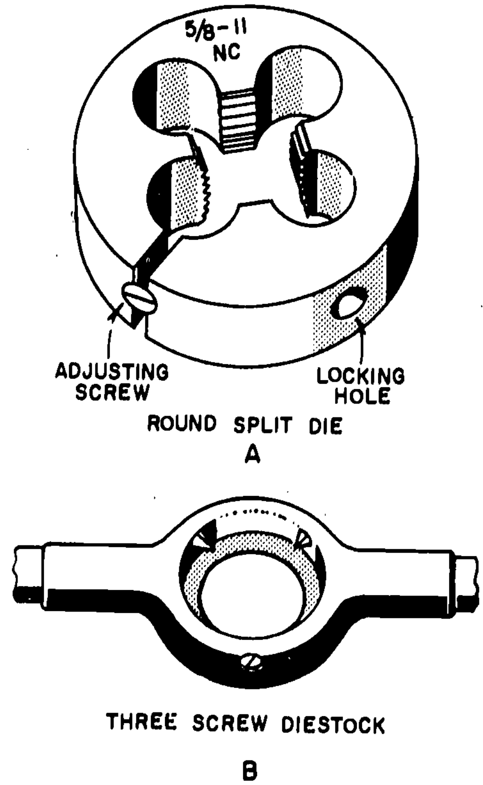
It is often necessary to remove burrs from around a hole that has been tapped. Do this with a file, by slowly hand-spinning a larger twist drill in the hole or by using a countersink.

A cutting oil should be used in most tapping operations. There are several commercial products available that greatly enhance the quality of thread produced. A heavy cutting oil with either a sulfur, mineral oil or lard oil base is available in the supply system. If no other cutting oil is available, a heavy mixture of soluble oil will be acceptable.

DIES.—Hand threading dies come in various styles, including unadjustable solid square and round shaped dies and adjustable single and two-piece dies. The most common die used in Navy machine shops is the adjustable single piece or round split die (fig. 3-37). The adjustable round split die is a round disk-shaped tool which has internal threads and usually four holes or flutes that interrupt the threads and present four sets of cutting edges. The die has a groove cut completely through one side and a setscrew to allow for a small amount of expansion and contraction of the die. This feature permits an adjustment for taking a rough and a finish cut on particularly hard or tough metals and also allows for slight adjustments to obtain a close fit with a mated nut or other internally threaded part. There is a difference in the two sides of the die—the starting side has about 3 full threads tapered and the trailing side has about 1 thread tapered. To prevent damage to the die and the threads being cut, the die should always be started with the greatest taper leading. The die is held in a diestock (fig. 3-37), a tool which has a circular recess to hold the die and three setscrews that fit into small indentations in the outside diameter of the die.

The size of a die is usually marked on the trailing face (the side that is up during threading) and follows the same format as a tap. A die marked 5/8 - 11 NC means that the part threaded will have a 5/8-inch diameter, 11 threads per inch and is of the American National Coarse thread series. The G, H, L, and associated numbers found on a tap are not normally marked on a die because they represent a fixed tolerance and the die is adjustable.

The steps involved in threading a part with a die are similar to those for a tap. The part to be threaded should have a chamfer ground or cut on the end to help in starting the die square with the part. Select the correct die and insert it in the diestock with the longest tapered side opposite the square shoulder. Apply cutting oil and place the die over the part by grasping the diestock in the middle with one hand. Turn the



11.5.2-6

Figure 3-37.—Die and diestock.

die several turns, then look carefully at the die and the part to ensure that they are square. Threads that are deeper on one side than the other indicate a misaligned die. Turn the die about three turns and then back it off one turn to break the chip. After the die has been run down the part far enough to get a few full depth threads, remove it from the part and check the fit with the part that will mate with it. Make any adjustments necessary at this time. Replace the die on the part and continue threading until the desired thread length is reached. If the threads are being cut to a shoulder, you may turn the die over and cut the last 2 or 3 threads with the short tapered side.

Removing Broken Taps

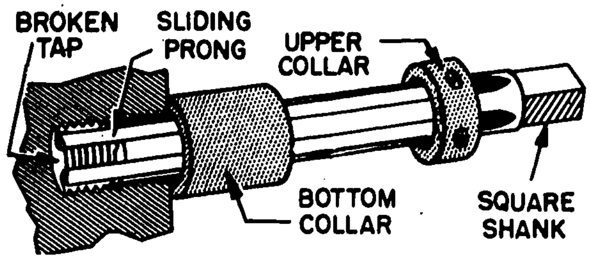
Removing a broken tap is usually a difficult operation and requires slow, deliberate actions to remove it successfully without damaging the part involved. There is no single method that can be applied to all the different circumstances you may experience. The following information describes briefly some of the methods that have proven to be effective. You will need to evaluate the particular problem and attempt removal with the method that will work best.

A tap that has broken and has at least 1/4 inch left protruding above the part can sometimes be grasped by vise grips and removed. Use a scribe first to remove as many as possible of the chips from the hole and the flutes of the tap. Do not use compressed air to remove the chips because there is always a chance that a small chip will be blown into either your eyes or someone's near by. Apply penetrating oil around the threads if possible. Use a small hand grinder to shape the end of the tap to provide a good grip for the vise grips. If they are permitted to slip on the tap, additional fragments will probably break away, giving you less surface to grasp. Apply a slow, even force. Excessive force or jerky movements will cause more damage. You may need to carefully rock or reverse the direction in which you are turning the tap in order to free it. This is especially true in beginning the removal. Use a lubricant once the tap has been loosened in the hole. When the tap has been removed, examine the hole and threads closely to ensure that no fragments of the tap or jagged threads remain to cause problems when another tap is used to finish or clean up the threads.

Another method is to use a punch and apply sharp blows to the broken tap. This method will probably be used when the tap is broken below the surface of the part. Always wear safety goggles and a face shield to protect your face and eyes from flying fragments. Do not allow anyone to stand near you while doing this type operation. The punch should be smaller than the tap to permit placing the point on selected areas of the tap. As a fragment of the tap is broken away, remove it from the hole. This method will probably cause serious damage to the threaded hole when the punch strikes the threads, or an oversized condition can result from forcing the tap around in the hole. You should be sure that there is an approved method of repair or modification of the threaded hole before undertaking this method of removal.

It is sometimes possible to weld a stud to the top of a tap that is broken off below the surface. The tap diameter must be large enough for insertion of both the stud and the welding rod in the hole without running the risk of having the welding rod touch or splatter the threads. There are materials that can be used to help protect the threads. Unless you are an accomplished welder, do not attempt this job. Discuss it with a Hull Maintenance Technician (HT) and let him do it. A more even pressure can be applied in removing the tap if a square is ground on the top of the stud so that a tap wrench can be used. The heat generated by the welding process could expand the tap slightly so that when it cooled and contracted, the tap may have loosened slightly. On the other hand, the tap may bind even more and the structure and condition of the surrounding metal could be changed.

If the tap is broken off below the surface of the part, a tool called a tap extractor (fig. 3-38) can be used to remove it. You should try this method first as it does no damage to the threads. Tap extractors ave available for each of the standard diameter taps over about 3/16 inch. As you see in figure 3-38, the tap extractor has a square end for using a tap wrench and sliding prongs or fingers that fit into each of the flutes



4.37:44.191

Figure 3-38.—Tap extractor.

on the tap. The upper collar is secured in place by setscrews while the bottom collar is free to move. Position the bottom collar as close as possible to the tap of the hole to prevent the sliding prongs from twisting. The best results are obtained from this tool when the sliding prongs have a minimum amount of unsupported length exposed. Apply a slow, even pressure to the tap wrench in removing the tap.

In all of the methods listed, remove all chips prior to beginning the removal process. There are several methods for helping to free the tap that can be used with any of the removal methods if the particular situation lends itself to their use. As previously mentioned, penetrating oil can be applied around the threads. A controlled heat may be applied to the area surrounding the tap to cause expansion. Be very careful to limit the heat so that the tap does not begin to expand also. Since most taps are made from high-speed steel, this probably will not occur, but do not overlook the possibility. You must also consider damage to the part from heat. If the part is very big and has a large mass of metal in the immediate area, the heat will carry to the surrounding area rapidly, preventing adequate heat and expansion where it is needed.

Another method, one that must be conducted under strict safety conditions, is to apply a solution of 1 part nitric acid and 5 parts water to the threaded hole. The nitric acid solution will gradually eat away some of the surface metal and loosen the tap. After the acid solution has worked for a little while, pour it out and rinse the part thoroughly. This method is effective primarily on steel parts. When mixing the acid solution add the acid to the premeasured amount of water. The procedure of adding the acid to the water is a safety measure because some acids react violently when water is added to them. You should wear chemically resistant goggles, a face shield, rubber or plastic gloves, and an apron. Nitric acid can damage your eyes, burn your skin and eat holes in your clothes. If any acid gets on your skin, immediately flush with water for at least 15 minutes and seek medical attention. You will use nitric acid often in identifying metals and you should treat each occasion as seriously as the first, strictly observing every safety precaution.

There is one other method for removing broken taps that is used primarily on tenders, repair ships and shore based repair activities. It involves the use of a special machine (metal disintegrator), electrodes and a coolant. Any metal that will conduct electricity can be worked with this machine. The action of the electrode and the coolant combined create a hole through the part that is equal in size to the diameter of the electrode. There are portable models available; however, most models either have their own cabinet or they are used in a drill press. Detailed information on this method can be found la' r in this manual.

Classes of Fits

The following information concerns fit as applied to plain cylindrical parts such as sleeves, bearings, pump wearing rings and other nonthreaded round parts that fit together. Fit is defined as the amount of tightness or looseness between two mating parts when certain allowances are designed into them. As defined earlier in this chapter, an allowance is the total difference between the size of a shaft and the hole in the part that fits over it. This allowance and the resulting fit can be a clearance (loose) fit, an interference (tight) fit, or a transitional (somewhere between loose and tight) fit. These three general types of fits are further identified by classes of fits, with each class having a different allowance based on the intended use or function of the parts involved. A brief description of each type fit will be given in the following paragraphs. Any good handbook for machinists has complete charts with detailed information on each individual class of fit. The majority of equipment repaired in Navy machine shops will have the dimensional sizes and allowances already specified in either the manufacturer's technical manual, *NAVSHIPS' Technical Manual*, or the appropriate Preventive Maintenance System Maintenance Requirement Card, which is the priority reference on maintenance matters.

CLEARANCE FITS.—Clearance fits or running and sliding fits as they are often called, provide a varying degree of clearance (looseness) depending on which one of the nine classes is selected for use. The classes of fits range from class 1 (close sliding fit), which permits a clearance allowance of from +0.0004 to +0.0012 inch on mating parts with a 2.500 inch basic diameter, to class 9 (loose running fit), which permits a clearance allowance of from +0.009 to +0.0205 inch on the same parts. Even for a basic diameter, the small (2.500 inch) clearance allowance from a class 1 minimum to a class 9 maximum differs by +0.0201 inch. As the basic diameter increases, the allowance increases. Although the class of fit may not be referenced on a blueprint, the dimensions given for the mating parts are based on the service performed by the parts and the specific conditions under which they operate as described in each of the classes of fits. Some parts that fall within these classes of fits are a shaper ram (close sliding), a babbitt-lined bearing, and pump wearing rings (loose removal).

TRANSITIONAL FITS.—Transitional fits are subdivided into three types known as locational clearance, locational transition and locational interference fits. Each of these three subdivisions contains different classes of fits which provide either a clearance or an interference allowance, depending on the intended use and class selected. All of the classes of fits in the transitional category are primarily intended for the assembly and disassembly of stationary parts. Stationary in this sense means that the parts will not rotate against each other although they may rotate together as part of a larger assembly. The allowances used as examples in the following descriptions of the various fits represent the sum of the tolerances of the external and internal parts. To achieve maximum standardization and to permit common size reamers and other fixed sized boring tools to be used as much as possible, it is best to use the unilateral tolerance method previously explained and consult one of the class of fit charts in a handbook for machinists.

Locational clearance fits are broken down into 11 different classes of fits. The same basic diameter with a class 1 fit ranges from a zero allowance to a clearance allowance of +0.0012 inch, while a class 11 fit ranges from a clearance allowance of +0.014 to +0.050 inch. The nearer a part is to a class 1 fit, the more accurately it can be located without use of force.

Locational transition fits have six different classes providing either a small amount of clearance or an interference allowance, depending on the class of fit selected. The 2.500-inch basic diameter in a class 1 fit ranges from an interference allowance of -0.0003 inch to a clearance allowance of +0.0015 inch while a class 6 fit ranges from an interference allowance of -0.002 inch to a clearance allowance of +0.0004 inch. The interference allowance fits may require a very light pressure to assemble or disassemble the parts.

Locational interference fits are divided into five different classes of fits, all of which provide an interference allowance of varying amounts. A class 1 fit for a 2.500-inch basic diameter ranges from an interference allowance of -0.0001 to -0.0013 inch, while a class 5 fit ranges from an interference allowance of from -0.0004 to -0.0023 inch. These classes of fits are used when parts must be located very accurately while maintaining alignment and rigidity. They are not suitable for applications where one part is subjected to a force that causes it to turn on the other part.

INTERFERENCE FITS.—There are five classes of fits within the interference type. They are all fits that require force to assemble or disassemble parts. These fits are often called force fits and in certain classes of fits they are referred to as shrink fits. Using the same basic diameter as an example, the class 1 fit ranges from an interference allowance of -0.0006 to -0.0018 inch and a class 5 fit ranges from an interference allowance of -0.0032 to -0.0062 inch. The class 5 fit is normally considered to be a shrink fit class because of the large amounts of interference allowance required.

A shrink fit requires that the part with the external diameter be chilled or that the part with the internal diameter be heated. You can chill a part by placing it in a freezer, packing it in dry ice, spraying it with CO_2 (do not use a CO_2 bottle from a fire station) or by submerging it in liquid nitrogen. All of these methods except the freezer are potentially dangerous, especially the liquid nitrogen and should NOT be used until all applicable safety precautions have been reviewed and implemented. When a part is chilled, it actually shrinks in size a certain amount depending on the type of material, design, chilling medium, and length of time of exposure to the chilling medium. You can heat a part by using an oxyacetylene torch, a heat-treating oven, electrical strip heaters or by submerging it in a heated liquid. As with chilling, all applicable safety precautions must be observed. When a part is heated, it expands in size, allowing easier assembly. All materials expand a different amount per degree of temperature increased. This is called the coefficient of expansion of a metal. Most handbooks for machinists include a chart of the factors and explain their use. It is important that you calculate this information to determine the maximum temperature increase required to expand the part the amount of the shrinkage allowance plus enough clearance to allow assemble. Overheating a part can cause permanent damage and produce so much expansion that assembly becomes difficult.

A general rule of thumb for determining the amount of interference allowance on parts requiring a force or shrink fit is to allow approximately 0.0015 inch per inch of diameter of the internally bored part. There are many variables that will prohibit the use of this general rule. The amount of interference allowance recommended decreases as the diameter of the part increases. The dimensional difference between the inside and outside diameter (wall thickness) also has an effect on the interference allowance. A part that has large inside and outside diameters and a relatively thin wall thickness will split if installed with an excessive interference allowance. All of these variables must be considered before you select a fit when there are no blueprints or other dimensional references available.

Hydraulic and Arbor Presses

Hydraulic and arbor presses are used in many Navy machine shops. They are used to force broaches through parts, assemble and disassemble equipment with force fitted parts, and many other shop projects.

Arbor presses are usually bench mounted with a gear and gear rack arrangement. They are used for light pressing jobs, such as pressing arbors or mandrels into a part for machining or forcing a small broach through a part.

Hydraulic presses can be either vertical or horizontal, although the vertical design is probably more common and versatile. The pressure that a hydraulic press can generate ranges from about 10 to 100 tons in most of the Navy machine shops. The pressure can be exerted by either a manually operated pump or an electrohydraulic pump.

Regardless of the type of press equipment used, be sure that it is operated correctly. The only method for determining the amount of pressure exerted by a hydraulic press is by watching the pressure gage. A part being pressed can reach the breaking point without any visible indication that too much pressure is being applied. When using the press, you must consider the interference allowance between mating parts; corrosion and marred edges; and overlooked fastening devices, such as pins, setscrews, and retainer rings.

To prevent damage to the work, observe the following precautions when using the hydraulic press:

- Ensure that the work is adequately supported.

- Place the ram in contact with the work by hand, so that the work is positioned accurately in alignment with the ram.

- Use a piece of brass or other material (preferably slightly softer than the workpiece) between the face of the ram and the work to prevent mutilation to the surface of the workpiece.

- Watch the pressure gage. You cannot determine the pressure exerted by "feel." If excessive pressure is required, release the pressure and doublecheck the work to find the cause.

- When pressing parts together, use a lubricant between the mating parts to prevent seizing.

Information concerning the pressure required to force fit two mating parts together is available in most handbooks for machinists. The distance that the parts must be pressed usually increases the required pressures, and increased interference allowance require greater pressures. As a guideline for force-fitting a cylindrical shaft, the maximum pressure, in tons, should not exceed 7 to 10 times the shaft's diameter in inches.

Oxyacetylene Equipment

As a Machinery Repairman, you may have to use an oxyacetylene torch to heat parts to expand them enough to permit assembly or disassembly. This should be done with great care, and only with proper supervision. The operation of the oxyacetylene, torch is used in heating parts only is explained in this chapter with safety precautions which must be observed when using the torch and related equipment.

Oxyacetylene equipment consists of a cylinder of acetylene, a cylinder of oxygen, two regulators, two lengths of hose with fittings, a welding torch with tips, and either a cutting attachment of a separate cutting torch. Accessories include a spark lighter to light the torch; an apparatus wrench to fit the various connections on regulators, cylinders, and torches; goggles with filter lenses, for eye protection; and gloves for protection of the hands. Flame-resistant clothing is worn when necessary.

Acetylene (chemical formula C_2H_2) is a fuel gas made up of carbon and hydrogen. When burned with oxygen, acetylene produces a very hot flame having a temperature between 5700° and 6300°F. Acetylene gas is colorless, but has a distinct, easily recognized odor. The acetylene used onboard ship is usually taken from compressed gas cylinders.

Oxygen is a colorless, tasteless, odorless gas which is slightly heavier than air. Oxygen will not burn by itself, but it will support combustion when combined with other gases. You must be extremely careful to ensure that compressed oxygen does not become contaminated with hydrogen or hydrocarbon gases or liquids, unless the oxygen is controlled by such means as the mixing chamber of a torch. A highly explosive mixture will be formed if uncontrolled compressed oxygen becomes contaminated. Oxygen should NEVER come in contact with oil or grease.

The gas pressure in a cylinder must be reduced to a suitable working pressure before it can be used. This pressure reduction is accomplished by a LC REGULATOR or reducing valve. Regulators that control the flow of gas from the cylinder are either the single-stage or the double-stage type. Single-stage regulators reduce the pressure of the gas in one step; two-stage regulators do the same job in two steps, or stages. Less adjustment is generally necessary when two-stage regulators are used.

The hose connection between the torch and the regulators is strong, nonporous, and sufficiently flexible and light to make torch movements easy. The hose is made to withstand high, internal pressures, and the rubber from which it is made is specially treated to remove sulfur to avoid the danger of spontaneous combustion. Welding hose is available in various sizes, depending upon the size of work for which it is intended. Hose used for light work is 3/16 or 1/4 inch in diameter, and it has one or two plies of fabric. For heavy duty welding and cutting operations, hose with an inside diameter of 1/4 or 5/16 inch and three to five plies of fabric is used. Single hose comes in lengths of 12 1/2 feet to 25 feet. Some manufacturers make a double hose which conforms to the same general specifications. The hoses used for acetylene and oxygen are the same in grade but they differ in color and have different type of threads on the hose fittings. The oxygen hose is GREEN and the acetylene hose is RED. The oxygen hose has right hand threads and the acetylene hose has left hand threads for added protection against mixing the hose up during connection.

The oxyacetylene torch is used to mix oxygen and acetylene gas in the proper proportions and to control the volume of these gases burned at the torch tip. Torches have two needle valves, one for adjusting the flow of oxygen and the other for adjusting the flow of acetylene. In addition, they have a handle (body), two tubes (one for oxygen and one for acetylene), a mixing head, and a tip. Torch tips are made from a special copper alloy which dissipates heat (less than 60% copper) and are available in different sizes to handle a wide range of plate thicknesses.

Torch tips and mixers made by different manufacturers, differ in design. Some makes of torches have an individual mixing head or mixer for each size of tip. Other makes have only one mixer for several tip sizes. Tips come in various types. Some are one-piece, hard copper tips. Others are two-piece tips that include an extension tube to make connection between the tip and the mixing head. When used with an extension tube, removable tips are made of hard copper, brass, or bronze. Tip sizes are designated by numbers, and each manufacturer has its own arrangement for classifying them. Tips have different hole diameters.

No matter what type or size tip you select, the tip must be kept clean. Quite often the orifice becomes clogged. When this happens, the flame will not burn properly. Inspect the tip before you use it. If the passage is obstructed, you can clean it with wire tip cleaners of the proper diameter, or with soft copper wire. Tips should not be cleaned with machinist's drills or other sharp instruments.

Each different type of torch and tip size requires a specific working pressure to operate properly and safely. These pressures are set by adjusting the regular gages to the setting prescribed by charts provided by the manufacturer.

PROCEDURE FOR SETTING UP OXYACETYLENE EQUIPMENT.—Take the following steps in setting up oxyacetylene equipment:

1. Secure the cylinders so that they cannot be upset. Remove the protective caps.

2. Crack the cylinder valves slightly to blow out any dirt that may be in the valves. Close the valves and wipe the connections with a clean cloth.

3. Connect the acetylene pressure regulator to the acetylene cylinder and the oxygen pressure regulator to the oxygen cylinder. Using the appropriate wrench provided with the equipment tighten the connecting nuts.

4. Connect the red hose to the acetylene regulator and the green hose to the oxygen regulator. Tighten the connecting nuts enough to prevent leakage.

5. Back off on the regulator screws, and then open the cylinder valves slowly. Open the acetylene valve 1/4 to 1/2 turn. This will allow an adequate flow of acetylene and the valve can be turned off quickly in an emergency. (NEVER open the acetylene cylinder valve more than 1 1/2 turns.) Open the oxygen cylinder valve all the way to eliminate leakage around the stem. (Oxygen valves are double seated or have diaphragms to prevent leakage when open.) Read the high-pressure gage to check the pressure of each cylinder.

6. Blow out the oxygen hose by turning the regulator screw in and then back out again. If it is necessary to blow out the acetylene hose, do it ONLY in a well-ventilated place that is free from sparks, flames, or other possible sources of ignition.

7. Connect the hoses to the torch. Connect the red acetylene hose to the connection gland that has the needle valve marked AC or ACET.

Connect the green oxygen hose to the connection gland that has the needle valve marked OX. Test all hose connections for leaks by turning both regulator screws IN, while the needle valves are closed. Then turn the regulator screws OUT, and drain the hose by opening the needle valves.

8. Adjust the tip. Screw the tip into the mixing head and assemble in the torch body. Tighten by hand and adjust to the proper angle. Secure this adjustment by tightening with the wrench provided with the torch.

9. Adjust the working pressures. Adjust the acetylene pressure by opening the acetylene torch needle valve and turning the regulator screw to the right. Adjust the acetylene regulator to the required working pressure for the particular tip size. (Acetylene pressure should NEVER exceed 15 psig.)

10. Light and adjust the flame. Open the acetylene needle valve on the torch and light the acetylene with a spark lighter. Keep your hand out of the way. Adjust the acetylene valve until the flame just leaves the tip face. Open and adjust the oxygen valve until you get the proper neutral flame. Notice that the pure acetylene flame which just leaves the tip face is drawn back to the tip face when the oxygen is turned on.

PROCEDURE FOR ADJUSTING THE FLAME.–A pure acetylene flame is long and bushy and has a yellowish color. It is burned by the oxygen in the air, which is not sufficient to burn the acetylene completely; therefore, the flame is smoky, producing a soot of fine, unburned carbon. The pure acetylene flame is unsuitable for use. When the oxygen valve is opened, the mixed gases burn in contact with the tip face. The flame changes to a bluish-white color and forms a bright inner cone surrounded by an outer flame envelope. The inner cone develops the high temperature required.

The type of flame commonly used for heating parts is a neutral flame. The neutral flame is produced by burning one part of oxygen to one part of acetylene. Together with the oxygen in the air, it produces complete combustion of the acetylene. The luminous white cone is well-defined and there is no greenish tinge of acetylene at its tip, nor is there an excess of oxygen. A neutral flame is obtained by gradually opening the oxygen valve to shorten the acetylene flame until a clearly defined inner luminous cone is visible. This is the correct flame to use for many metals. The temperature at the tip of the inner cone is about 5900°F, while at the extreme end of the outer cone it is only about 2300°F. This gives you a chance to exercise some temperature control by moving the torch closer or farther from the work.

EXTINGUISHING THE OXYACETYLENE FLAME.–To extinguish the oxyacetylene flame and to secure equipment after completing a job, or when work is to be interrupted temporarily, you should take the following steps:

1. Close the acetylene needle valve first; this extenguishes the flame and prevents flashback. (Flashback is discussed later.) Then close the oxygen needle valve.

2. Close both oxygen and acetylene cylinder valves. Leave the oxygen and acetylene regulators open temporarily.

3. Open the acetylene needle valve on the torch and allow gas in the hose to escape for 5 to 15 seconds. Do NOT allow gas to escape into a small or closed compartment. Close the acetylene needle valve.

4. Open the oxygen needle valve on the torch. Allow gas in the hose to escape for 5 to 15 seconds. Close the valve.

5. Close both oxygen and acetylene cylinder regulators by backing out the adjusting screws until they are loose.

Follow the foregoing procedure whenever work is interrupted for an indefinite period. If work is to stop for only a few minutes, securing cylinder valves and draining the hose is not necessary. However, for any indefinite work stoppage, follow the entire extinguishing and securing procedure. For overnight work stoppage in areas other than the shop, it is a good idea to remove pressure regulators and torch from the system and to double check the cylinder valves to make sure that they are closed securely.

SAFETY: OXYACETYLENE EQUIPMENT

When you are heating with oxyacetylene equipment, certain safety precautions must be observed to protect personnel and equipment from injury by fire or explosion. The precautions which follow apply specifically to oxyacetylene work.

- Use only approved apparatus that has been examined and tested for safety.

- When cylinders are in use, keep them far enough away from the actual heating area so they will not be reached by the flame or sparks from the object being heated.

- NEVER interchange hoses, regulators, or other apparatus intended for oxygen with those intended for acetylene.

- Keep valves closed on empty cylinders.

- Do NOT stand in front of cylinder valves while opening them.

- When a special wrench is required to open a cylinder valve, leave the wrench in position on the valve stem while the cylinder is being used so that the valve can be closed rapidly in an emergency.

- Always open cylinder valves slowly. (Do NOT open the acetylene cylinder valve more than 1 1/2 turns.)

- Close cylinder valves before moving the cylinders.

- NEVER attempt to force unmatching or crossed threads on valve outlets, hose couplings, or torch valve inlets. The threads on oxygen regulator outlets, hose couplings, and torch valve inlets are right-handed; for acetylene, these threads are left-handed. The threads on acetylene cylinder valve outlets are right-handed, but have a pitch that is different from the pitch of the threads on the oxygen cylinder valve outlets. If the threads do not match, the connections are mixed.

- Always use the correct tip or nozzle and the correct pressure for the particular work involved. This information should be taken from tables or worksheets supplied with the equipment.

- Do NOT allow acetylene and oxygen to accumulate in confined spaces. Such a mixture is highly explosive.

- Keep a clear space between the cylinder and the work so that the cylinder valves may be reached quickly and easily if necessary.

- When lighting the torch, use friction lighters, stationary pilot flames, or some other suitable source of ignition. The use of matches may cause serious hand burns. Do NOT light a torch from hot metal. When lighting the torch, open the acetylene valve first and ignite the gas with the oxygen valve closed. Do NOT allow unburned acetylene to escape into a small or closed compartment.

- When extinguishing the torch, close the acetylene valve first and then close the oxygen valve.

- Do NOT use lubricants that contain oil or grease on oxyacetylene equipment. OIL OR GREASE IN THE PRESENCE OF OXYGEN UNDER PRESSURE WILL IGNITE VIOLENTLY. Consequently, oxygen must not be permitted to come in contact with these materials in any way. Do NOT handle cylinders, valves, regulators, hose, or any other apparatus which uses oxygen under pressure with oily hands or gloves. Do NOT permit a jet of oxygen to strike an oily surface or oily clothes. NOTE: A suitable lubricant for oxyacetylene equipment is glycerin.

- NEVER use acetylene from cylinders without reducing the pressure through a suitable pressure reducing regulator. Acetylene working pressures in excess of 15 pounds per square inch must be avoided. Oxygen cylinder pressure must likewise be reduced to a suitable low working pressure; high pressure may burst the hose.

- Stow all cylinders carefully in accordance with prescribed procedures. Cylinders should be stowed in dry, well-ventilated, well-protected places away from heat and combustible materials. Do NOT stow oxygen cylinders in the same compartment with acetylene cylinders. All cylinders should be stowed in an upright position. If they are not stowed in an upright position, they should not be used until they have been allowed to stand upright for at least 2 hours.

Do not use the torch to heat material without first making certain that hot sparks or hot metal will not fall on the legs or feet of the operator, on the hose and cylinder, or on any flammable materials. A fire watch should be posted as required to prevent accidental fires.

Flameproof protective clothing and shaded goggles should be worn by both the operator of the equipment and anyone nearby to prevent serious burns to the skin or the eyes. A number 5 or 6 shaded lens should be sufficient for the heating operations performed by Machinery Repairmen.

These precautions are by no means all the safety precautions that pertain to oxyacetylene equipment, and they only supplement those specified by the manufacturer. Always read the manufacturer's manual and adhere to all precautions and procedures for the specific equipment you are going to be using.

Flashback and Backfire

A backfire or a flashback are two common problems encoutered in using an oxyacetylene torch.

Unless the system is thoroughly purged of air and all connections in the system are tight before the torch is ignited, the flame is likely to burn inside the torch instead of outside the tip. The difference between the two terms backfire and flashback is this: in a backfire, there is a momentary burning back of the flame into the torch tip; in a flashback, the flame burns in or beyond the torch mixing chamber. A backfire is characterized by a loud snap or pop as the flame goes out. A flashback is usually accompanied by a hissing or squealing sound. At the same time, the flame at the tip becomes smoky and sharp-pointed. When a flashback occurs, immediately shut off the torch oxygen valve, then close the acetylene valve.

A flashback indicates that something is radically wrong either with the torch or with the manner of handling it. A backfire is less serious. Usually the flame can be relighted without difficulty. If backfiring continues whenever the torch is relighted, check for these causes: overheated tip, gas working pressures greater than that recommended for the tip size being used, loose tip, or dirt on the torch tip seat. These same difficulties may be the cause of a flashback, except that the difficulty is present to a greater degree. For example, the torch head may be distorted or cracked.

In most instances, backfires and flashbacks result from carelessness. To avoid these difficulties, always make certain that (1) all connections in the system are clean and tight, (2) torch valves are closed (not open or merely loosely closed) when the equipment is stowed, (3) the oxygen and acetylene working pressures used are those recommended for the torch employed, and (4) the system is purged of air before using the apparatus. Purging the system of air is especially necessary when the hose and torch have been newly connected or a new cylinder is incorporated into the system.

PURGING THE OXYACETYLENE TORCH.–

1. Close torch valves tightly, then slowly open the cylinder valves.
2. Open the regulator slightly.
3. Open the torch acetylene valve and allow acetylene to escape for 5 to 15 seconds, depending on the length of the hose.
4. Close the acetylene valve.
5. Repeat the procedure on the oxygen side of the system.

After purging air from the system, light the torch as described previously.

FASTENING DEVICES

Component parts of machinery and equipment are held together by several types of fastening devices. The fastening devices commonly used by the Machinery Repairman are classified in three general groups: threads, keys, and pins.

The selection of the correct fastener (specified in blueprints, lists of material blocks, and technical manuals) and the use of an approved installation method are important factors in the efficiency and reliability of a piece of equipment. Improper use of fasteners will lead to equipment failures and possible personnel injuries.

Threaded Fastening Devices

Bolts, studs, nuts, capscrews, machine screws and setscrews are all either externally or internally threaded devices used to clamp or secure mating parts together. Each of the different types has a specific range of applications and is available in various sizes, designs and material specifications. The most common sizes evolve from the established diameters, threads per inch, and classes of fits as described in the Unified (UNC, UNF) and the American National (NC, NF) thread systems explained in chapter 9. The design of each of the types usually permits a wide range of general applications for any given fastener. However, some equipment requires specialization of fasteners to such a degree that only a single application can be made. The material specification for a certain application of a fastener is based on the function of the mating parts, stresses, and temperatures applied to the fasteners and on the elements to which the equipment is exposed, such as steam, saltwater and oil. Table 3-3 is a general guide for material usage and the different identifying markings found on fasteners.

BOLTS.—A bolt is an externally threaded fastener, with a threaded diameter of 1/4 inch, or larger, and either a square or hexagon shaped head. Bolts are designed to be inserted into holes slightly larger than the diameter of the fastener. A nut is attached to the threaded end, and the mating parts are clamped together. As a general rule, the width of the head across the flats is approximately 1 1/2 times the diameter of the threads. The percentage of the bolt that is threaded ranges from 2 times the threaded diameter plus 1/4 inch to a point just below the head, depending on the intended use. It is best to use a bolt that has an unthreaded diameter length slightly less than the combined thickness of the parts being mated. The overall length should allow a minimum of 1 full thread and a maximum of 10 threads (space permitting) to protrude above the nut after the assembly is completely torqued down. The class of fit normally found on the threads of bolts and the nuts used with them is a class 2A for the bolt and a class 2B for the nut. This class of fit permits an allowance so that the bolt and nut can be assembled without seizing or galling. Detailed information on the different classes of fit for threads is covered later in this manual.

STUDS.—A stud is an externally threaded fastener with threads on both ends. It can either be inserted through a clearance hole and secured by a nut on each end, or it can be used in an assembly where one part has a tapped hole and the second part has a clearance hole. In the latter case, the stud is screwed into the tapped hole and a nut is screwed on the other end of the stud. A stud may be a continuous threaded design with threads beginning at one end and running the entire length of the stud. Another type of stud has threads beginning at each end and an unthreaded diameter in the center of the stud. The unthreaded diameter may be the same size as the major diameter of the threads, or it may be recessed to provide clearance. A continuous threaded stud generally has a class 2A or 3A fit to allow relative ease in assembly. A stud with the center part unthreaded may have a different class of fit on each end. One end will have a class 2A or 3A fit. This is the end on which the nut is screwed. The end of the stud that screws into the tapped hole will have an interference fit that will require a torque wrench to install it. The interference fit is a class 5 fit

Table 3-3.—Specifications and Uses of Fasteners

MATERIAL	MATERIAL SPECS.	GRADE	CONDITION	MARKING ON FASTENER	INTENDED USE
CARBON STEEL	SAE 10XX SERIES STEEL WITH A MAXIMUM OF 0.55%	5	HEAT TREATED	3 EQUALLY SPACED RADIAL LINES	GENERAL USE
CARBON STEEL	CARBON	8	HEAT TREATED	6 EQUALLY SPACED RADIAL LINES	GENERAL USE
ALLOY STEEL	SAE 4140 TO SAE 4145	B7	HEAT TREATED	B7	FOR USE UP TO 775°F WITH GRADE 2H AND GRADE 4 NUTS
ALLOY STEEL	ASTM A 193	B16	HEAT TREATED	B16	FOR USE UP TO 1000°F WITH GRADE 4 NUT
CORROSION RESISTANT STEEL	FED. STD. 66	303	ANNEALED	303	FOR USE WHERE LOW MAGNETIC AND CORROSION RESISTANT PROPERTIES ARE REQUIRED
CORROSION RESISTANT STEEL	FED. STD. 66	410T	HEAT TREATED	410	FOR USE WHERE LOW MAGNETIC AND CORROSION RESISTANT PROPERTIES ARE REQUIRED
NAVAL BRASS	QQ-B-637	482	–	482	FOR CONNECTING NON-FERROUS MATERIALS IN CONTACT WITH SALT WATER
SILICON BRONZE	QQ-C-591	651	–	651	FOR CONNECTING NON-FERROUS MATERIALS IN CONTACT WITH SALT WATER
NICKLE COPPER	QQ-N-281 CL. A&B	400	–	400	FOR CONNECTING FERROUS AND NON-FERROUS MATERIALS (EXCEPT ALUMINUM) IN CONTACT WITH SALT WATER
NICKLE COPPER ALUMINUM	QQ-N-286 CL.A	500	–	500	FOR CONNECTING FERROUS AND NON-FERROUS MATERIALS (EXCEPT ALUMINUM) IN CONTACT WITH SALT WATER
CARBON STEEL	SAE 10XX SERIES STEEL WITH A MAX. OF 0.55% CARBON	2H	HEAT TREATED	2H (NUTS ONLY)	FOR USE UP TO 775°F WITH GRADE B7 STUD OR BOLT
ALLOY STEEL	SAE 4140 to SAE 4145	4	HEAT TREATED	4 (NUTS ONLY)	FOR USE UP TO 1000°F WITH GRADE B16 AND B7 STUD OR BOLT

and is divided into several subdivisions to provide the correct fit for different materials and lengths of engagement. A stud of this type is screwed into the tapped hole the maximum distance possible without either jamming the end of the stud against the bottom of the hole or the shoulder of the unthreaded part of the stud against the top of the tapped hole. A small amount of lubricant approved for use in the temperature range in which the equipment is exposed should be applied to the threads. You will find the correct tolerances and torque required for each application in charts in most handbooks for machinists.

NUTS.—A nut is an internally threaded fastener with the same size threads as the external part to which it will be attached. They come in either a square or a hexagon shape and have standard widths and thicknesses based on the basic thread size. Any application of threaded fasteners that are subjected to working conditions, which could cause the nut to loosen through heat or vibration, usually has some method of locking the mating parts securely. Several methods are available to you: use different styles of lock washers, deform the area around the threads by staking or peening with a center punch, install setscrews, or use locknuts.

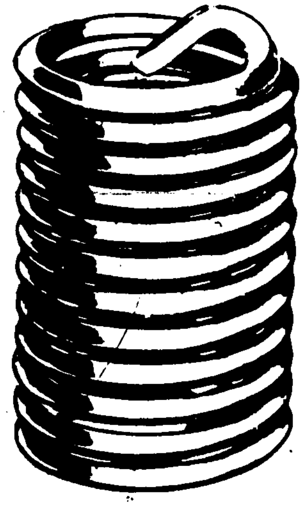
Locknuts in common use are of two types. One type applies pressure to the bolt or stud thread without permanent damage to the thread and is used when the nut must be removed frequently. Included in this type are jam nuts, a thin nut that goes under the regular nut; plastic angular ring and nylon plug insert nuts that use the resiliency of the plastic and nylon to create large frictional pressures on the bolt or stud; spring nuts that use springs of different types to apply pressure between the nut and the working surface; and spring beam nuts that have a slight taper in the upper portion of the nut with slots cut to form segments which permit expansion when the nut is screwed onto a bolt or stud. The other type of locknut deforms the threads on the bolt or stud and should be used only when removal is seldom required. This type includes (1) a distorted collar nut that has an oval shaped opening at the top and applies pressure when forced over the bolt or stud and (2) a distorted thread nut that has depressions in the face or threads of the nut.

MACHINE SCREWS AND CAPSCREWS.—Machine screws and capscrews are similar except for size range. Machine screws have diameters up to 3/4 inch (including size numbers from 0 to 12), while capscrews come in sizes above 1/4-inch diameter. Both machine screws and capscrews are available in several head shapes, such as flat, fillister, and hexagonal. These screwheads are slotted so that they can be tightened with a screwdriver.

SETSCREWS.—Setscrews are available in several different styles of heads including square, hexagon, slotted and the most common type, the recessed hexagon socket. The points on setscrews differ from the points on other threaded fasteners to permit a positive engagement with a prepared recess in the external surface on one of the mating parts it is being used on. Points are available in either a cone (90° point), a cup (recessed point), an oval, a flat or half-dog (a short, reduced diameter). The correct selection depends on whether the setscrew is intended to prevent slippage when holding a pulley or gear on a shaft or holding nonrotating parts in place. There is a definite relationship between the holding power and the diameter of a setscrew and between the number of setscrews required to transmit rotational movement of equipment rotating at any given rovolutions per minute and horsepower. If the equipment specifications do not specify this information, it may be obtained from most handbooks for machinists. Setscrews are normally made of hardened steel, although nonferrous (stainless steel, nickel-copper, bronze) setscrews are available for use when saltwater or corrosive liquids are involved.

Screw Thread Inserts

A screw thread insert (figure 3-39) is a helically wound coil of wire designed to screw into an internally threaded hole and receive a standard sized externally threaded fastener. A screw thread insert can be used to repair a threaded hole when the threads have been corroded or stripped away and to provide an



28.326

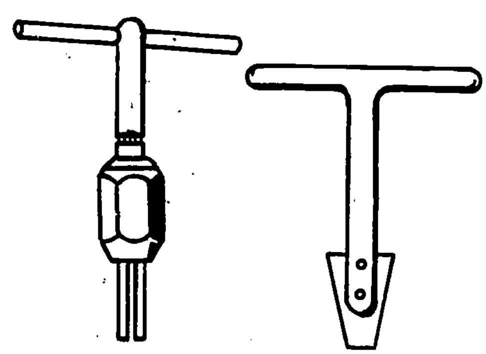
Figure 3-39.—Screw thread insert.

increased level of thread strength when the base metal of the part is aluminum, zinc, or other soft materials. Before using screw thread inserts for a repair job, carefully evaluate the feasibility of using this method. When you have no specific guidance, ask your supervisor for advice.

Screw thread inserts come in sizes up to 1 1/2-inch diameter in both American National and Unified, coarse and fine thread series. The overall length of an insert is based on a fractional multiple of its major diameter. A 1/2-inch screw thread insert is available in lengths of 1/2, 3/4, 1 inch, etc. Screw thread inserts are normally made from stainless steel, however a phosphor bronze and a nickel alloy insert are available by special order. A stainless steel insert should NOT be used in any application where the temperature exceeds 775°F or where a corrosive material such as acid or saltwater is involved.

There are several tools associated with the installation and removal of screw thread inserts that are essential if the job is to be done correctly. The most important tool is the tap used to thread the hole that the insert will be screwed into. These taps are oversized by specific amounts according to the size of the insert, so that after installation of the insert, a class 2B or 3B fit will have been made. The "H" limit numbers that refer to the pitch diameter tolerance, as previously explained in the section on hand taps, are marked on the taps. As an example of the amount of oversize involved, a tap required for a 1/2 - 13 UNC insert has a maximum major diameter of 0.604 inch. Because of the increase in the size of the hole required, it is important to ensure that there is sufficient material around the hole on the part to provide strength. A rule of thumb is that the minimum amount of material around the hole should equal the thread size of the insert, measured from the center of the hole. Using this rule, a 1/2 - 13 UNC insert will require a 1/2-inch distance from the center of the hole to the nearest edge of the part. The tap drill size for each of the taps is marked on the shank of the tap. The diameter of this drill will sometimes vary according to the material being tapped.

The next tool that you will use is an inserting tool (fig. 3-40). There are several different styles of inserting tools that are designed to be used for a specific range of insert sizes and within each of these styles are tools for each individual size of insert. All of the inserting tools have similar operating characteristics. Either slip the insert over or screw it onto the shank of the tool until the tang (the horizontal strip of metal shown at the top of the insert in figure 3-39) solidly engages the shoulder or recess on the end of the tool. Then install the



28.327

Figure 3-40.—Screw thread insert tools.

insert by turning the tool until the correct depth is reached. Remove the tool by reversing the direction of rotation.

After the insert has been properly installed, you must break off the tang to prevent any interference with the fastener that is to be screwed into the hole. A tang break-off tool is available for all insert sizes of 1/2 inch and below. The tang has a slight notch ground into it that will give way and break when struck with the force of the punch-type, tang break-off tool. On insert sizes over 1/2 inch use a long-nosed pair of pliers to move the tang back and forth until it breaks off.

When it is necessary to remove a previously installed screw thread insert, use an extracting tool (fig. 3-40). There are several different sized tools that cover a given range of insert sizes; be sure you select the correct sized tool. Insert the tool into the hole so that the blade contacts the top coil of the insert approximately 90° from the beginning of the insert coil. Then, lightly hit the tool to cause the blade to cut into the coil. Turn the tool counterclockwise until the insert is clear.

The steps involved in repairing a damaged threaded hole with a screw thread insert is as follows:

1. Determine the original threaded hole size. Select the correct standard sized screw thread insert with the length that best fits the application. Be sure the metal from which the insert is made is recommended for the particular application.

2. Select the correct tap for the insert to be installed. Some taps come in sets of a roughing and a finishing tap. You should ensure that both taps are used prior to an attempt to install the insert.

3. Select the correct sized drill based on the information on the shank of the tap or from charts normally supplied with the insert kits. Measure the part with a rule to determine if the preveiously referenced minimum distance from the hole to the edge of the part exists. With all involved tools and parts secured rigidly in place, drill the hole to a minimum depth that will permit full threads to be tapped a distance equaling or exceeding the length of the insert, not counting any spot-faced or countersunk area at the tap of the hole. Remove all chips from the hole.

4. Tap the hole. Use standard tapping procedures in this step. Lubricants should be used to improve the quality of the threads. When completed, inspect the threads to ensure that full threads have been cut to the required depth of the hole. Remove all chips.

5. Next, install the insert. If the hole being repaired is corroded badly, apply a small amount of preservative, such as zinc chromate, to the tapped threads immediately prior to installing the insert. Position the insert on the insert tool as required by the particular style being used. Turn the tool clockwise to install the insert. Continue to turn the tool until the insert is approximately 1/2 turn below the surface of the part. Remove the tool by turning it counterclockwise.

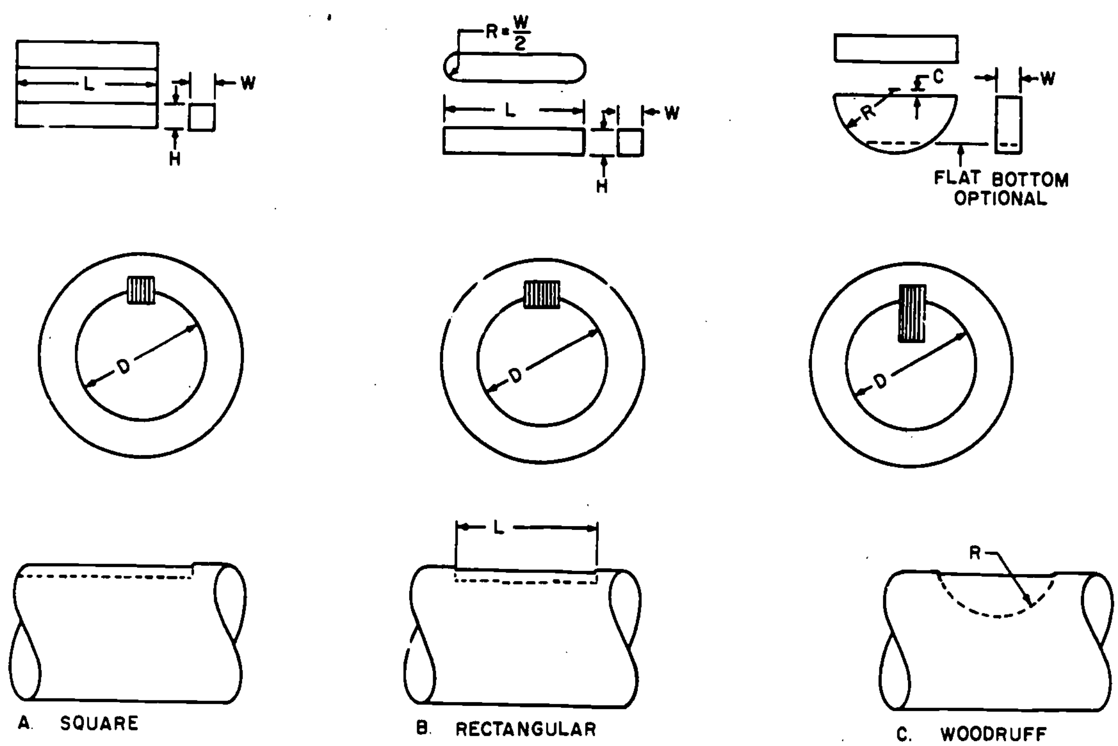
6. Use an approved antiseize compound when screwing the threaded bolt or stud into the insert. The use of similar metals such as a stainless insert and a stainless bolt should be avoided to prevent galling and seizing of the threads.

Keyseats and Keys

Keyseats are grooves cut along the axis of either the cylindrical surface of a shaft or the bored hole in a hub. Metal keys of various shapes are fitted into these grooves to provide a positive means for transmitting torque between a shaft and a hub. There are basically three types of keys: taper, parallel and Woodruff. The standard taper keys have a taper of 1/8 inch per foot and are either a plain taper or a gib head taper style key. Taper keys are not often found on marine equipment and will not be covered in this text. Parallel keys consist mainly of square and rectangular shaped keys. These are probably the most common types of keys that you will work with. A Woodruff key is a semicircular shaped key designed primarily to permit easy removal of pulleys from shafts. Keys are made from several different types of metal including medium carbon steel, nickel steel, nickel-copper alloy, stainless steel and several bronze alloys. Each different key style and material has a particular use for which it is best suited, depending on the forces and elements subjected to the equipment. Consult blueprints or other technical references when replacing a key to prevent selecting one that will not perform as required.

Square keys (fig. 3-41A) are recommended for applications where the shaft diameter is 6 1/2 inches and below, while rectangular keys (fig. 3-41B) are recommended for shaft diameters over 6 1/2 inches. Some applications may require that two keys be installed to drive equipment having a high torque value. The width and the height of a key are dependent on the diameter of the shaft that it will be used on, while the length of the key is based on the key's width. A chart giving some of the more common sizes of shafts and recommended key size combinations is provided in table 3-4.

Parallel keys (square and rectangular) and the keyseats machined to accept them are designed to provide assembly fits of three different classes. Each of the classes gives the recommended tolerance on both the key and the keyseat for the fit on the sides and the top and bottom of the keyed assembly. The top and bottom tolerances for the key and keyseat assemblies generally provide a range of fits from a metal to metal up to approximately 0.040-inch clearance (depending on the width of the key) for all three classes of fits. The side fit for a class 1 fit allows for a metal to metal 0.017-inch clearance fit. The amount of clearance increases as the width of the key increases. A class 2 fit allows for a side fit ranging from a 0.002-inch clearance to an interference fit of up to 0.003 inch. A class 3 fit allows only an interference fit for the sides of the key with individual applications determining the amount of interference involved. Complete charts showing the recommended key sizes for different shaft diameters and the allowable tolerance for each of the classes of fits are available in most handbooks for machinists.



28.34

Figure 3-41.—Types of keys and keyseats.

Table 3-4.—Key Size Versus Shaft Diameter

SHAFT DIAMETER		KEY SIZE			KEY LENGTH "L"	
		WIDTH "W"	HEIGHT "H"		MIN.	MAX.
FROM	TO		SQUARE	RECTANGULAR	4 X W	16 X W
7/8"	1 1/4"	1/4"	1/4"	3/16"	1"	4"
1 1/4"	1 3/8"	5/16"	5/16"	1/4"	1 1/4"	5"
1 3/8"	1 3/4"	3/8"	3/8"	1/4"	1 1/2"	6"
1 3/4"	2 1/4"	1/2"	1/2"	3/8"	2"	8"
4 1/2"	5 1/2"	1 1/4"	1 1/4"	7/8"	5"	20"
6 1/2"	7 1/2"	1 3/4"	1 3/4"	1 1/2"	7"	28"

Selective excerts extracted from "American Society of Mechanical Engineers" USAS B17.1-1967 Page 2, table 1

The ends of square or rectangular keys are often prepared with a radius equal to one-half the width as shown in the top illustration of figure 3-41B. This design permits a snug assembly fit when the machining on the keyseat was done with a conventional milling machine and an end mill cutter.

Woodruff keys (fig. 3-41C) are manufactured in various diameters and thicknesses. The circular side of the key is seated in a keyseat milled in the shaft with a cutter having the same radius and thickness as the key.

The size of a Woodruff key is designated by a system of numbers which represent the nominal key dimensions. The last two digits of the number indicate the diameter of the key in eighths of an inch, while the digit or digits preceding them indicate the width of the key in thirty-seconds of an inch. Thus, a number 404 key would be 4/8 or 1/2 inch in diameter and 4/32 or 1/8 inch wide, while a number 1012 key would be 12/8 or 1 1/2 inches in diameter and 10/32 or 5/16 inch wide.

To make proper assembly of keyed members, clearance is required between the top surface of the key and the key seat. This clearance is normally approximately 0.006 inch.

Positive fitting of the key in the keyseat is provided by making the key 0.0005 to 0.001 inch wider than the seat.

Information on the machining of keyseats for parallel and Woodruff keys is included in chapter 11.

Pins

The three pins commonly used in the machine shop are the dowel pin, the taper pin, and the cotter pin. The DOWEL PIN, which is made of machine-finished round stock, is used for aligning parts. It is used in such applications as pump housings. A hole in the housing matches with a hole in the end casing and a dowel pin is inserted to provide exact alignment. As this is an aligning pin the dowel must have a light drive fit. The TAPER PIN which has a 1/4-inch per foot taper is used to hold slow-speed, low-torque, rotor-shaft applications, such as hand-operated wheels and levers on machine tools. When taper pins are used, the hole must be drilled and then reamed with a taper pin reamer to obtain the correct fit. COTTER PINS are doubled sections of half-round metal stock which are used primarily to lock nuts in place on bolts. All pins come in a variety of standared sizes and lengths. Most machinist's handbooks give information on hole sizes and numbers for specific dimensions of pins.

Gaskets, Packing and Seals

Many of the repair jobs that you do will require installation of gaskets, packing, or seals to prevent leakage. Gaskets are used mainly for sealing fired type joints such as flanged pipe and valve joints and pump casings, while packing and seals are used for sealing joints where one part moves in relation to the other. All of these sealing devices are available in a wide range of diameters, thicknesses and classifications (grades) to provide suitable sealing of any system or equipment. A general knowledge of the different sealing materials is important however, the proper selection of a gasket, packing or other seal must never be based on general application guidelines or memory. The modern ships of today have systems that reach 1000°F in temperature and 2050 psi in pressure under normal operating conditions. A wrong selection can cause serious injury to personnel and major damage to equipment. The equipment's technical manual, allowance parts list, ship's plan on the appropriate PMS Maintenance Requirement Card are sources that can provide the exact specifications required for the sealing device.

A brief description of some of the more common types of gaskets, packing, and seals used in shipboard equipment and their general application is provided in the following paragraphs.

Gaskets

Spiral wound, metallic-asbestos gaskets are composed of alternate layers of dovetailed stainless steel ribbon and strips of asbestos spirally wound, ply upon ply, to the desired diameter. The gasket is then placed in a solid steel retainer ring to keep the gasket material intact, to assist in centering the gasket on the flange, and to act as a reinforcement to prevent blowouts. This type gasket is used on steam, boiler feedwater, fuel and lubricating oil systems. System pressures of 100 to 2050 psi and normal operating temperatures of 150° to 1000°F are within the range that these gaskets can effectively seal a system. Each application requires a specific gasket and substitutions should not be considered. When installing this gasket, tighten the fasteners evenly on diagonally opposite bolts until the gasket is compressed to the thickness required for the particular application.

Synthetic rubber and cloth inserted rubber gaskets are used on freshwater and seawater systems with pressures of 5[illegible] to [illegible]0 psi and temperatures of 150° to 250°F.

Gasoline and JP-5 systems require a gasket made from Buna-N and cork. The use of the wrong gasket material in these systems will result in deterioration of the gasket with contamination of the system and a hazardous situation when a leak develops.

Prior to installing any gasket, carefully inspect the surfaces of the mating parts for cuts or scratches that will prevent the proper sealing of the gasket. When any doubt exist, refinish the surface. You will find additional information on flange refinishing later in this manual.

Packing

The packing used to seal against leakage around equipment, such as valve stems on pump shafts, is available in many different material types, shapes and sizes. Specific recommendations on packing selection is best left to the appropriate technical document; however, there are some common errors made in packing selection and installation that are important to note. Packing that has a metallic or semimetallic base should not be used on a brass or bronze part. Parts that are softer than 250 BRINELL hardness should not be packed with a copper bearing packing. The surface condition of the valve stem or shaft and the stuffing box into which the packing is placed are important also. A surface that has pitts and scratches which could provide a path for leakage should be repaired. An out-of-round condition will cause excessive clearance between the packing and the rotating part. A relatively new type of packing called corrugated ribbon packing, which is intended for steam valves, requires very close control over the finishes, dimensions, and concentricity of the parts that contact it. Each part must be measured and checked carefully before this type packing can be used.

Seals

The types of seals you will work with most often are oil seals, mechanical seals and O-rings. Each type requires careful attention to the contact area and the installation procedures to ensure a good seal against leakage.

Oil seals are probably the simplest of the three types of seals to install. They consist of a metal cup or flange retainer, which press fits into a cylindrical bore, and a spring-loaded rubber or neoprene lip, which make contact with the shaft. The spring will cause the seal to maintain a firm contact with the shaft even if there is a small amount of shaft runout. The seal contact area on the shaft must be free of pitts, scratches and old wear patterns to operate as designed. When replacing a seal of this type, be particularly careful in selecting the proper seal as indicated by the equipment manufacturer. The type of fluid being sealed and the operating temperature are as important in correct seal selection as the dimensions of the seal.

Mechanical seals are considerably more difficult to install correctly. The majority of mechanical seals consist of one part that is sealed against the housing or seal retainer with a gasket or O-ring, while another part of the seal is attached to the shaft and is sealed by a rubber or neoprene bellows. Each of these two parts has a flat-faced seal that makes a rubbing contact when the shaft is turning. One of the flat-faced seals is spring-loaded to maintain a constant contact pressure when end play occurs in the equipment operation. The flat-faced seals may be made from carbon, alloy steel, ceramic, or several other materials. Regardless of the material used for these parts, they should be handled very carefully to avoid damage. The installation instructions provided by the seal or equipment manufacturer should be followed very closely to ensure the correct loading and proper functioning of the seal. Shaft runout, alignment, and end play (thrust) must be within the limitations prescribed for the equipment.

O-rings may be used as a static seal where no motion exists between the mating parts or as a dynamic seal where a reciprocating, an oscillating, or a rotary motion exist between the mating parts. O-rings are made from either synthetic or natural materials which have the capability of returning to their original shape and size after being deformed. The substance being sealed and the operating pressures and temperatures are very important factors in determining the exact O-ring to use in any given application. Preparation of the O-ring groove requires special care to ensure that the specified finish and dimensions are obtained. The annular or circular finish pattern (lay) produced by a lathe provides a surface that allows a more effective seal than one produced by an end mill cutter in a milling machine. A roughness valve of 32 microinches for a static seal and 15 microinches for a dynamic seal is generally acceptable for the O-ring groove. To achieve maximum effectiveness, an O-ring should not be stretched more than 5% beyond the designed dimension of the inside diameter after the O-ring is in position in the groove. This can be controlled only by accurate machining and measuring of the depth of the O-ring groove. Excessive width of the groove will allow the O-ring to roll or twist during installation and operation. Many applications require the use of backup rings which are placed on one or both sides of the O-ring to provide additional protection against O-ring distortion under pressure. The equipment specifications should be reviewed carefully to determine if a backup ring is required. An approved O-ring lubricant is essential during installation to prevent damage to the O-ring and to enhance the sealing effectiveness. The lubricant selected should be one that will not affect the O-ring material or contaminate the substance being sealed.

CHAPTER 4

METALS AND PLASTICS

A Machinery Repairman is expected to repair broken parts and to manufacture replacements in accordance with samples and blueprints. To choose the metals and plastics best suited for fabrication of replacement parts, you must have a knowledge of the physical and mechanical properties of materials and know the methods of identifying materials that are not clearly marked. For instance, stainless steel and nickel-copper are quite similar in appearance, but completely different in their mechanical properties and cannot be used inte-changeably. A themosetting plastic may look like a thermoplastic but the former is heat resistant, whereas the latter is highly flammable. Some of the properties of materials that an MR3 and MR2 must know are considered in this chapter.

PROPERTIES OF METALS

The physical properties of a metal determine its behavior under stress, heat, and exposure to chemically active substances. In practical application, behavior of a metal under these conditions determines its mechanical properties; indentation and rusting. The mechanical properties of a metal, therefore, are important considerations in selecting material for a specific job.

STRESS

Stress in a metal is its internal resistance to a change in shape (deformation) when an external load or force is applied to it. There are three different forms of stress to which a metal may be subjected. Tensile stress is a force that pulls a metal apart. Compression stress is a force that squeezes the metal. Shear stress is forces from opposite directions that work to separate the metal. When a piece of metal is bent, both tensile and compression stress are applied. The side of the metal where the force is applied undergoes compression stress as the metal is squeezed, while the opposite side is stretched under tensile stress. When a metal is subjected to a torsional load such as a sump shaft driven by an electric motor, all three forms of stress are applied to a certain degree.

STRAIN

Strain is the deformation or change in shape of a metal that results when a stress or load is applied. When the load is removed, the metal is no longer under a strain. The type of deformations which result when a metal is subjected to a stress will be similar to the form of stress applied.

STRENGTH

Strength is the property of a metal which enables it to resist strain (deformation) when a stress (load) is applied. The strength of a metal may be expressed in several different terms; the most commonly used term is tensile strength. Tensile strength is the maximum force required to pull metal apart. To find the tensile strength of a metal, divide the force required to pull the metal apart by the area in square inches of a prepared specimen.

The maximum force required to cause a metal to part is called its ultimate strength. To minimize the possibility of a part failing due to excessive working stresses, a factor of safety is designed into the part. A factor of safety means that the ultimate strength of the metal is two or more times greater than the working stresses to which the part will be subjected.

Another term used often to describe the strength of a metal is yield strength. The yield strength is determined during the same test that establishes the tensile strength. The yield strength is established when the metal specimen first begins to elongate (stretch) while pressure is gradually applied. A relationship between the tensile strength of metals and their hardness is often present. As the hardness of a metal is increased, the tensile strength is also increased and vice versa. Charts are available that provide these values for the more commonly used metals.

Some other terms that may be used to describe a metal's strength are compression strength, shear strength, and torsional strength. You will not see these terms often. However, in certain design applications, where stress would result in strains of one of these types being applied to a part, you would need to establish and use specific values in safety computations.

PLASTICITY

Plasticity is the ability of a metal to withstand extensive permanent deformation without breaking or rupturing. Modeling clay is an example of a highly plastic material, since it can be deformed extensively and permanently without rupturing. Metals with a high plasticity value will produce long, continuous chips when machined to a lathe.

ELASTICITY

Elasticity is the ability of a metal to return to its original size and shape after an applied force has been removed. Steel used to make springs is an example of this property.

DUCTILITY

Ductility is the ability of a metal to be permanently deformed by bending or by being stretched into wire form without breaking. To find the ductility of a metal, measure the percentage of elongation which results when the metal is stretched during the tensile strength test. Copper is an example of a very ductile metal.

MALLEABILITY

Malleability is the ability of a metal to be permanently deformed by a compression stress produced by hammering, stamping, or rolling the metal into thin sheets. Lead is a highly malleable metal.

BRITTLENESS

Brittleness is the tendency of a metal to break or crack with no prior deformation. Generally, the harder a metal, the more brittle it is, and vice versa. Pot metal and cast iron are examples of brittle metals.

TOUGHNESS

Toughness is the quality that enables a material to withstand shock, to endure stresses and to be deformed without breaking. A tough material is not easily separated or cut and can be bent first in one direction and then in the opposite without fracturing.

HARDNESS

Hardness of a metal is generally defined as its ability to resist indentation, abrasion or wear, and cutting. The degree of hardness of many metals may be either increased or decreased by being subjected to one or more heat treatment processes. In most cases, as the hardness of a steel is decreased, the toughness is increased.

HARDENABILITY

Hardenability is a measure of the depth (from the metal's surface toward its center) that a metal can be hardened by heat treatment. A metal that achieves a shallow depth of hardness and retains a relatively soft and tough core has a low hardenability value. The hardenability of some metals can be changed by the addition of certain alloys during the manufacturing process.

FATIGUE

Fatigue is the action which takes place in a metal after a repetition of stress. When a sample is broken in a tensile machine, a definite load is required to cause that fracture; however, the same material will fail under a much smaller load if that load is applied and removed many times. In this way, a shaft may break after months of use even though the weight of the load has not been changed. The pieces of such a part will not show any sign of deformation; but the mating areas of the section that fractured last will usually be quite coarse grained, while the mating areas of other sections of the break will show signs of having rubbed together for quite some time.

CORROSION RESISTANCE

Corrosion resistance is the ability of a material to withstand surface attack by the atmosphere, fluids, moisture, and acids. Some metals are highly resistant to practically all types of corrosive agents, others to some types of corrosive agents, and still others to only a very few types of corrosive substances. Some metals, however, can be made less susceptible to corrosive agents by coating or by alloying them with other metals that are corrosion resistant.

HEAT RESISTANCE

Heat resistance is that property of a steel or alloy that permits the steel or alloy to retain its properties at elevated temperatures. For example; red ÷ hardness in tungsten steel; high strength for chromium molybdenum steel; nondeforming qualities for austinitic stainless steel; malleability for forging steels. Tungsten steel (which even when red hot can be used to cut other metals) and chromium molybdenum steel (which is used for piping and valves in high temperature, high-pressure steam systems) are examples of heat resistant metals.

WELDABILITY

Weldability refers to the relative ease with which a metal can be welded. The weldability of a metal part depends on many different factors. The basic factor is the chemical composition or the elements that were added during the metal's manufacture. A steel with a low carbon content will be much easier to weld than a metal with a high carbon content. A low alloy steel that has a low hardenability value will lend itself more readily to welding than one with a high hardenability value. The welding procedure, such as gas or arc welding, also must be considered. The design of the part, its thickness, surface condition, prior heat treatments, and the method of fabrication of the metal also affect the weldability. Charts are available that provide guidelines concerning the weldability of a metal and the recommended welding procedure. The weldability of a metal should be considered an integral part of planning a job that requires the manufacture or repair of equipment components if any metal buildup or weld joint is involved.

MACHINABILITY

Machinability is the relative ease with which a metal can be machined. Table 4-1 lists the relative machinability rating of most of the metals used in machine shops. The machinability of each metal is given as a percentage of 100, with B1112, a resulphurized, force-machining steel, used as a basis for comparison. The higher rated metals can be cut using a higher cutting speed or surface feet per minute than those metals with lower percentage values.

There are several factors that affect the machinability of a metal: a variation in the amount or type of alloying element, the method used by the manufacturer to form the metal bar (physical conditon), any heat treatment which has changed the hardness, the type of cutting tool used (high-speed steel or carbide) and whether or not a cutting fluid is used. Information concerning some of these factors will be discussed later in this chapter and in chapter 8. Details of the AISI and SAE designations used in the chart are explained later in this chapter.

METALS

Metals are divided into two general types–ferrous and nonferrous. Ferrous metals

Table 4-1.—Machinability Rating

Plain Carbon Steels

SAE-AISI Numbers	BHN	Machinability %
B-1006	147	78
B-1010	147	78
C-1008	175	66
C-1010	172	65
C-1015	160	72
C-1016	148	78
C-1017	163	72
C-1019	146	78
C-1020	162	72
C-1022	147	78
C-1023	154	75
C-1025	162	72
C-1030	164	70
C-1035	162	70
C-1040	179	64
C-1043	178	64
C-1045	199	60
C-1046	203	57
C-1050	210	55
C-1054	217	53
C-1055	221	52
C-1059	222	52
C-1060	223	51
C-1064	224	50
C-1065	229	50
C-1069	231	48
C-1070	230	49
C-1075	238	48
C-1080	271	42
C-1085	269	42
C-1090	273	42
C-1095	274	42

Resulphurized Carbon Steels
Bessemer FCC

SAE-AISI Numbers	BHN	Machinability %
C-1106	150	79
C-1108	149	80
C-1109	152	81
C-1110	148	83
B-1111	131	94
B-1112	122	100
B-1113	101	132
C-1113	120	100
C-1115	150	81
C-1116	139	94
C-1118	139	91
C-1119	120	100
C-1125	152	81
C-1126	150	81
C-1137	169	72
C-1138	164	75
C-1140	171	72
C-1146	167	76
C-1151	180	70

Manganese Steels
Mn 1.75%

SAE-AISI Numbers	BHN	Machinability %
1320	210	57
1321	212	59
NE 1330	210	60
1335	211	60
NE 1340	216	57

Nickel Steels
Ni 3.50%

SAE-AISI Numbers	BHN	Machinability %
2317	185	66
2330	220	55
2335	242	51
2340	210	57
2345	231	51

Nickel Steels
Ni 5.00%

SAE-AISI Numbers	BHN	Machinability %
2512	210	51
2515	212	52
NE 2517	215	51

Nickel-Chrome Steels
Ni 1.25%
Cr 0.65% or 0.80%

SAE-AISI Numbers	BHN	Machinability %
3115	191	66
3120	190	66
3130	213	57
3135	225	53
3140	282	44
3145	192	64
3150	201	60

Nickel-Chrome Steels
Ni 3.50% Cr 1.55%

SAE-AISI Numbers	BHN	Machinability %
E 3310	241	51
E 3316	250	55

Molybdenum Steels
Mo 0.25%

SAE-AISI Numbers	BHN	Machinability %
4017	185	78
4023	182	78
4024	182	78
4027	212	66
4028	191	72
4032	184	76
4037	189	73
4042	198	70
4047	204	65
4053	261	53
4063	153	52

Chrome-Moly Steels
Cr 0.95% Mo 0.20%

SAE-AISI Numbers	BHN	Machinability %
4130	181	72
E 4132	190	72
4135	189	70
4137	209	65
E 4137	205	67
4140	212	62
4142	227	59
4145	221	60
4147	219	60
4150	242	59

Nickel-Chrome-Moly Steels
Ni 1.80% Cr 0.50% Mo 0.25%

SAE-AISI Numbers	BHN	Machinability %
4317	215	60
4320	201	63
E 4337	243	54
4340	240	57
E 4340	239	57

Nickel-Moly Steels
Ni 1.80% Mo 0.25%

SAE-AISI Numbers	BHN	Machinability %
4608	242	58
E 4617	201	66
4615	192	66
4620	198	64
X 4620	193	66
E 4620	202	64
4621	199	66
4640	198	66
E 4640	245	51

Nickel-Moly Steels
Ni 3.50% Mo 0.25%

SAE-AISI Numbers	BHN	Machinability %
4812	249	51
4815	256	51
4817	251	51
4820	248	53

Chrome Steels
Cr 0.30% or 0.60%

SAE-AISI Numbers	BHN	Machinability %
5045	188	70
5046	186	70

Chrome Steels
Cr 0.80%, 0.95% or 1.05%

SAE-AISI Numbers	BHN	Machinability %
5120	187	75
5130	241	57
5132	189	72
5135	188	72
5140	192	70
5145	210	65
5147	211	66
5150	215	64
5152	216	[illegible]

Carbon-Chrome Steels
C 1.00%
Cr 0.50%, 1.00% or 1.45%

SAE-AISI Numbers	BHN	Machinability %
E 50100	211	45
E 51100	221	40
E 52100	220	40

Chrome-Vanadium Steels
Cr 0.85% or 0.95%
V 0.10% or 0.15%

SAE-AISI Numbers	BHN	Machinability %
6102	202	57
6145	182	66
6150	192	60
6152	195	60

Nickel-Chrome-Moly Steels
Ni 0.55% Cr 0.50% Mo 0.20%

SAE-AISI Numbers	BHN	Machinability %
8617	182	66
8620	183	66
8622	185	65
8625	189	62
8627	188	64
8630	161	72
8635	165	70
8637	164	70
8640	172	66
8642	177	65
8645	182	64
8647	194	60
8650	195	60
8653	203	56
8655	205	57
8660	215	54

Nickel-Chrome-Moly Steels
Ni 0.55% Cr 0.50% Mo 0.259%

SAE-AISI Numbers	BHN	Machinability %
8719	175	67
8720	178	66
8735	171	70
8740	183	66
8742	185	64
8747	192	60
8750	194	60

Manganese-Silicon Steels
Mn 0.55% Si 2.00%

SAE-AISI Numbers	BHN	Machinability %
9255	122	54
9260	238	51
9262	235	49

Nickel-Chrome-Moly Steels
Ni 3.25% Cr 1.20% Mo 0.12%

SAE-AISI Numbers	BHN	Machinability %
E 9310	243	48
E 9315	238	50
E9317	239	49

Manganese-Nickel-Chrome-Moly Steels
Mn 1.00% Ni 0.45%
Cr 0.40% Mo 0.12%

SAE-AISI Numbers	BHN	Machinability %
9437	182	66
9440	183	66
9442	179	66
9445	181	64

Nickel-Chrome-Moly Steels
Ni 0.55% Cr 0.17% Mo 0.20%

SAE-AISI Numbers	BHN	Machinability %
9747	187	64
9763	215	54

Nickel-Chrome-Moly Steels
Ni 1.00% Cr 0.80% Mo 0.25%

SAE-AISI Numbers	BHN	Machinability %
9840	232	50
9845	238	49
9850	242	45

Stainless Steels

SAE-AISI Numbers	BHN	Machinability %
302		45
303*		60
304		45
308+		27
309+		28
314+		32
317+		29
321		36
330*		27
347		36
403		39
410		54
416*		72
420		57
420 F*		79
430		54
430 F**		91
440		37
440 A		45
440 B		42
440 C		40
440 F*		59

\+ Poorest Machining Properties.
* Fairly Good Machining-Contain Sulfur and Selenium.
** Best Machining Properties.

Cast Iron

SAE-AISI Numbers	BHN	Machinability %
Soft	130	81
Medium	168	64
Hard	243	47

Malleable Iron

SAE-AISI Numbers	BHN	Machinability %
Malleable Iron	115	106
Malleable Iron	135	[illegible]

Cast Steel

SAE-AISI Numbers	BHN	Machinability %
Cast Steel	121	85
Cast Steel	219	50
Cast Steel	245	44

are those whose major element is iron. Iron is the basis for all steels. Nonferrous metals are those whose major element is not iron, but they may contain a small amount of iron as an impurity.

FERROUS METALS

Iron ore, the basis of all ferrous metals, is converted to metal (pig iron) in a blast furnace. Alloying elements can be added later to the pig iron to obtain a wide variety of metals with different characteristics. The characteristics of metal can be further changed and improved by heat treatment and by hot or cold working.

Pig Iron

The product of the blast furnace is called pig iron. In early smelting practice, the arrangement of the sand molds into which the molten crude iron was drawn resembled groups of nursing pigs, hence the name.

Pig iron which is composed of approximately 93% iron, 3% to 5% carbon, and varying amounts of impurities, is seldom used directly as an industrial manufacturing material. It is, however, used as the basic ingredient in making cast iron, wrought iron, and steel.

Cast Iron

Cast iron is produced by resmelting a charge of pig iron and scrap iron in a furnace and removing some of the impurities from the molten metal by using various fluxing agents. There are many grades of cast iron as to strength and hardness. The quality depends upon the extent of refining, the amount of scrap iron used, and the method of casting and cooling the molten metal when it is drawn from the furnace. The higher the proportion of scrap iron, the lower the grade of cast iron. Cast iron has some degree of corrosion resistance and great compressive strength, but at best the metal is brittle and has a comparatively low tensile strength and, accordingly, has very limited use in marine service.

Wrought Iron

Wrought iron is a highly refined pure iron which has uniformly distributed particles of slag in its composition. Wrought iron is considerably softer than cast iron and has a fibrous internal structure, created by the rolling and squeezing given to it when it is being made. Like cast iron, wrought iron is fairly resistant to corrosion and fatigue. Wrought iron, because of these characteristics, is used extensively for low-pressure pipe, rivets, and nails.

Plain Carbon Steels

Pig iron is converted into steel by a process which separates and removes impurities from the molten iron by use of various catalytic agents and extremely high temperatures. During the refining process, practically all of the carbon originally present in the pig iron is burned out. In the final stages when higher carbon alloys are desired, measured amounts of carbon are added to the relatively pure liquid iron to produce carbon steel of a desired grade. The amount of carbon added controls the mechanical properties of the finished steel to a large extent, as will be pointed out in succeeding paragraphs. After the steel has been drawn from the furnace and allowed to solidify, it may be sent either to the stockpile or to shaping mills for rolling and forming into plates, billets, bars or structural shapes.

Plain steels that have small additions of sulfur (and sometimes phosphorous) are called free cutting steels. These steels have good machining characteristics and are used in applications similar to carbon steels. Addition of sulfur and phosphorous results in limiting its ability to be formed hot.

LOW CARBON STEEL (0.05% to 0.30%), usually referred to as mild steel, can be easily cut and bent and does not have great tensile strength, as compared with other steels. Low carbon steels which have less than 0.15% carbon are usually more difficult to machine than steel with a higher carbon content.

MEDIUM CARBON STEEL (0.30% to 0.60% carbon) is considerably stronger than low carbon steel. Heat treated machinery parts are made of this steel.

HIGH CARBON STEEL (0.60% to 1.50%) is used for many machine parts, handtools, and cutting tools, and is usually referred to as carbon tool steel. Cutting tools of high carbon steel should not be used when the cutting temperature will exceed 400°F.

Alloy Steels

The steels discussed thus far are true alloys of iron and carbon. When other elements are added to iron during the refining process, the resulting metal is called alloy steel. There are many types, classes, and grades of alloy steel.

Alloy steels usually contain several different alloying elements with each one contributing a different characteristic to the metal. Alloying elements can change the machinability, hardenability, weldability, corrosion resistance and the surface appearance of the metal. Knowledge of how each of the alloying elements affect a metal will allow you to more readily select the best metal for a given application and then to determine which, if any, heat treatment process should be applied to achieve the best mechanical properties. A few of the more common alloy steels and the effects of certain alloying elements upon the mechanical properties of steel are discussed briefly in the following paragraphs.

CHROMIUM.–Chromium is added to steel to increase the hardenability, corrosion resistance, toughness, and wear resistance The most common uses of chromium are in corrosion resistant steel (commonly called stainless steel) and high-speed cutting tools. Stainless steel is often used to manufacture parts which will be subjected to acids and saltwater and for such parts as ball bearings, shafts and valve stems in applications involving high-pressure and high temperature.

VANADIUM.–Vanadium is added in small quantities to steel to increase tensile strength, toughness, and wear resistance. It is most often combined with chromium in a steel and is used for crankshafts, axles, piston rods, springs, and other parts where high strength and fatigue resistance are required. Greater amounts of vanadium are added to high-speed steel cutting tools to prevent tempering when high temperatures are generated by the cutting action.

NICKEL.–Nickel is added to steel to increase corrosion resistance, strength, toughness, and wear resistance. Nickel is used in small amounts in the steel for armor plating of a ship due to its resistance to cracking when penetrated. Greater amounts of nickel are added to chromium to produce a metal which withstands severe working conditions. Crankshafts, rear axles, and other parts subjected to repeated shock are made from nickel chrome steel.

MOLYBDENUM.–Molybdenum is added to steel to increase toughness, hardenability, shock resistance and resistance to softening at high temperatures. Molybdenum steel is used for transmission gears, heavy duty shafts, and springs. Carbon molybdenum (CMo) and chrome molybdenum (CrMo) are two alloy steels with molybdenum added that are widely used in high temperature piping systems in Navy ships. Relatively large amounts of molybdenum are used to form some of the cutting tools used in the machine shop.

TUNGSTEN.–Tungsten is used primarily in high-speed steel or cemented carbide cutting tools. It is this alloy that gives the cutting tools their hard, wear resistant and heat resistant characteristics. Tungsten has the additional property of being air-hardening and allows tools to be hardened without using oil or water to cool the tool after heating.

NONFERROUS METALS

Copper, nickel, lead, zinc, tin, and aluminum are included among the nonferrous metals. You will find that these metals, and the many combinations in the form of alloys such as brass, bronze, copper-nickel, and so on, are used in large amounts in the construction and maintenance of Navy ships.

Copper Alloys

Copper is a metal which lends itself to a variety of uses. You will see it aboard ship in the form of wire, rod, bar, sheet, plate, and pipe. As a conductor of both heat and electricity, it ranks next to silver; it also offers a high resistance to saltwater corrosion.

Copper becomes hard when worked but can be softened easily by heating to a cherry red and then cooling. Its strength, however, decreases rapidly with temperatures above 400°F.

Pure copper is normally used in molded or shaped forms when machining is not required. Copper for normal shipboard use generally is alloyed with an element that provides good machinability characteristics.

BRASS.–Brass is an alloy of copper and zinc. Complex brasses are those containing additional alloying agents, such as aluminum, lead, iron, manganese, or phosphorus. Naval brass is a true brass containing about 60% copper, 39% zinc, and 1% tin added for corrosion resistance. It is used for propeller shafts, valve stems, and marine hardware.

Brass used by the Navy is classified as leaded or unleaded, meaning that small amounts of lead may or may not be used in the copper-zinc mixture. The addition of lead improves the machinability of brass.

BRONZE.–Bronze is primarily an alloy of copper and tin, although several other alloying elements are added to produce special bronze alloys. Aluminum, nickel, phosphorous, silicon and manganese are the most widely used alloys. Some of the more common alloys, their chemical analysis and some general uses are listed in the following paragraphs to give you an idea of how basic bronze is changed.

GUN METAL.–Gun metal, a copper-tin alloy, contains approximately 86%-89% copper (Cu), 7 1/2%-9% ting (Sn), 3%-5% zinc (Zn), 0.3% lead (Pb), 0.15% iron (Fe), 0.05% phosphorous (P) and 1% nickel (Ni). As you can see by the rather complex analysis of this bronze alloy, the term "copper-tin" is used only to designate the major alloy elements. Gun metal bronze is used for bearings, bushings, pump bodies, valves, impellers, and gears.

ALUMINUM BRONZE.–Aluminum bronze is actually a copper-aluminum alloy that does not contain any tin. It is made of 86% copper, 8 1/2%-99% aluminum (Al), 2 1/2%-4% iron and 1% of miscellaneous alloys. It is used for valve seats and stems, bearings, gears, propellers, and marine hardware.

COPPER-NICKEL.–Copper-nickel alloy is used extensively aboard ship because of its high resistance to the corrosive effect of saltwater. It is used in piping and tubing. In sheet form it is used to construct small storage tanks and hot water reservoirs. Copper-nickel alloy may contain either 70% copper and 30% nickel or 90% copper and 10% nickel. It has the general working characteristics of copper but must be worked cold.

These and the many other copper alloys commonly used by the Navy have certain physical and mechanical properties (imparted by the various alloying elements) which cause each different alloy to be more efficient for a given application than another would be. It is important to remember this when you go to the metal storage rack and select a bronze-looking metal without regard to the specific type. The part you make may fail prematurely in spite of the skill and attention to detail that you use to machine it.

Nickel Alloys

Nickel is a hard, malleable, and ductile metal. It is resistant to corrosion and therefore often is used as a coating on other metals. Combined with other metals in an alloy, it makes a tough strong alloy.

NICKEL-COPPER.–Nickel-copper alloys are stronger and harder than either nickel or copper. They have high resistance to corrosion and are strong enough to be substituted for steel when corrosion resistance is of primary importance. Probably the best known nickel-copper alloy is Monel metal (the trademark for a product of the International Nickel Company). Monel contains approximately 65% nickel, 30% copper, and a small percentage of iron, manganese, silicon, and cobalt. Monel is used for pump shafts and internal parts, valve seats and stems, and many other applications requiring both strength and corrosion resistance.

K-MONEL.–K-Monel, also a trademark, is essentially the same as Monel except that it contains about 3% aluminum and is harder and stronger than other grades of Monel. K-Monel stock is very difficult to machine; however, its machinability can be improved considerably by annealing the metal immediately before machining. K-Monel is used for the shaft sleeves on many pumps because of its resistance to the heat and rubbing action of the packing.

There are several other nickel alloys that you may find used in Navy equipment. INCONEL, INCONEL-X; H, S, R, and KR MONEL are a few of the more common alloys.

Aluminum Alloys

Aluminum is being used more and more in ship construction because of light weight, easy workability, good appearance, and other desirable properties. Pure aluminum is soft and not very strong. When alloying elements such as magnesium, copper, nickel, and silicon are added, however, a much stronger metal is produced.

Each of the aluminum alloys has properties developed specifically for a certain type of application. The hand aluminum alloys are easier to machine than the soft alloys and often are equal to low carbon steel in strength.

Zinc Alloys

Zinc is a comparatively soft, yet somewhat brittle metal. Its tensile strength is only slightly greater than that of aluminum. Because of its resistance to corrosion, zinc is used as a protective coating for less corrosion resistant metals, principally iron and steel. There are three methods of applying a zinc coating: (1) electroplating in a zinc-acid solution; (2) hot dipping, where the metal is dipped into a bath of molten zinc; (3) sherardizing, where zinc is reduced to a gaseous state and deposited on the base metal.

Pure zinc, having a strong anodic potential, is used to protect the hulls of steel ships against electrolysis caused by electric currents set up by saltwater between dissimilar metals. Zinc plates bolted on the hull, especially those near the propellers, decompose quite rapidly, but in doing so, greatly reduce localized pitting of the hull steel.

Another common use of zinc alloys is in making die cast parts such as carburetors, small engines blocks (lawn mower), and many small parts used in electrical appliances. This alloy is often mistakenly referred to as the copper and lead alloy called "pot-metal."

Tin Alloys

Pure tin is seldom used except as a coating for food containers, sheet steel and in some applications involving electroplating to build up the metal surfaces of some equipment (motor end bell bearing housings). Several different grades of tin solder are made by adding either lead or antimony. One of the primary uses of tin by the Navy is to make bearing babbitt. About 5% copper and 10% antimony are added to 85% tin to make this alloy. There are various grades of babbitt used in bearings and each grade may have additional alloying elements added to give the babbitt the properties required.

Lead Alloys

Lead is probably the heaviest metal with which you will work. A cubic foot of it weighs approximately 700 pounds. It is a grayish color and is amazingly pliable. It is obtainable in sheets and pigs. The sheets normally are wound around a rod and pieces can be cut off quite easily. One of the most common uses of lead is as an alloying element in soft solder.

DESIGNATIONS AND MARKINGS OF METALS

Knowledge of the standard designations of metals and the systems of marking metals used by the Navy and industry is necessary so that you can select the proper material for a specific job. There are several different numbering systems currently in use by different trade associations, societies, and producers of metals and alloys that you may find on blueprints and specifications of equipment that you will be required to repair. It is not uncommon to find several different designations which refer to a metal with the same chemical composition, or to find several identical designations which refer to metals with different chemical compositions. A book published by the Society of Automotive Engineers, Inc. (SAE), entitled *Unified Numbering System Of Metals and Alloys and Cross Index of Chemically Similar Specifications*, provides a clear and easily understood cross reference from the designation of one numbering system to other systems where a similar metal is involved. Some of the numbering systems included in the book that you may have a need to identify are:

Aluminum Association (AA)

American Iron and Steel Institute (AISI)

Society of Automotive Engineers (SAE)

Aerospace Materials Specifications (AMS)

American National Standards Institute (ANSI)

American Society of Mechanical Engineers (ASME)

American Society for Testing and Materials (ASTM)

Copper Development Association (CDA)

Military Specification (MIL-S-XXXX, MIL-N-XXXX)

Federal Specification (QQ-N-XX, QQ-S-XXX)

The unified numbering system, which is presented in the book, lists all the different designation for a metal with similar chemical specifications and assigns one number that identifies the metal. This system of numbering covers only the composition of the metal and not the condition, quality or form of the metal. Use of the unified numbering system is voluntary by the various metal producers, and it could be some time before any widespread use is evident. (Another publication that will be useful is NAVSEA 0900-LP-038-8010, Ship Metallic Material Comparison and Use Guide.)

The two major systems used for iron and steel are Society of Automotive Engineers (SAE) and the American Iron and Steel Institute (AISI). The aluminum Association method is used for aluminum; other nonferrous metals are designated by the percentage and types of elements in their composition. The Navy uses these methods of designation as a basis for marking metals so that they can be identified readily.

FERROUS METAL DESIGNATIONS

You should be familiar with the SAE and AISI systems of steel classification. These systems, which are in common use, have a four-digit or five-digit number to indicate the composition of the steel. The major difference between the two systems is that the AISI system normally uses a letter before the numbers to show the process used in making the steel. The letters used are as follows: B—Acid Bessemer carbon steel; C—Basic open-hearth or basic electric furnace carbon steel; and E—Electric furnace alloy steel. Example:

SAE		10	20
AISI	C	10	20
	↑ Basic Open Hearth Carbon Steel	↑ Plain Carbon Steel	↑ Carbon Content

A description of these numbering systems is provided in the following paragraphs.

The first digit normally indicates the basic type of steel The different groups are designated as follows:

1 – Carbon steel

2 – Nickel steel

3 – Nickel-chromium steel

4 – Molybdenum steel

5 – Chromium steel

6 – Chromium-vanadium steel

8 – Nickel-chrome-molybdenum steel

9 – Silicon-manganese steel

The second digit normally indicates a series within the group. The term "series" in this connection usually refers to the percentage of the major alloying element. Sometimes the second digit gives the actual percentage of the chief alloying element; in other cases, the second digit may indicate the relative position of the series in a group without reference to the actual percentage.

The third, fourth, and fifth digits indicate the average carbon content of the steel. The carbon content is expressed in points; for example: 2 points = 0.02%, 20 points = 0.20%, and 100 points = 1.00%. To make the various steels fit into this classification, it is sometimes necessary to vary the system slightly. However, such variations are easily understood if you understand the system. Let's take a few examples:

(1) SAE 1035: The first digit is 1, so this is a carbon steel. The second digit, 0, indicates that there is no other important alloying element; hence, this is a PLAIN carbon steel. The next two digits, 35, indicate that the AVERAGE percentage of carbon in steels of this series is 0.35%. There are also small amounts of other elements in this steel, such as manganese, phosphorus, and sulfur.

(2) SAE 1146: This is a resulfurized carbon steel (often called free cutting steel). The first digit indicates a carbon steel with an average manganese content of 1.00% (second digit) and an average carbon content of 0.46%. The amount of sulfur added to this steel ranges from 0.08% to 0.13%. These two elements, (manganese and sulfur) in this great a quantity, make this series of steel one of the easiest machining available.

(3) SAE 4017: The first digit, 4, indicates that this is a molybdenum steel. The second digit, 0, indicates that there is no other equally important alloying element; hence, this is a plain molybdenum steel. The last two digits, 17, indicate that the average carbon content is 0.17%.

Other series within the molybdenum steel group are indicated by the second digit. If the second digit is 1, the steel is chromium-molybdenum steel; if the second digit is 3, the steel is a nickel-chromium-molybdenum steel; if the second digit is 6, the steel is a nickel-molybdenum steel. In such cases, the second digit does not indicate the actual percentage of the alloying elements, other than molybdenum.

(4) SAE 51100: This number indicates a chromium steel (first digit) with approximately 1.0% chromium (second digit) and an average carbon content of 1.00% (last three digits). The actual chromium content of SAE 51100 steels may vary from 0.95% to 1.10%.

(5) SAE 52100: This number indicates a chromium steel (first digit) of a higher alloy series (second digit) than the SAE 51100 steel just described. Note, however, that in this case the second digit, 2, merely identifies the series but does NOT indicate the percentage of chromium. A 52100 steel will actually have from 1.30% to 1.60% chromium with an average carbon content of 1.00% (last three digits).

There used to be a 7000 series SAE steel, but under a new system it is no longer used. The current commonly used tool steels are classified by the American Iron and Steel Institute into seven major groups and each commonly accepted group or subgroup is assigned an alphabetical letter. Methods of quenching, applications, special characteristics, and steels for particular industries are considered in this type classification of tool steels as follows:

Group	Symbol and type
Water hardening	W
Shock resisting	S
Cold work	O–Oil hardening A–Medium alloy D–High carbon - high-chromium
Hot work	H–(H1 to H19 incl. chromium base, H20 to H39 incl. tungsten base, H40 to H59 incl. Molybdenum base)
High-speed	T–Tungsten base M–Molybdenum base
Special purpose	L–Low alloy F–Carbon tungsten
Mold steels	P

Navy blueprints and the drawings of equipment furnished in the manufacturers' technical manuals usually specify materials by Federal or Military specification numbers. For example, the coupling on a particular oil burner is identified as "cast steel, class B, MIL-S-15083." This particular cast steel does not have any other designation under the various other metal identification systems as there are no chemically similar castings. On the other hand, a valve stem which has a designated material of "MIL-S-862, class 410" (a chromium stainless steel) may be cross referenced to several other systems. Some of the chemically similar designations for "MIL-S-862, class 410" are as follows:

SAE = J405 (51410)

Federal Spec. = QQ-S-763 (410)

AISI = 410

ASTM = A176 (410)

ASM = 5504

ASME = SA194

NONFERROUS METAL DESIGNATIONS

Nonferrous metals are generally grouped according to the alloying elements. Examples of these groups are brass, bronze, copper-nickel, and nickel-copper. Specific designations of an alloy are described by the amounts and chemical symbols of the alloying elements. For example, a copper-nickel alloy might be described as copper-nickel, 70 Cu-30 Ni. The 70 Cu represents the percentage of copper, and the 30 Ni represents the percentage of nickel.

Common alloying elements and their chemical symbols are:

Aluminum	Al
Carbon	C
Chromium	Cr
Cobalt	Co
Copper	Cu
Iron	Fe
Lead	Pb
Manganese	Mn
Molybdenum	Mo
Nickel	Ni
Phosphorus	P
Silicon	Si
Sulphur	S
Tin	Sn
Titanium	Ti
Tungsten	W
Vanadium	V
Zinc	Zn

In addition to the type of designations previously described, a trade name (such as Monel or Inconel) is used sometimes to designate certain alloys.

The Aluminum Association uses a four-digit designation system similar to the SAE/AISI system described for steels. The numerals assigned with their meaning for the first digits of this system are:

Aluminum (99.00% minimum and greater)	1xxx
Major Alloying Element	
Copper	2xxx
Manganese	3xxx
Silicon	4xxx
Magnesium	5xxx
Magnesium and silicon	6xxx
Zinc	7xxx
Other element	8xxx

The first digit indicates the major alloying element and the second digit indicates alloy modifications or impurity limits. The last two digits identify the particular alloy or indicate the aluminum purity.

In the 1xxx group for 99.00% minimum aluminum, the last two digits indicate the minimum aluminum percentage to the right of the decimal point. The second digit indicates modifications in impurity limits. If the second digit in the designation is zero, it indicates that there is no special control on individual impurities; if it is 1 through 9, it indicates some special control of one or more individual impurities. As an example, 1030 indicates a 99.30% minimum aluminum without special control on individual impurities, and 1130, 1230, 1330, etc. indicate the same purity with special control of one or more individual impurities.

Designations 2 through 8 are aluminum alloys. In the 2xxx through 8xxx alloy groups, the second digit in the designation indicates any alloy modification. The last two of the four digits in the designation have no special significance but serve only to identify the different alloys in the group.

In addition to the four-digit alloy designation, a leter or letter/number is included as a temper designation. The temper designation follows the four-digit alloy number and is separated from it by a dash. As an example, 2024-T6 is an aluminum-copper alloy, solution heat treated, then artificially aged; T6 is the temper designation. The aluminum alloy temper designations and their meanings are:

F Fabricated
O Annealed recrystallized (wrought only)
H Strain hardened (wrought only)
- H1, plus one or more digits, strain hardened only
- H2, plus one or more digits, strain hardened then partially annealed
- H3, plus one or more digits, strain hardened then stabilized

W Solution heat treated—unstable temper
T Treated to produce stable tempers other than F, O, or H
- T2 Annealed (cast only)
- T3 Solution heat treated, then cold worked
- T4 Solution heat treated and naturally aged to a substantially stable condition
- T5 Artificially aged only
- T6 Solution heat treated, then artificially aged
- T7 Solution heat treated, then stabilized
- T8 Solution heat treated, cold worked, then artificially aged
- T9 Solution heat treated, artificially aged, then cold worked
- T10 Artificially aged then cold worked

Note that some temper designations apply only to wrought products, others to cast products, but most apply to both. A second digit may appear to the right of the mechanical treatment. This second digit indicates the degree of hardening: 2 is 1/4 hard, 4 is 1/2 hard, 6 is 3/4 hard, and 8 is full hard. For example, the alloy 5456-H32 is an aluminum/magnesium alloy, strain hardened then stabilized, and 1/4 hard.

STANDARD MARKING OF METALS

Metals used by the Navy are usually marked with the continuous identification marking system. This system will be explained in the following paragraphs. It is important to remember that no marking in itself will assure you that the correct metal is being used. Often, the markings provided by the metal producer will be worn off or cut off and you are left with a piece of metal that you are not sure about. Additional systems, such as separate storage areas or racks for different types of metal or etching on the metal with an electric etcher could save you time later on.

CONTINUOUS IDENTIFICATION MARKING

The continuous identification marking system, which is described in Federal Standards is a means for positive identification of metal products even after some portions have been used. In the continuous identification marking system, the markings appear at intervals of not more than 3 feet. Thus, if you cut off a piece of bar stock, the remaining portion will still carry the proper identification. Some metals, such as small tubing, coils of wire, and small bar stock cannot be marked readily by this method. On these items, tags with the required marking information are fastened to the metal.

The continuous identification marking is actually "printed" on the metals with a heavy ink that is almost like a paint.

The manufacturer is required to make these markings on materials before delivery. The marking intervals for various shapes and forms, are specified in the Federal Standards previously mentioned. Figure 4-1 shows the normal spacing and layout.

For metal products, the continuous identification marking must include (1) the producer's name or registered trademark and (2) the commercial designation of the material. In nonferrous metals the government specification for the material is often used. The producer's name or trademark shown is that of the producer who performs the final processing or finishing operation before the material is marketed. The commercial designation includes (1) a material designation such as an SAE number, an AISI number, or an ASTM (American Society of Testing Materials) specification and (2) a "physical condition" and quality designation—that is, the designation of temper or other physical condition approved by a nationally recognized technical society or industrial association such as the American Iron and Steel Institute. Some of the physical conditions and quality designations for various metal products are listed here:

CR . . . . . . cold rolled

CD . . . . . . cold drawn

HR . . . . . . hot rolled

AQ . . . . . . aircraft quality

CQ . . . . . . commercial quality

1/4H . . . . . quarter hard

1/2H . . . . . half hard

H . . . . hard

HTQ . . . . . high tensile quality

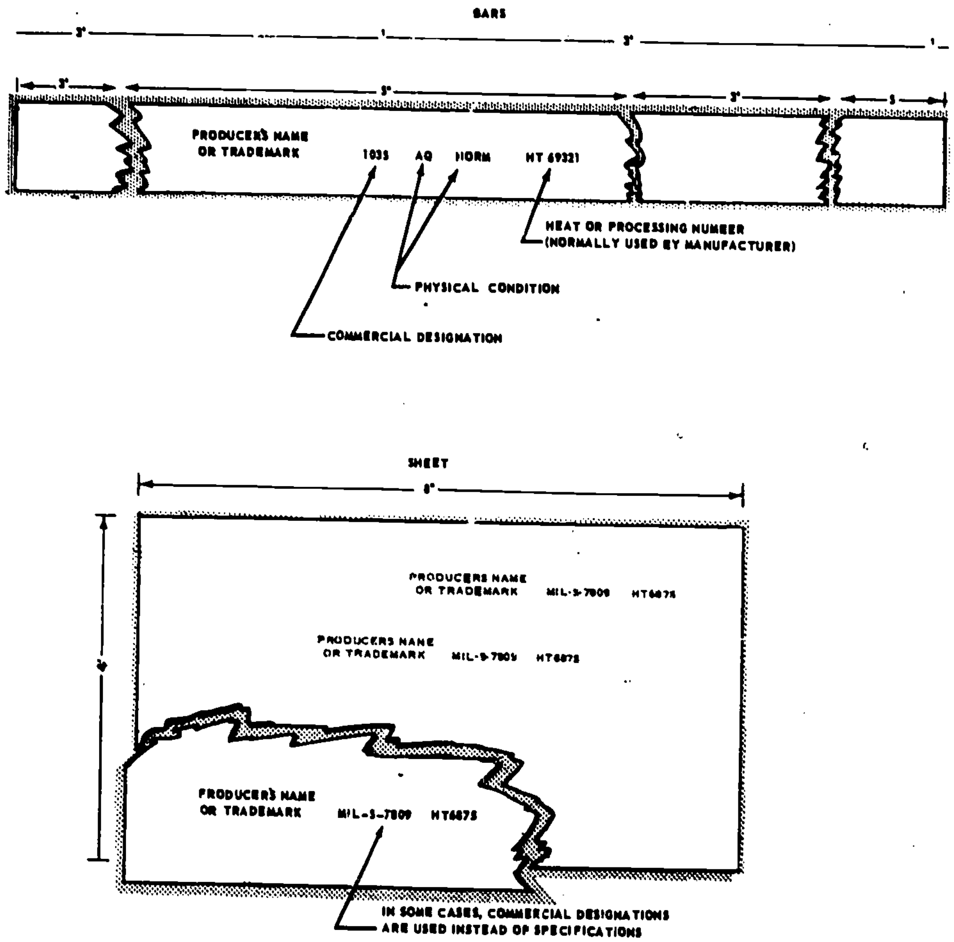
AR . . . . . . as rolled

HT . . . . . . heat treated

G . . . . ground

IDENTIFICATION OF METALS

The various base metals, such as iron, copper, lead, zinc, and aluminum have certain identifying characteristics—surface appearance and weight—by which persons who work with or handle these materials readily distinguish one from the other. There are, however, a number of



28.37

Figure 4-1.—Examples of continuous identification marking.

related alloys which resemble each other and their base metal so closely that they defy accurate identification by simple means.

There are other means of rapid identification of metals. These methods by no means provide positive identification and should not be used in critical situations where a specific metal is desired. Some of the methods that will be discussed here are magnet tests, chip tests, file tests, acid reaction tests, and spark tests. The latter two are the most commonly used by the Navy. Table 4-2 contains information related to surface appearance, magnetic reaction, lathe chip test, and file test. The acid test and the spark test are discussed in more detail in the next sections. When performing these tests, you should have a known sample of the desired material and make a comparison. When using these tests, you will need good lighting, a strong permanent magnet, and access to a lathe. A word of caution when performing these tests: DO NOT be satisfied with the results of only one test; use as many tests as possible so that you can increase the chances of making an accurate identification.

SPARK TEST

Spark testing is the identification of a metal by observing the color, size, and the shape of the

Table 4-2.—Rapid Identification of Metals

Metal	Surface Appearance or markings	Reaction to a Magnet	Lathe Chip test	Color of freshly filed surface
White cast iron	Dull gray	Strong	Short, crumbly chips	Silvery white
Gray cast iron	Dull gray	Strong	Short, crumbly chips	Light silvery gray
Aluminum	Light gray to white dull or brilliant	None	Easily cut, smooth long chips	White
Brass	Yellow to green or brown	None	Smooth long chips slightly brittle	Reddish yellow to yellowish white
Bronze	Red to brown	None	Short crumbly chips	Reddish yellow to yellowish white
Copper	Smooth; red brown to green (oxides)	None	Smooth long pliable chips	Bright copper color
Copper-nickel	Smooth; gray to yellow or yellowish green	None	Smooth, continuous chips	Bright silvery white
Lead	White to gray; smooth, velvety	None	Cut by knife, any shape chip	White
Nickel	Dark gray; smooth; sometimes green (oxides)	Medium	Cuts easily, smooth continuous chip	Bright silvery white
Nickel-copper	Dark gray, smooth	Very slight	Continuous chip; tough to cut	Light gray
Plain carbon steel	Dark gray; may be rusty	Strong	Varies depending upon carbon content	Bright silvery gray
Stainless steel (18-8) (25-20) "Note 1 below"	Dark gray; dull to brilliant; usually clean	None (faint if severely cold worked)	Varies depending upon heat treatment	Bright silvery gray
Zinc	Whitish blue, may be mottled	None	Easily cut; long stringy chips	White

1. Stainless steels that have less than 26 percent alloying elements react to magnet.

spark stream given off when the metal is held against a grinding wheel. This method of identification is adequate for most machine shop purposes. When the exact composition of a metal must be known, a chemical analysis must be made. Identification of metals by the spark test method requires considerable experience. To gain this experience, you will need to practice by comparing the spark stream of unknown specimens with that of sample specimens of known composition. It is the practice in many shops to maintain specimens of known composition for comparison with unknown samples.

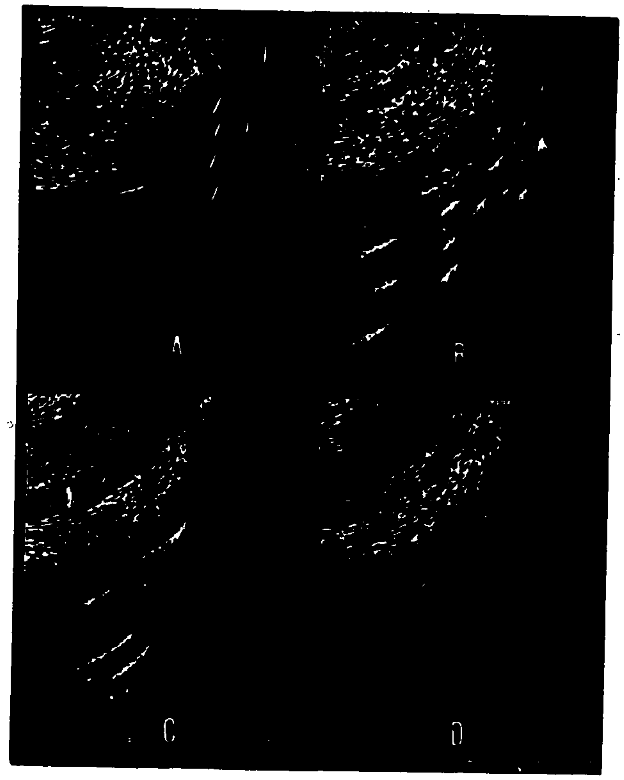
Proper lighting conditions are essential for good spark testing practice. The test should be performed in an area where there is enough light, but harsh or glaring light should be avoided. In many ships you may find that a spark test cabinet has been erected. Generally, these cabinets consist of a box mounted on the top of a workbench and have a dark painted interior. Inside the cabinet a bench grinder is mounted. Test specimens of known composition are contained in shelves at the end of the cabinet. Where possible, the testing area should be away from heavy drafts of air. Air drafts can change the tail of the spark stream and may result in improper identification of the sample.

Speed and pressure are factors that need to be considered in making the spark test. The faster the speed of the wheel, the larger and longer the spark stream will be. (Generally speaking, a suitable grinding wheel for spark testing is an 8-inch wheel turning 3600 rpm. This provides a surface speed of 7,537 feet per minute.) The pressure of the piece against the wheel has similar effects: the more pressure applied to the test piece, the larger and longer the spark stream. Hold the test piece lightly but firmly against the wheel with just enough pressure to prevent the piece from bouncing. Remember that you must apply the same amount of pressure to the test specimen as to the sample you are testing.

The grain size of the grinding wheel should be from 30 to 60 grains. It is very important to remember that the wheel must be clean at all times. A wheel loaded with particles of metal will give off a spark stream of the type of metal in the wheel, mixed with the spark stream of the metal being tested. This will result in serious confusion as to the true nature of the metal. The wheel must be dressed before spark testing and before each new test of a different metal.

The spark test is made by holding a sample of the material against a grinding wheel. The sparks given off, or the lack of sparks, assist in identifying the metal. The length of the spark stream, its color, and the type of sparks are the features for which you should look. There are four fundamental spark forms produced when a sample of metal is held against a power grinder. (See fig. 4-2.) Part A shows shafts, bud, break, and arrow. The arrow or spearhead is characteristic of molybdenum, a metallic element of the chromium group which resembles iron and is used for forming steel-like alloys with carbon. The swelling or buds in the spark line indicate nickel with molybdenum. Part B of figure 4-2 shows shafts and sprigs or sparklers which indicate a high carbon content. Part C shows shafts, forks, and sprigs which indicate a medium carbon content. Part D shows shafts and forks which indicate a low carbon content.

The greater the amount of carbon present in a steel, the greater the intensity of bursting that will take place in the spark stream. To understand the cause of the bursts, remember that while the spark is glowing and in contact with the oxygen of the air, the carbon present in the particle is burned to carbon dioxide (CO_2). As the solid carbon combines with oxygen to form CO_2 in the gaseous state, the increase in volume builds up a pressure that is relieved by an explosion of the particle. An examination of the small steel particles under a microscope when they are cold reveals a hollow sphere with one end completely blown away.



11.371X

Figure 4-2.—Fundamental spark forms.

Steels having the same carbon content but different alloying elements are not always easily identified because alloying elements affect the carrier lines, the bursts, or the forms of characteristic bursts in the spark picture. The effect of the alloying element may retard or accelerate the carbon spark or make the carrier line lighter or darker in color. Molybdenum, for example, appears as a detached, orange-colored, spearhead on the end of the carrier line. Nickel seems to suppress the effect of the carbon burst. But the nickel spark can be identified by tiny blocks of brilliant white light. Silicon suppresses the carbon burst even more than nickel. When silicon is present, the carrier line usually ends abruptly in a flash of white light.

To make the spark test, hold the piece of metal on the wheel in such a manner as to throw the spark stream about 12 inches at a right angle to your line of vision. You will need to spend a little time to discover at just what pressure you must hold the sample to get a stream of this length without reducing the speed of the grinder. It is important that you do not press too hard because the pressure will increase the temperature of the spark stream and the burst. It will also give the appearance of a higher carbon content than that of the metal actually being tested. After practicing to get the feel of correct pressure on the wheel until you are sure you have it, select a couple of samples of metal with widely varying characteristics; for example, low-carbon steel and high-carbon steel. Hold first one and then the other against the wheel, always being careful to strike the same portion of the wheel with each piece. With your eyes focused at a point about one-third the distance from the tail end of the stream of sparks, watching only those sparks which cross the line of vision, you will find that after a little while you will form a mental image of the individual spark. After you can fix the spark image in mind, you are ready to examine the whole spark picture.

Notice that the spark stream is long (about 70 inches normally) and that the volume is moderately large in low-carbon steel, while in high carbon steel the stream is shorter (about 55 inches) and large in volume. The few sparklers which may occur at any place in low carbon steel are forked, while in high carbon steel the sparklers are small and repeating and some of the shafts may be forked. Both will produce a white spark stream.

White cast iron produces a spark stream approximately 20 inches in length (see fig. 4-3). The volume of sparks is small with many small, repeating sparklers. The color of the spark stream close to the wheel is red, while the outer end of the stream is straw-colored.

Gray cast iron produces a stream of sparks about 25 inches in length. It is small in volume with fewer sparklers than from white cast iron. The sparklers are small and repeating. Part of the stream near the grinding wheel is red, and the outer end of the stream is straw-colored.

The malleable iron spark test will produce a spark stream about 30 inches in length. It is of moderate volume with many small, repeating sparklers toward the end of the stream. The entire stream is straw-colored.

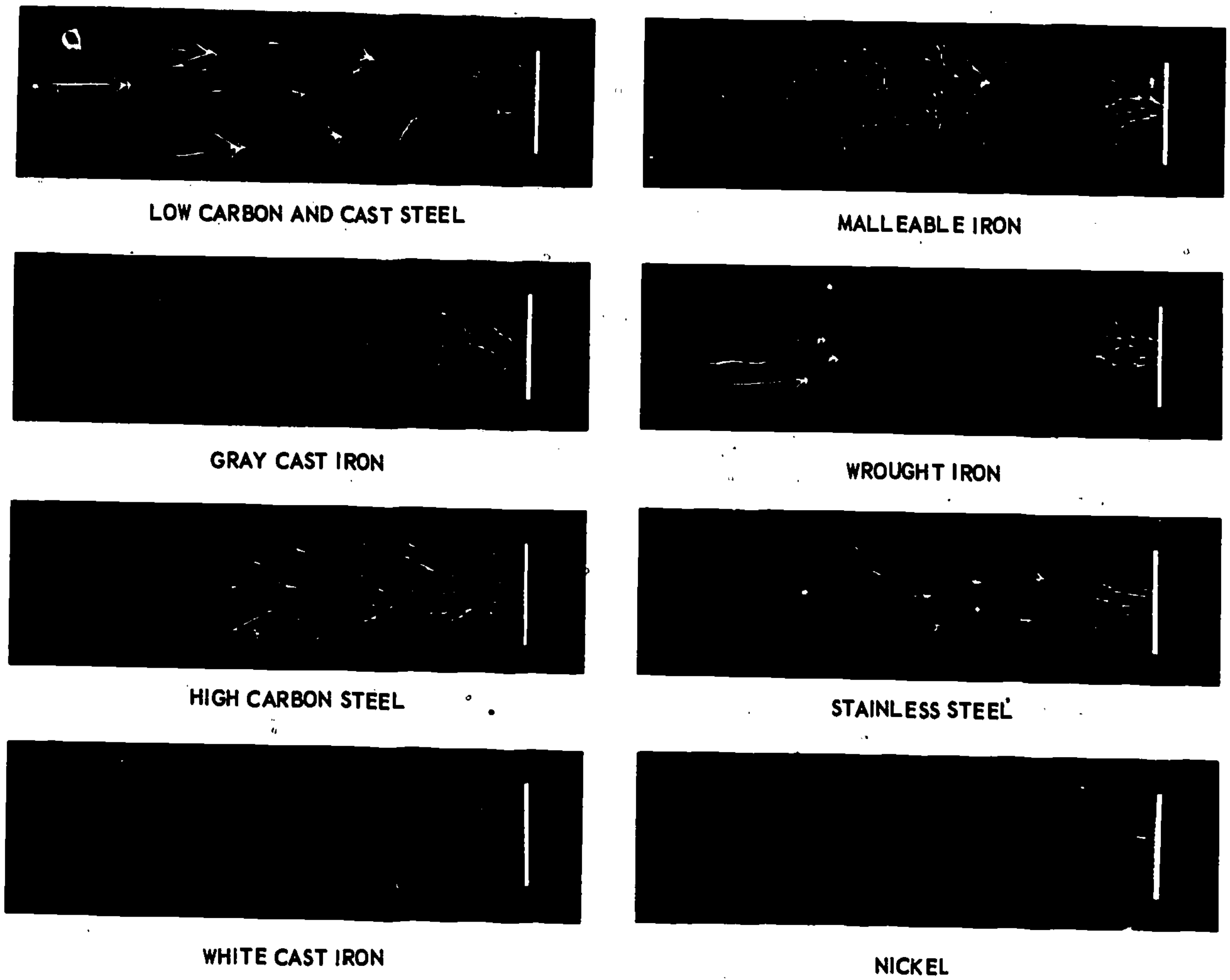
The wrought iron spark test produces a spark stream about 65 inches in length. The stream is of large volume with a few sparklers. The sparklers show up toward the end of the stream and are forked. The stream next to the grinding wheel is straw-colored, while the outer end of the stream is a bright red.

Stainless steel produces a spark stream approximately 50 inches in length, of moderate volume, and with few sparklers. The sparklers are forked. The stream next to the wheel is straw-colored, while at the end it is white.

Nickel produces a spark stream only about 10 inches in length. It is small in volume and orange in color. The sparks form wavy streaks with no sparklers.

Monel metal forms a spark stream almost identical to that of nickel and must be identified by other means. Copper, brass, bronze, and lead form no sparks on the grinding wheel, but they are easily identified by other means, such as color, appearance, and chip tests.

You will find the spark tests easy and convenient to make. They require no special equipment and are adaptable to most any



11.37X

Figure 4-3.—Spark pictures formed by common metals.

situation. Here again, experience is the best teacher.

ACID TEST

The nitric acid test is the most commonly used for metal identification by the Navy today; it is used only in noncritical situations. For rapid identification of metal, the nitric acid test is one of the easiest tests to use and requires no special training in chemistry to perform. It is most helpful in distinguishing between stainless steel, Monel, copper-nickel, and carbon steels. The following safety precautions must be observed in using or handling acids of any type.

- NEVER open more than one container of acid at one time.

- In mixing, always pour acid slowly into water. NEVER pour water into acid because an explosion is likely to occur.

- If any acid is spilled, dilute with plenty of water to weaken it so that it can be safely swabbed up and disposed of.

- If an acid is spilled on your skin, wash immediately with large quantities of water. Then wash with a solution of borax and water.

- Wear CLEAR-LENS safety goggles to ensure the detection of the reaction of metal to an acid test which may be evidenced by a color change, the formation of a deposit, or the development of a spot.

- Conduct tests in a well-ventilated area.

To perform the nitric acid test, place one or two drops of concentrated (full strength) nitric acid on a metal surface that has been cleaned by grinding or filing. Observe the resulting reaction (if any) for about 2 minutes. Then, add three or four drops of water, one drop at a time, and continue observing the reaction. If there is no reaction at all, the test material may be one of the stainless steels. A reaction that results in a brown-colored liquid indicates a plain carbon steel. A reaction producing a brown to black color indicates a gray cast iron or one of the alloy steels containing as its principal element either chromium, molybdenum, or vanadium. Nickel steel reacts to the nitric acid test by forming a brown to greenish-black liquid, while a steel containing tungsten reacts slowly to form a brown-colored liquid with a yellow sediment.

When nonferrous metals and alloys are subjected to the nitric acid test; instead of the brown-black colors that usually appear when ferrous metals are tested, various shades of green and blue appear as the material dissolves. Except with nickel and Monel, the reaction is vigorous. The reaction of nitric acid on nickel proceeds slowly, developing a pale green color. On Monel, the reaction takes place at about the same rate as on ferrous metals, but the characteristic color of the liquid is greenish-blue. Brass reacts vigorously with the test material changing to a green color. Tin bronze, aluminum bronze, and copper all react vigorously in the nitric acid test with the liquid changing to a blue-green color. Aluminum and magnesium alloys, lead, lead-silver, and lead-tin alloys are soluble in nitric acid, but the blue or green color is lacking.

From the information given thus far, it is easy to see that you will need considerable visual skill to identify the many different reactions of metals to nitric acid. There are acid test kits available containing several different solutions to identify the different metals. Some of the kits can identify between the different series of stainless steels (300, 400 series). Low alloy steels, nickel steels, and various bronze alloys are also identified quickly with these tests. A chemical laboratory is available in most large repair ships and shore repair facilities. The personnel assigned are also available to identify various metals in more critical situations or when a greater degree of accuracy is required on a repair job.

HEAT TREATMENT

Heat treatment is the operation or combination of operations, including heating and cooling of a metal in its solid state, to develop or enhance a particular desirable mechanical property, such as hardness, toughness, machinability, or uniformity of strength. The theory of heat treatment is based upon the effect that the rate of heating, degree of heat, and the rate of cooling have on the molecular structure of a metal.

There are several forms of heat treating. The forms commonly used for ferrous metals are: annealing, normalizing, hardening, tempering, and case-hardening. Detailed procedures for the various heat treatments of metals and the theories behind them are beyond the scope of this manual. However, since you will run across the terms from time to time and will probably perform some of the heat treatment processes under the supervision of an MR1 or MRC, we will discuss some of the general terminology.

ANNEALING

The chief purposes of annealing are (1) to relieve internal strains and (2) to make a metal soft enough for machining. Annealing is the process of heating to and holding at a suitable temperature and then cooling at a suitable rate, for such purposes as reducing hardness, improving machinability, facilitating cold working, producing a desired microstructure or obtaining desired mechanical, physical or other properties.

Besides rendering metal more workable, annealing can also be used to alter other physical properties, such as magnetism and electrical conductivity. Annealing is often used for softening nonferrous alloys and pure metals after they have been hardened by cold work. Some of these alloys require annealing operations which are different from those for steel.

For ferrous metals, the annealing method most commonly used, if a controlled atmosphere furnace is not available, is to place the metal in a cast iron box and cover it with sand or fire clay. Packing this material around the metal prevents oxidation. The box is then placed in the furnace, heated to the proper temperature, held there for a sufficient period, and then allowed to cool slowly in the sealed furnace.

Instructions for annealing the more common metals:

CAST IRON: Heat slowly to between 1400° and 1800°F, depending on composition. Hold at the specific temperature for 30 minutes, and then allow the metal to cool slowly in the furnace or annealing box.

STAINLESS STEEL (Austenitic): For full annealing, heat to between 1850° and 2050°F. Cool rapidly. For partial annealing, heat to between 1600° and 1700°F.

COPPER: Heat to 925°F. Quench in water. A temperature as low as 500°F will relieve most of the stresses and strains.

ZINC: Heat to 400°F. Cool in open, still air.

ALUMINUM: Heat to 750°F. Cool in open air. This reduces hardness and strength but increases electrical conductivity.

NICKEL-COPPER ALLOYS INCLUDING MONEL: Heat to between 1400° and 1450°F. Cool by quenching in water or oil.

NICKEL-MOLYBDENUM-IRON and NICKEL-MOLYBDENUM-CHROMIUM ALLOYS (Stellite): Heat to between 2100° and 2150°F. Hold at this temperature a suitable time, depending on thickness. Follow by rapid cooling in a quenching medium.

BRASS: Annealing to relieve stress may be accomplished at a temperature as low as 600°F. Fuller anneals may be accomplished with increased temperatures. Larger grain size and loss of strength will result from too high temperatures. Do NOT anneal at temperatures exceeding 1300°F. Brass should be slowly cooled to room temperature. Either wrap the part with heat retarding cloth or bury it in slaked lime or other heat retarding material.

BRONZE: Heat to 1400°F. Cool in an open furnace to 500°F or place in a pan to avoid uneven cooling caused by draft.

NORMALIZING

Normalizing is the process of heating a ferrous alloy to a suitable temperature above the critical temperature or transformation range (see section on hardening) and then cooling in still air. Normalizing relieves stresses and strains caused by welding, forging and uneven cooling. Normalizing also removes the effects of previous heat treatments.

HARDENING

Cutting tools, chisels, twist drills, and many other pieces of equipment and tools must be hardened to enable them to retain their cutting edges. Surfaces of roller bearings, parallel blocks, and armor plate must be hardened to prevent wear or penetration. Metals and alloys can be hardened in several ways; a brief general description of one method of hardening follows.

Each steel has a critical temperature at which a marked change will occur in the grain structure and physical properties. This critical temperature varies according to the carbon content of the steel. To be hardened, steel must be heated to a little more than this critical temperature—to ensure that every point in it will have reached critical temperature and to allow for some slight loss of heat when the metal is transferred from the furnace to the cooling medium. It is then cooled rapidly by being quenched in oil, freshwater, or brine. Quenching firmly fixes the structural changes which occurred during heating and thus causes the metal to remain hard.

If allowed to cool too slowly, the metal will lose its hardness. On the other hand, to prevent too rapid quenching which would result in warping and cracking, it is sometimes necessary to use oil instead of freshwater or saltwater for high carbon and alloy steels. Saltwater, as opposed to freshwater, produces greater hardness.

To prevent hard and soft spots when quenching, hold the part with a set of tongs made with long handles and grips or jaws that will hold the part firmly but with a minimum amount of surface contact. When you submerge the part in the cooling medium, rapidly move it up and down while moving it around the cooling medium container in a clockwise or counterclockwise direction.

TEMPERING

The tempering process relieves strains that are brought about in steel during the hardening process. Tempering makes the metal tougher and less brittle. Tempering is accomplished by heating the hardened steel to a temperature below the critical range, holding this temperature for a sufficient time to penetrate the whole piece, and then cooling the piece. In this process, ductility and toughness are improved, but tensile strength and hardness are reduced.

CASE HARDENING

Case hardening is a process of heat treating by which a hard skin is formed on metal, while the inner part remains relatively soft and tough. A metal that is originally low in carbon is packed in a substance high in carbon content and heated above the critical range. The case hardening furnace must give a uniform heat. The length of time the piece is left in the oven at this high heat determines the depth to which carbon is absorbed. A commonly used method of case hardening is to (1) carburize the material (an addition of carbon during the treatment), (2) allow it to cool slowly, (3) reheat, and (4) harden in water. Small pieces such as bolts, nuts, and screws, however, can be dumped into water as soon as they are taken out of the carburizing furnace.

HARDNESS TESTS

A number of tests are used to measure the physical properties of metals and to determine whether a metal meets specification requirements. Some of the more common tests are hardness tests, tensile strength tests, shear strength tests, bend tests, fatigue tests, and compression tests. Of primary importance to a Machinery Repairman is the hardness test.

Most metals possess some degree of hardness—that is, the ability to resist penetration by another material. Many tests for hardness are used; the simplest is the file hardness test. While fair estimates of hardness can be made by an experienced workman, more consistent quantitative measurements are obtained with standard hardness testing equipment. Such equipment eliminates the variables of size, shape, and hardness of the file selected, and of the speed, pressure, and angles of the file used by a workman when conducting a test. Before discussing the hardness test equipment, let us consider hardness itself, and the value of such information to a Machinery Repairman.

Hardness has been defined in various ways: resistance to penetration, resistance to abrasion, resistance to machine tool cutting, and resistance to bending (stiffness) by wrought products. Except for resistance to penetration, these characteristics of hardness are not readily measurable. Consequently, most hardness tests are based on the principle that a hard material will penetrate a softer one. In a scientific sense, then, hardness is a measure of the resistance of a material to penetration or indentation by an indenter of fixed size and geometrical shape, under a specific load.

The information obtained from a hardness test has many uses. It may be used to compare alloys and the effects of various heat treatments on them. Hardness tests are useful as a rapid, nondestructive method for inspecting and controlling certain materia'- and processes and to ensure that heat-treated objects have developed the hardness desired or specified. The results of hardness tests are useful not only for comparative purposes, but also for estimating other properties. For example, the tensile strength of carbon and low-alloy steels can be estimated from the hardness test number. There is also a relationship between hardness and endurance or fatigue characteristics of certain steels.

Hardness may be measured by many types of instruments. The most common are the Rockwell and Brinell hardness testers. Other hardness tests include the Vickers, Eberbach, Monotron, Tukon, and Scleroscope. Since there are many tests and the hardness numbers derived are not equivalent, the hardness numbers must be designated according to the test and the scale used in the test. Since you are more likely to have access to a Rockwell tester than any other, this method is discussed in detail. The essential differences between the Rockwell and Brinell tests will also be discussed in the sections which follow. In addition, the Scleroscope and Vickers hardness tests will be covered briefly.

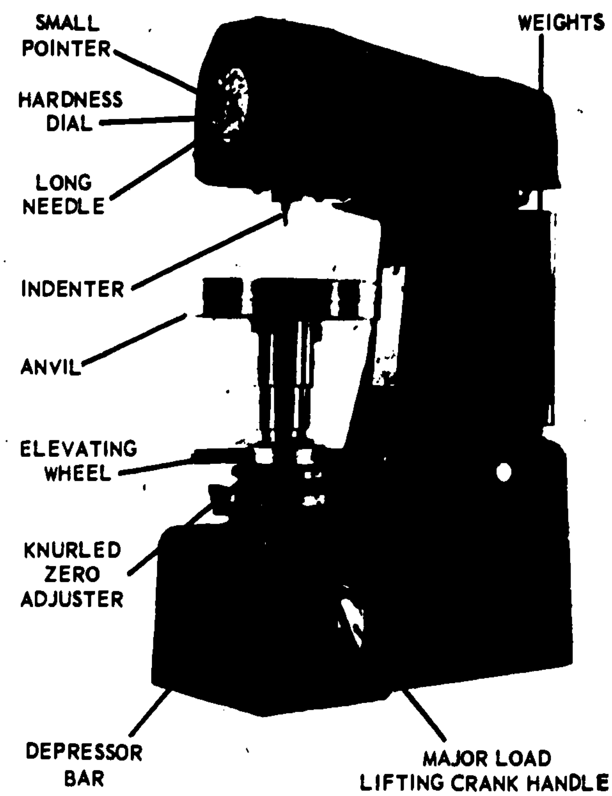


Figure 4-4.—Standard Rockwell hardness testing machine.

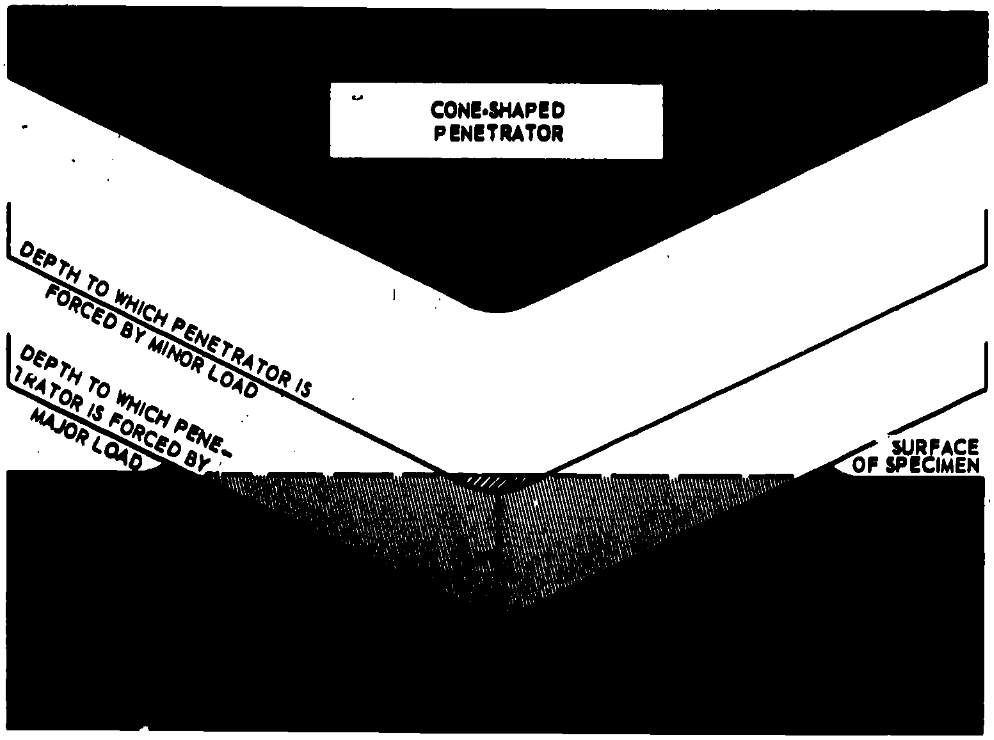
ROCKWELL HARDNESS TEST

Of all the hardness tests, the Rockwell is the one most frequently used. The basic principle of the Rockwell test (like that of the Brinell, Vickers, Eberbach, Tukron, and Monotron tests) is that a hard material will penetrate a softer one. This test operates on the principle of measuring the indentation, in a test piece of metal, made by a ball or cone of a specified size which is being forced against the test piece of metal with specified pressure. In the Rockwell tester shown in figure 4-4, the hardness number is obtained by measuring the depression made by a hardened steel ball (indenter) or a spheroconical diamond penetrator of a given size under a given pressure.

With the normal Rockwell tester shown, the 120° spheroconical penetrator is used in conjunction with a 150-kilogram (kg) weight to make impressions in hard metals. The hardness number obtained is designated Rockwell C (Rc). For softer metals, the penetrator is a 1/16-inch steel ball used in conjunction with a 100-kg weight. A hardness number obtained under these conditions is designated Rockwell B (Rb).

Figure 4-5 illustrates the principle of indenter hardness tests. Although the conical penetrator is shown, the principle is the same for a ball penetrator. (The geometry of the indentations will, of course, differ slightly.)

With the Rockwell tester, a deadweight, acting through a series of levers, is used to press the ball or cone into the surface of the metal to be tested. Then the depth of penetration is measured. The softer the metal being tested, the deeper the penetration will be under a given load. The average depth of penetration on samples of very soft steel is only about 0.008



126.87X

Figure 4-5.—Principle of Rockwell hardness test.

inch. The hardness is indicated on a dial, calibrated in the Rockwell B and the Rockwell C hardness scales. The harder the metal, the higher the Rockwell number will be. Ferrous metals are usually tested with the spheroconical penetrator, with hardness numbers being read from the Rockwell C scale. The steel ball is used for nonferrous metals and the results are read on the B scale.

With most indenter-type hardness tests, the metal being tested must be sufficiently thick to avoid bulging or marking the opposite side. The specimen thickness should be at least 10 times the depth of penetration. It is also essential that the surface of the specimen be flat and clean. When hardness tests are necessary on thin material, a superficial Rockwell tester should be used.

The Rockwell superficial tester differs from the normal Rockwell tester in the amount of load applied to perform the test and in the kind of scale used to interpret the results. When the major loads on the normal tester are 100 and 150 kg, the major loads on the superficial tester are 15, 30, and 45 kg. One division on the dial gage of the normal tester represents a vertical displacement of the indenter of 0.002 millimeter (mm). One division of the dial gage of the superficial tester represents a vertical displacement of the indenter of 0.001 mm. Hardness scales for the Rockwell superficial tester are the N and T scales. The N scale is used for materials of such hardness that, were they of sufficient thickness, they would be tested with the normal tester using the C scale. The T scale is comparable to the B scale used with the normal tester. In other respects the normal and superficial Rockwell testers are much alike.

Assuming the sample is properly prepared and the appropriate penetrator and weights are selected, the following step-by-step procedure will indicate how a Rockwell tester is used:

1. Place the piece to be tested on the testing table, or anvil.
2. Turn the wheel that elevates the testing table until the piece to be tested comes in contact with the testing cone or ball. Continue to turn the elevating wheel until the small pointer on the indicating gage is nearly vertical and slightly to the right of the dot.
3. Watch the long pointer on the gage; continue raising the work with the elevating wheel until the long pointer is nearly upright—within approximately five divisions, plus or minus, on the scale. This step of the procedure sets the minor load.
4. Turn the zero adjuster, located below the elevating wheel, to set the dial zero behind the pointer.
5. Tap the depressor bar downward to release the weights and apply the major load. Watch the pointer until it comes to rest.
6. Turn the crank handle upward and forward, thereby removing the major but not the minor load. This will leave the penetrator in contact with the specimen but not under pressure.
7. Observe where the pointer now comes to rest and read the Rockwell hardness number on the dial. If the test has been made with the 1/16-inch ball and a 100-kilogram weight, the reading is taken from the red, or B, scale. If the test has been made with the spheroconical penetrator and a weight of 150 kilograms, the reading is taken from the black, or C scale. (In the first example the number is prefixed by Rb, and in the latter instance by Rc.)
8. Turn the handwheel to lower the anvil. Then remove the test specimen.

BRINELL HARDNESS TEST

The Brinell hardness testing machine provides a convenient and reliable hardness test. The machine is not suitable, however, for thin or small pieces. This machine has a vertical hydraulic press design and is generally hand operated. A lever is used to apply the load which forces a 10-millimeter diameter hardened steel or tungsten-carbide ball into the test specimen. For ferrous metals, a 3,000-kilogram load is applied. For nonferrous metals, the load is 500 kilograms. In general, pressure is applied to ferrous metals for 10 seconds, while 30 seconds is required for nonferrous metals. After the pressure has been applied for the appropriate time, the diameter of the depression produced is measured with a microscope having an ocular scale.

The Brinell hardness number (Bhn) is the ratio of the load in kilograms to the impressed surface area in square millimeters. This number is found by measuring the distance the ball is forced, under a specified pressure, into the test piece. The greater the distance, the softer the metal, and the lower the Brinell hardness number will be. The width of the indentation is measured with a microscope, and the hardness number corresponding to this width is found by consulting a chart or table.

The Brinell hardness machine is of greatest value in testing soft and medium-hard metals and in testing large pieces. On hard steel the imprint of the ball is so small that it is difficult to read.

SCLEROSCOPE HARDNESS TEST

If you place a mattress on the deck and drop two rubber balls from the same height, one on the mattress and one on the deck, the one dropped on the deck will bounce higher. The reason is that the deck is the harder of the two surfaces; this is the principle upon which the Scleroscope works. When using the Scleroscope hardness test, drop a diamond-pointed hammer through a guiding glass tube onto the test piece and check the rebound (bounce) on a scale. The harder the metal being tested, the higher the hammer will rebound, and the higher will be the number on the scale. The Scleroscope is portable and can be used to test the hardness of pieces too large to be placed on the anvil or tables of other machines. Since the Scleroscope is portable and can be held in your hand, it can be used to test the hardness of large guns and marine and other forgings that cannot be mounted on stationary machines. Another advantage of the Scleroscope is that it can be used without damaging finished surfaces. The chief disadvantage, however, of this machine, is its inaccuracy. The accuracy of the Scleroscope is affected by the following factors:

1. Small pieces do not have the necessary backing and cannot be held rigidly enough to give accurate readings.

2. If large sections are not rigid, if they are oddly shaped, if they have overhanging sections, or if they are hollow, the readings may be in error.

3. If oil-hardened parts are tested, oil may creep up the glass tube and interfere with the drop of the diamond-pointed hammer in the instrument, thus causing an error.

VICKERS HARDNESS TEST

The Vickers test measures hardness by a method similar to that of the Brinell test. The indenter, however, is not a ball, but a square-based diamond pyramid, which makes it accurate for testing thin sheets as well as the hardest steels.

Up to an approximate hardness number of 300, the results of the Vickers and the Brinell tests are about the same. Above 300, Brinell accuracy becomes progressively lower. This divergence represents a weakness in the Brinell method—a weakness that is the result of the tendency of the Brinell indenter ball to flatten under heavy loads. For this reason, Brinell numbers over 600 are considered to be of doubtful reliability.

If a ship has one type of hardness tester and the specifications indicated by the blueprint are for another type, a conversion table, such as table 4-3, may be used to convert the reading.

File Hardness Test

Hardness tests are commonly used to determine the ability of a material under test to resist abrasion or penetration by another material. Many methods have evolved for measuring the hardness of metal. The simplest method is the file hardness test. This test cannot be used to make positive identification of metals but can be used to get a general idea of the type of metal being tested and to compare the hardness of various metals on hand. Thus, when identification of metal by other means is not possible, you can use a file to determine the relative hardness of various metals. The results of such a test may enable you to select a metal more suitable for the job being performed.

The file hardness test is simple to perform. You may hold the metal being tested in your hand and rested on a bench, or put it in a vise. Grasp the file with your index finger extended along the file and apply the file slowly but firmly to the surface being tested.

If the material is cut by the file with extreme ease and tends to clog the spaces between the file teeth, it is VERY SOFT. If the material offers some resistance to the cutting action of the file and tends to clog the file teeth, it is SOFT. If the material offers considerable resistance to the file but can be filed by repeated effort, it is HARD and may or may not have been heat treated. If the material can be removed only by extreme effort and in small quantities by the file teeth, it is VERY HARD and has probably been heat treated. If the file slides over the material and the file teeth are dulled, the material is EXTREMELY HARD and has been heat treated.

The file test is not a scientific method. It should not be used when positive identification of metal is necessary or when an accurate measurement of hardness is required. Tests already described should be used for positive identification of metals. Special machines, such as the Rockwell and Brinell testers, should be used when it is necessary to determine accurately the hardness of the material.

PLASTICS

Plastic materials are being increasingly used aboard ship. In some respects, they tend to surpass structural metals; plastic has proved to be shock resistant, not susceptible to saltwater corrosion, and in casting it lends itself to mass production and uniformity of end product.

CHARACTERISTICS

Plastics are formed from organic materials, generally with some form of carbon as their

Table 4-3.—Hardness Conversion Chart (Ferrous Metals)

Brinnel Hardness No. 3,000 kg	Rockwell Hardness No. C Scale	Tensile Strength Approximate X 1,000 psi	Brinnel Hardness No. 3,000 kg	Rockwell Hardness No. C Scale	Tensile Strength Approximate X 1,000 psi
	70C		477	50.3C	234
	69C		461	48.8C	226
	68C		444	47.2C	218
	67C		429	45.7C	210
767	66.4C	376	415	44.5C	203
757	65.9C	371	401	43.1C	196
745	65.3C	365	388	41.8C	190
733	64.7C	359	375	40.4C	184
722	64.0C	354	363	39.1C	178
710	63.3C	348	352	37.9C	172
698	62.5C	342	341	36.6C	167
682	61.7C	334	331	35.5C	162
670	61.0C	328	321	34.3C	157
653	60.0C	320	311	33.1C	152
638	59.2C	313	302	32.1C	148
627	58.7C	307	293	30.9C	144
601	57.3C	294	285	29.9C	140
578	56.0C	283	277	28.8C	136
555	54.7C	272	269	27.6C	132
534	53.5C	262	262	26.6C	128
524	52.1C	257	255	25.3C	125
495	51.0C	243			

Table 4-3.–Hardness Conversion Chart (Ferrous Metals)–Continued

Brinell Hardness No. 500 kg	Rockwell Hardness No. B Scale	Brinell Hardness No. 500 kg	Rockwell Hardness No. B Scale
201	99.0B	143	85.0B
195	98.2B	140	82.9B
189	97.3B	135	80.8B
184	96.4B	130	80.0B
179	95.5B	120	75.B
175	94.6B	110	70.0B
171	93.8B	100	63.5B
167	92.8B	95	60.0B
164	91.9B	90	56.0B
161	90.7B	85	52.0B
158	90.0B	80	47.0B
156	89.0B	75	41.0B
153	87.8B	70	34.0B
149	86.8B	65	26.0B
146	86.0B		

basic element. Plastics are referred to as synthetic material, but this does not necessarily mean that they are inferior to natural material. On the contrary, they have been designed to perform particular functions that no natural material can perform. Plastics can be obtained in a variety of colors, shapes, and forms–some are as tough, but not so hard, as steel; some are as pliable as rubber; some are more transparent then glass; and some are lighter than aluminum.

Plastic materials fall into two major divisions–THERMOSETTINGS and THERMOPLASTICS–and it is necessary, if you are going to perform any kind of shopwork on plastics, to know which of these two you are using.

Thermosettings are tough, brittle, and heat hardened. When placed in a flame, they will not burn readily, if at all. Thermosettings are so hard they resist the penetration of a knife blade; any such attempt will dull the blade. If the plastic is immersed in hot water and allowed to remain, it will neither absorb moisture nor soften.

Thermoplastics, on the other hand, when exposed to heat, become soft and pliable, or even melt. When cooled, they retain the shape that they took under the application of heat. Some thermoplastics will even absorb a small

Table 4-4.—Major Groups of Plastics

Plastic Trade Names in ( )	Advantages and Examples of Uses	Disadvantages
	THERMOPLASTICS	
Acrylic (Lucite, Plexiglass)	Formability; good impact strength; good aging and weathering resistance; high transparency, shatter-resistance, rigidity. Used for lenses, dials, etc.	Softening point of 170° to 220°F; low scratch resistance.
Cellulose nitrate (Celluloid)	Ease of fabrication; relatively high impact strength and toughness; good dimensional stability and resilience; low moisture absorption. Used for tool handles, mallet heads, clock dials, etc.	Extreme flammability; poor electrical insulating properties; harder with age; low heat distortion point.
Polyamide (Nylon)	High resistance to distortion under load at temperatures up to 300°F; high tensile strength, excellent impact strength at normal temperatures; does not become brittle at temperatures as low as minus 70°F; excellent resistance to gasoline and oil; low coefficient of friction on metals. Used for synthetic textiles, special types of bearings, etc.	Absorption of water; large coefficient of expansion; relatively high cost; weathering resistance poor.
Polyethylene (Polythene)	Inert to many solvents and corrosive chemicals; flexible and tough over wide temperature range, remains so at temperatures as low as minus 100°F; unusually low moisture absorption and permeability; high electrical resistance; dimensionally stable at normal temperatures; ease of molding; low cost. Used for wire and cable insulation, and acid resistant clothing.	Low tensile, compressive, flexural strength; very high elongation at normal temperatures; subject to spontaneous cracking when stored in contact with alcohols, toluene, and silicone grease, etc.; softens at temperatures above 200°F; poor abrasion and cut resistance; cannot be bonded unless given special surface treatment.

28.37.02

Table 4-4.—Major Groups of Plastics—Continued

Plastic Trade Names in ( )	Advantages and Examples of Uses	Disadvantages
	THERMOPLASTICS	
Polytetrafluoroethylene (Teflon)	Extreme chemical inertness; high heat resistance; nonadhesive; tough; low coefficient of friction. Used for preformed packing and gaskets.	Not easily cemented; cannot be molded by usual methods; generates toxic fumes at high temperatures; high cost.
	THERMOSETTING PLASTICS	
Phenolformaldehyde (Bakelite, Durez, Resinox)	Better permanence characteristics than most plastics; may be used at temperatures from 250° to 475°F; good aging resistance; good electrical insulating properties; not readily flammable, does not support combustion; inserts can be firmly embedded; strong, light; low water absorption; low thermal conductivity; good chemical resistance; economical in production of complex shapes; free from cold flow; relatively insensitive to temperature; low coefficient of thermal expansion; no change in dimensions under a load for a long time; does not soften at high temperatures or become brittle down to minus 60°F; inexpensive. Used for handles, telephone equipment, electrical insulators, etc.	Difficult to mold when filled for greatest impact strength, or when in sections less than 3/32-inch thick; can be expanded or contracted by unusually wet or dry atmosphere.
Urea-formaldehyde (Beetle, Bakelite Urea, Plaskon)	High degree of translucency and light finish; hard surface finish; outstanding electrical properties when used within temperature range of minus 70° to plus 170°F; complete resistance to organic solvents; dimensionally stable under moderate loadings and exposure conditions. Used for instrument dials, electric parts, etc.	Low impact strength; slight warping with age; poor water resistance.

28.37.02

amount of moisture, if placed in hot water. A knife blade will cut easily into thermoplastics.

When testing a plastic by inserting it into a fire, you should exercise caution, because thermoplastics will burst into sudden intense flame, and give off obnoxious gases. If you use the fire test, be sure to hold the plastic piece a considerable distance away from you.

MAJOR GROUPS

While it is not necessary for you to know the exact chemical composition of the many plastics in existence, it will be helpful to have a general idea of the composition of the plastics you are most likely to use. Table 4-4 provides information on some groups of plastics which are of primary concern to a Machinery Repairman.

Laminated plastics are made by dipping, spraying, or brushing flat sheets or continuous rolls of paper, fabric, or wood veneer with resins, and then pressing several layers together to get hard, rigid, structural material. The number of layers pressed together into one sheet of laminated plastic will depend upon the thickness desired. The choice of paper, canvas, wood veneer, or glass fabric will depend upon the end use of the product. Paper-base material is thin and quite brittle, breaking if bent sharply, but canvas-base material will be difficult to break. As layers are added to paper-base material, it gains in strength, but it is never as tough and strong in a laminated part as layers of glass fabric or canvas.

Laminated materials are widely used aboard ship. For example, laminated gears are used on internal-combustion engines, usually as timing or idler gears; on laundry equipment; and on certain pumps. In comparison with metal gears, plastic gears are quieter in operation, pick up less heat when friction is generated, and wear longer.

Plastics are identified by several commercial designations, trade names, and by Military and Federal specifications. There is such a large number of types, grades, and classes of plastics within each major group that to rely on the recognition of a trade name only would result in the wrong material being used. The appropriate Federal Supply Catalog should be used to cross reference the Military (MIL-P-XXXX) or Federal (FED-L-P XXXX) designations to the correct procuring data for the Federal Supply System.

MACHINING OPERATIONS

Machining operations that a Machinery Repairman will perform on plastics include cutting parts from sheet or rod stock, using various metal cutting saws; removing stock from parts by rotating tools as in a drill press or a milling machine; cutting moving parts by stationary tools, as on a lathe; and finishing operations.

Sawing

Several types of saws—bandsaw, jigsaw, circular saw—may be used to cut blanks from plastic stock. Speeds should be watched carefully. Since almost none of the heat generated will be carried away by the plastic, there is always danger that the tool will be overheated to the point that it will burn the work.

Drilling

In drilling plastics, back the drill out frequently to remove the chips and cool the tool. A liberal application of kerosene will help keep the drill cool. To obtain a smooth, clean hole, use paraffin wax on the drill; for the softer plastics, a special coolant may be preferred.

Lathe Operations

Lathe operations are substantially the same for plastics as for metals, except for type of tool, and the manner in which contact is made with the work. For plastics, set the tool slightly below center. Use cutting tools with zero or slightly negative back rake.

For both thermosettings and thermoplastics, recommended cutting speeds are: 200 to 500 fpm with high-speed steel tools and 500 to 1500 fpm with carbide-tipped tools.

Finishing Operations

Plastic must be given a finishing process to remove tool marks and produce a clean, smooth surface. Usually, sanding and buffing are sufficient for this purpose.

Surface scratches and pits can be satisfactorily removed by hand sandpapering with dry sandpaper of fine grit. Wet sanding can also be done by hand, with water and abrasive paper of fine grade. If a large amount of material must be removed, it will be more advantageous to use sanding wheels or disks.

After pits and scratches have been removed, the plastic should be buffed. This can be done on a wheel made of loose muslin buffs. Buffing compounds in common use are tripoli and rouge; a layer of the compound is deposited on the outside of the buffing wheel. The compound must be renewed frequently.

When large flat sheets are being buffed, be careful not to use too much pressure, nor to hold the work too long in one position. In buffing small plastic parts, be careful that the wheel does not seize the piece and pull it out of your grasp.

CHAPTER 5

POWER SAWS AND DRILLING MACHINES

Machine shop work is generally understood to include all cold metal work in which a portion of the metal is removed by either power driven tools or handtools. In your previous studies you have become familiar with common handtools. This chapter and the following chapters contain information on power driven, or machine tools.

The term MACHINE TOOL refers to any piece of power driven equipment that drills, cuts, or grinds metals and other materials. Through the use of attachments, some machine tools will perform two or more of these operations. Machine tools actually hold and work the material. The operator guides the mechanical movements by properly setting up the work and by adjusting the gearing or linkage controls. In this chapter we will deal primarily with power saws and drilling machines.

POWER SAW SAFETY PRECAUTIONS

Before taking up the operation of power saws, you must realize the importance of observing safety precautions. Carelessness is one of the prime causes of accidents in the machine shop. Moving machinery is always a potential danger. When this machinery is associated with sharp cutting tools, the hazard is greatly increased. Some of the more important safety precautions are listed here:

- Only authorized and fully qualified personnel may operate power saws.

- Goggles or face shields must be worn at all times when operating the power saw.

- NEVER make adjustments to the saw or relocate the stock to be sawed while the saw is in operation.

- Keep hands away from the saw blade as far as possible while the saw is in operation.

- NEVER attempt to move a large heavy piece of stock to or from the saw without help.

- Always support protruding ends of long pieces of stock so they will not fall and cause injury to either the machine or personnel.

- NEVER use bare hands to clean the saw cuttings from the machine.

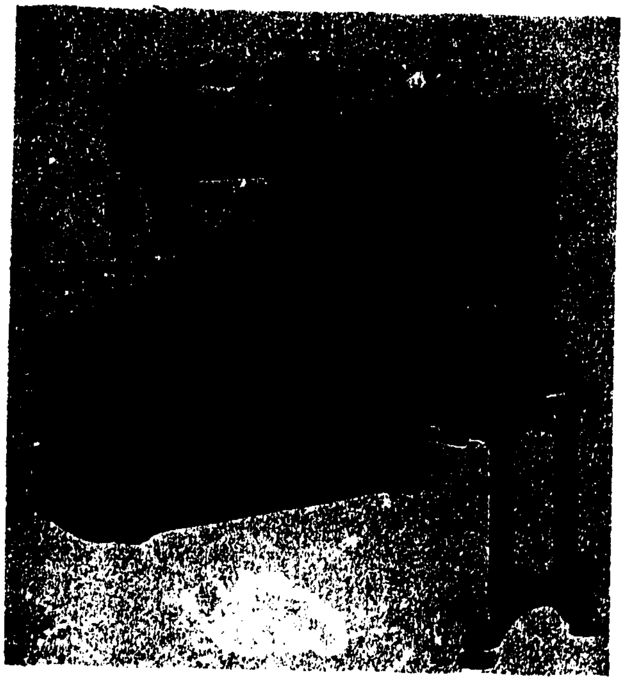
- Be alert for sharp burrs on the sawed end of stock and remove such burrs with a file to prevent injury to personnel.

- Inspect the blade at frequent intervals and NEVER use a saw with a dull, pinched, or burned blade.

- In all sawing jobs, the golden rule of safety is SAFETY FIRST, ACCURACY SECOND, and SPEED LAST.

POWER HACKSAWS

The power hacksaw is found in many Navy machine shops. It is used for cutting bar stock, pipe, tubing, or other metal stock. The power hacksaw consists of a base, a saw frame, and a work-holding device. Figure 5-1 is an illustration of a standard power hacksaw.



11.18X
Figure 5-1.—Standard power hacksaw.

The base consists of a reservoir to hold the coolant, a coolant pump, the drive motor and a transmission for speed selection. Some models may have the feed mechanism attached to the base.

The saw frame consists of linkage and a circular disk with an eccentric (offcenter) pin designed to convert circular motion into reciprocating motion. The blade is inserted between the two blade holders and securely attached by either hardened pins or socket head screws. The inside blade holder is stationary while the outside blade holder is adjustable. This adjustable blade holder allows the correct tension to be put on the blade to ensure that it is held rigidly enough to prevent the blade from wandering and causing a slanted cut. The feed control mechanism is also attached to the saw frame on many models.

The work holding device is normally a vise with one stationary jaw and one movable jaw. The movable jaw is mounted over a toothed rack to permit a rapid and easy initial adjustment close to the material to be cut. Final tightening is made by turning the vise screw until the material is held securely. An adjustable stop permits pieces of the same length to be cut without measuring each piece separately. A stock support stand (available for both sides of the saw) serves to keep long stock from falling when being cut.

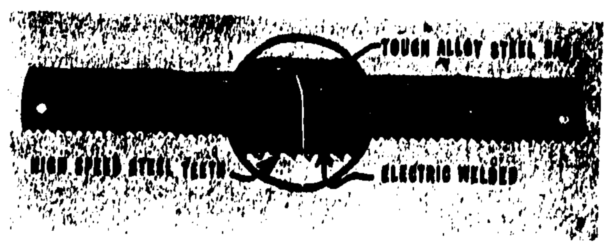
The capacity designation of the power hacksaw illustrated is 4 inches X 4 inches. This means that it can handle material up to 4 inches wide and 4 inches thick.

BLADE SELECTION

The blade shown in figure 5-2 is especially designed for use with the power hacksaw. It is made with a tough alloy steel back and high-speed steel teeth, a combination which gives a strong blade, and at the same time, a cutting edge suitable for high-speed sawing.

These blades vary as to the pitch of the teeth (number of teeth per inch). The correct pitch of teeth for a particular job is determined by the size and the material of the section to be cut. Use coarse pitch teeth for wide, heavy sections to provide ample chip clearance. For thinner sections, use a blade with a pitch that keeps two or more teeth in contact with the work so that the teeth do not straddle the work. Straddling strips the saw. In general, select blades according to the following information:

1. Coarse (4 teeth per inch), for soft steel, cast iron, and bronze.
2. Regular (6 to 8 teeth per inch), for annealed high carbon steel and high-speed steel.



11.19X
Figure 5-2.—Hacksaw blade.

3. Medium (10 teeth per inch), for solid brass stock, iron pipe, and heavy tubing.

4. Fine (14 teeth per inch), for thin tubing and sheet metals.

COOLANT

The use of a coolant is recommended for most power hacksawing operations. (Cast iron can be sawed dry.) The coolant keeps the kerf (narrow slot created by the cutting action of the blade) clear of chips so that the blade does not bind up and start cutting crooked. The teeth of the blade are protected from overheating by the coolant, permitting the rate of cutting to be increased beyond the speed possible when sawing without coolant. A soluble oil solution with a mixture of the oil and water, made so that no rust problems will occur, should be suitable for most sawing operations. The normal mixture for soluble oil is 40 parts water to 1 part oil.

FEEDS AND SPEEDS

Power hacksaws have one of three feed mechanisms:

1. Mechanical feed, which ranges from 0.001 to 0.025 inch per stroke, depending upon the class and type of material being cut.

2. Hydraulic feed, which normally exerts a constant pressure but is designed so that when hard spots are encountered the feed is automatically stopped or shortened to decrease the pressure on the saw until the hard spot has been cut through.

3. Gravity feed, where weights are provided on the saw frame and can be shifted to give more or less pressure of the saw blade against the material being cut.

To prevent unnecessary wear on the back side of the saw blade teeth, the saw frame and blade are automatically raised clear of the last cutting depth on the return stroke. The rate of feed or the pressure exerted by the blade on the cutting stroke is dependent on several factors—the toughness and hardness of the material, the size, and in the case of a hollow pipe, the wall thickness. A hard large diameter piece of stock must be cut with a slower or lighter feed rate than if it were soft and had a small diameter. Pipe with thin walls should be cut with a relatively light feed rate to prevent stripping the teeth from the saw blade or collapsing the walls of the pipe. A feed rate that is too heavy or fast will often cause the saw blade to wander, producing an angled cut.

Speed on hacksaws is stated in strokes per minute, counting only those strokes on which the blade comes in contact with the stock. Speed is changed by a gear shift lever. There may be a chart attached to or near the saw, giving recommended speeds for cutting various metals. The following speeds, however, can be used:

1. Medium and low carbon steel, brass, and soft metals–136.

2. Alloy steel, annealed tool steel, and cast iron–90.

3. Unannealed tool steel, and stainless steel–60.

POWER HACKSAW OPERATION

A power hacksaw is relatively simple to operate. There are, however, a few checks you should make to ensure goods cuts. Support overhanging ends of long pieces to prevent sudden breaks at the cut before the work is completely cut through. Block up irregular shapes so that the vise holds firmly. Check the blade to ensure that it is sharp and that it is secured at the proper tension.

Place the workpiece in the clamping device, adjusting it so that the cutting off mark is in line with the blade. Turn the vise lever to clamp the material in place. Be sure the material is held firmly.

See that the blade is not touching the workpiece when you start the machine. Blades are often broken when this rule is not followed. Feed the blade slowly into the work, and adjust the coolant nozzle so that it directs the fluid over the saw blade.

CONTINOUS FEED CUTOFF SAW

Figure 5-3 illustrates a type of cutoff saw that is now being used throughout the Navy. There are different models of this saw, but the basic design and operating principles remain the same.

BAND SELECTION AND INSTALLATION

The bands for the continuous feed cutoff saw are nothing more than an endless hacksaw blade. With this thought in mind, one can see that all the factors that were discussed for power hacksaw blade selection can be applied to this saw. This saw is also equipped with a band selection chart (fig. 5-3) to help you in making the proper selection. The bands come in two different forms; readymade loops of the proper length and coils of continuous lengths of 100 feet or more. Nothing must be done to the presized band, but the coils of saw bands must be cut to the proper length and then butt welded. (Butt welding is covered later in this chapter.)

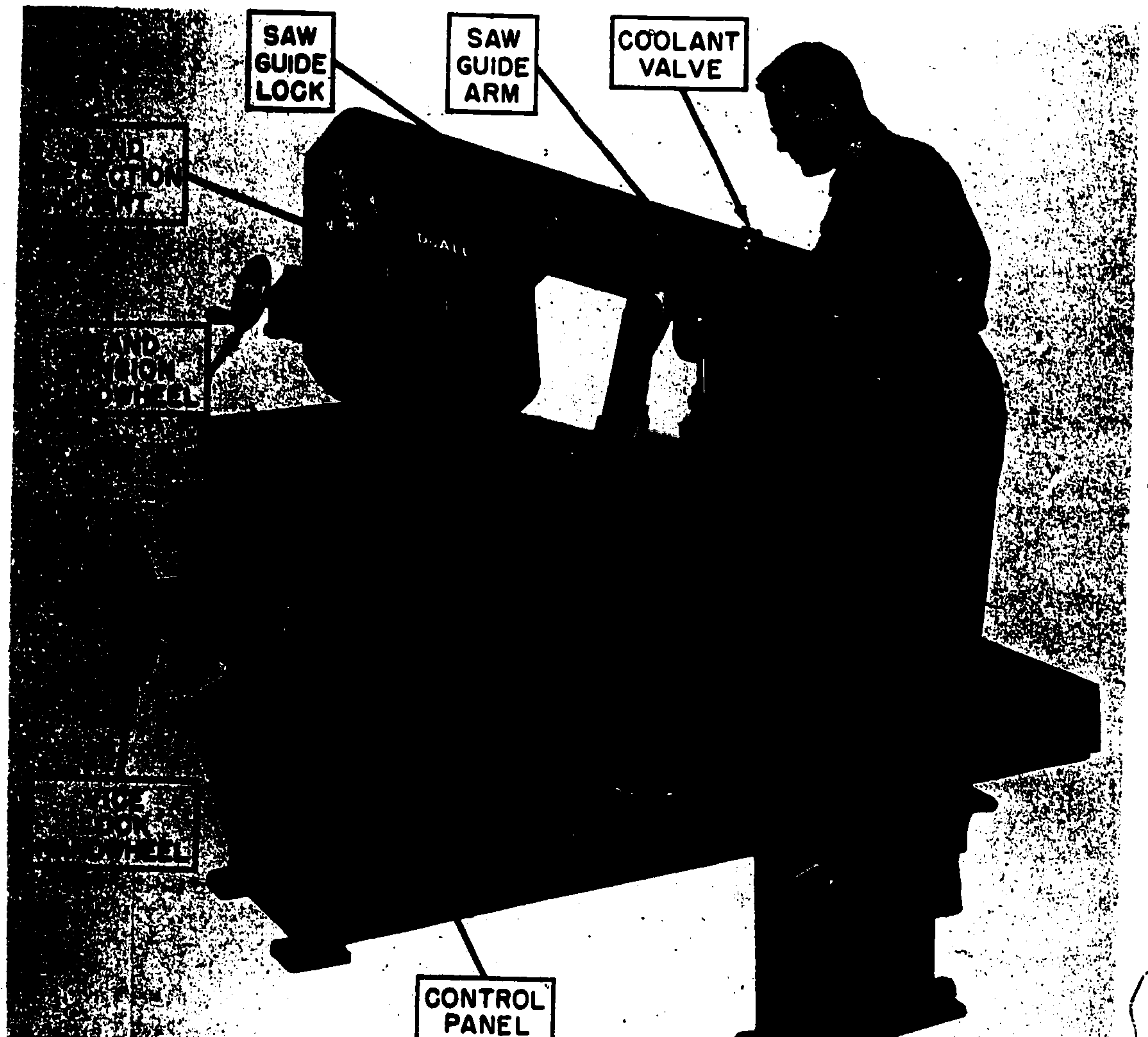


Figure 5-3.—Continuous feed cutoff saw (DoAll Company). 28.297X

Once you have selected the saw band, install it in the following manner:

1. Lift the cover on the saw head to expose the band wheels.

2. Place the band on the wheels with the teeth down, or toward the deck, and pointing in the direction of the band rotation.

3. Twist the lower part of the band and insert it between the saw guides at the lower end of the saw guide arms.

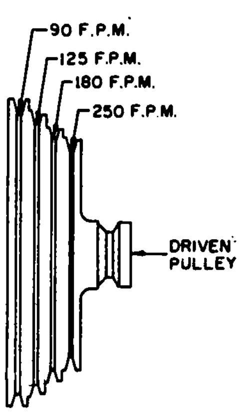
4. Turn the tension handwheel two full turns. This action applies enough tension to hold the band on the wheels. When the machine is operating, the hydraulic system maintains the proper band tension.

5. Adjust the saw guides in accordance with the manufacturer's manual. The distance between the two guide arms should not be more than necessary or the blade will wander.

6. Select the proper surface speed (feet-per-minute), and adjust the V-belt for that speed. (See fig. 5-4.)

CUTOFF SAW OPERATION

When the machine is ready for the actual sawing operation, you must ensure that the stock to be sawed is held securely in the machine. The movement of the saw head is controlled from the control panel (fig. 5-5). The operator can raise, stop, and feed the machine with the main control handle. The FEED portion of the control is divided into vernier and rapid. The RAPID area is used to bring the saw band down close to the work; the VERNIER controls the feed pressure. Figure 5-5 shows the vernier control knob with graduations from 0 to 9. By using this vernier, the operator can get the maximum cutting efficiency for the type of material being cut. Upon completion of the cut, the machine will automatically stop. To raise the head above the workpiece for the next cut, you must push the start button and place the control lever in the RAISE position. You may have to hold the start button down for a second or two until the saw head starts to raise.



28.295

Figure 5-4.—Speed change pulley.

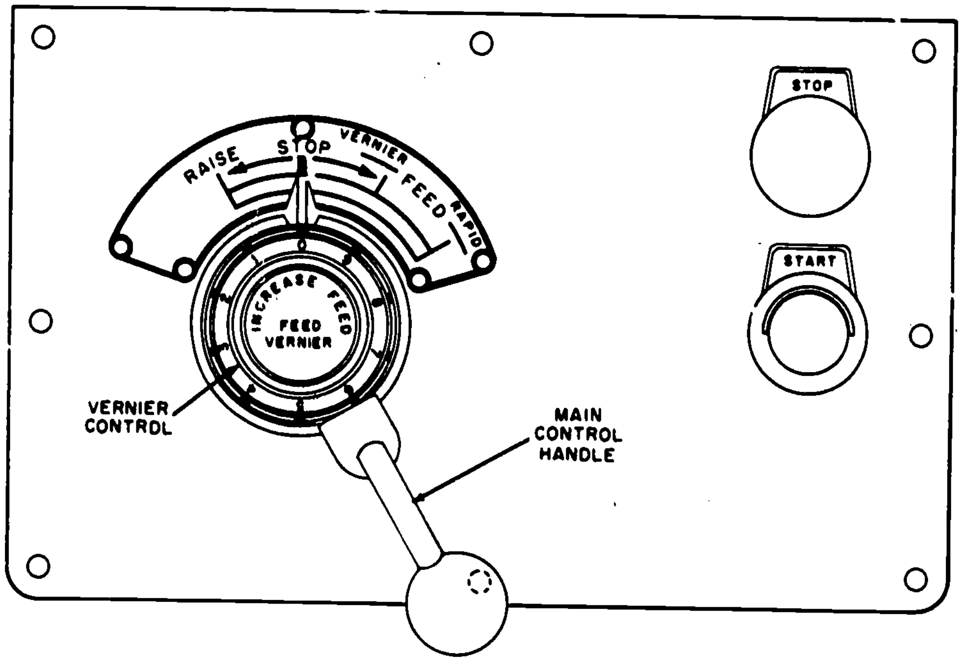
METAL CUTTING BANDSAWS

Metal cutting bandsaws are standard equipment in repair ships and tenders. These machines can be used for nonprecision cutting similar to that performed by power hacksaws. Some types can be used for precision cutting, filing, and polishing. A bandsaw has a greater degree of flexibility for straight cutting than a power hacksaw in that it can cut objects of any reasonable size and of regular and irregular shapes. A bandsaw cuts faster than a power hacksaw because the cutting action of the blade is continuous.

Figure 5-6 illustrates a metal cutting bandsaw of the tiltable table type. On the type shown, work is fed either manually or by power to the blade which runs in a fixed position.

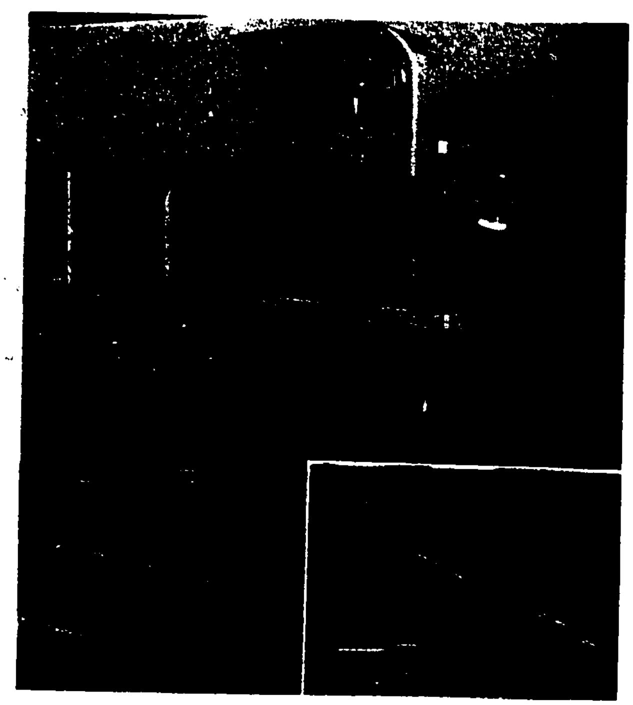
The tiltable band type saw is particularly suited to taking straight and angle cuts on large, long, or heavy pieces because the work remains in an upright position throughout the cutting operation and because there is no interference on long cutoffs from the free half of the band.

The tiltable table type is convenient for contour cutting because the angle at which work is fed to the blade can be changed readily. This machine usually has special attachments and accessories for precision inside or outside cutting of contours and disks and for mitering and has special bands for filing and polishing work.



28.296

Figure 5-5.—Control panel (DoAll saw).



11.21X

Figure 5-6.—Tiltable (contour) metal-cutting bandsaw.

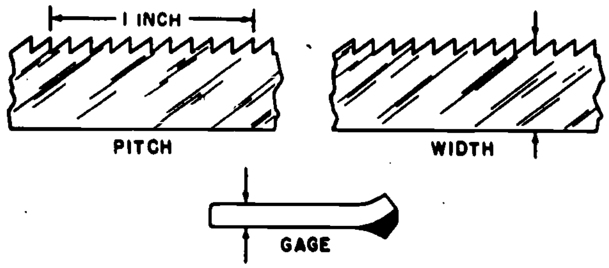
BANDSAW TERMINOLOGY

As was previously mentioned, the metal cutting bandsaws installed in machine shops in tenders and repair ships generally are the tiltable table type which can cut, file, or polish work when appropriate bands are mounted on the band wheels. The saw bands, file bands, and polishing bands used on these machines are called BAND TOOLS, and the machine itself is often referred to as a BAND TOOL MACHINE. Definitions which will be helpful in understanding band tool terminology are given below for saws, files, and polishing bands, in that order.

Saw Bands

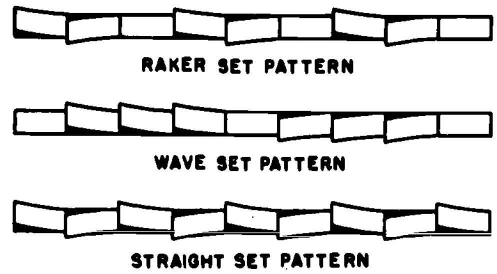
A saw band has the following characteristics which are illustrated in figures 5-7 through 5-9.

PITCH: Designates the number of teeth per linear inch.



29.15X

Figure 5-7.—Pitch, width, and gage.



28.43X

Figure 5-9.—Set pattern.

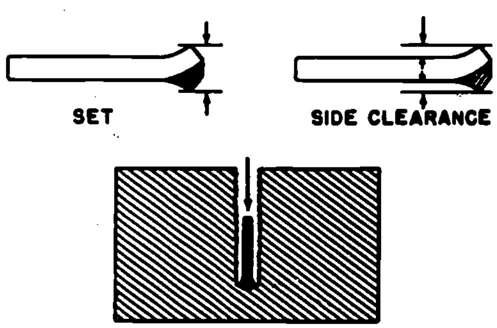
WIDTH: The distance across the flat face of the band. The width measurement is always expressed in inches, or fractions of an inch.

GAGE: The thickness of the band back. This measurement is expressed in thousandths of an inch.

SET: The bend or spread given to the teeth to provide clearance for the body or band back when a cut is being made.

SIDE CLEARANCE: The difference between the dimension of the band back (gage) and the set of the teeth. Side clearance provides running room for the band back in the kerf or cut. Without side clearance, a band will bind in the kerf.

SET PATTERN: There are three distinct patterns to which teeth are set: raker, wave, and straight. Raker set bands are generally used for solid cross section work. Wave set bands are used for cutting hollow materials, such as pipes and tubing, and for other work where there is a great deal of variation in thickness. Straight set bands are not used to any great extent for metal cutting work.



28.39X

Figure 5-8.—Set and side clearance.

TEMPER: Temper, or degree of hardness, of the teeth is indicated by the letters A and B, temper A being the harder. Temper A saws are used for practically all bandsaw metal cutting work.

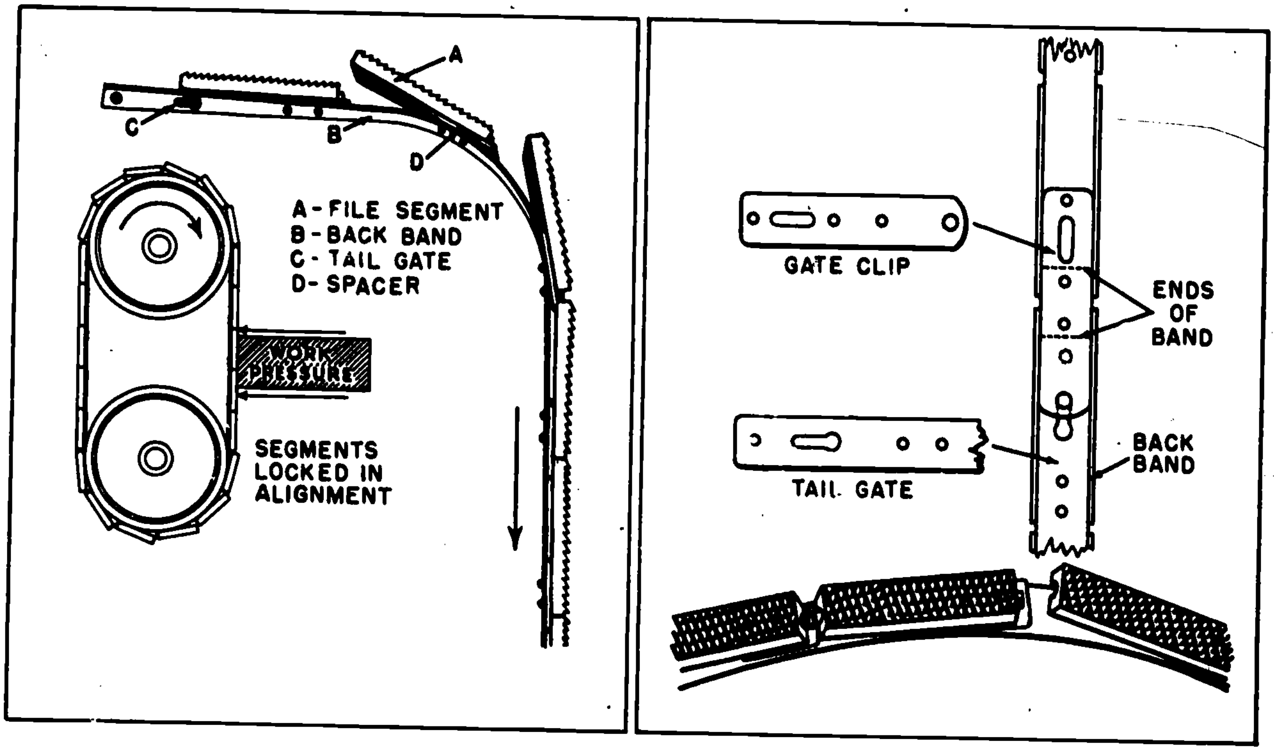
File Bands

A file band consists of a long steel strip upon which is mounted a number of file segments that can be flexed around the band wheels of the machine and still allow the file segments to present a straight line at the point of work. Figure 5-10 illustrates the file band flexing principle and shows the construction of a file band. The parts of a file band and their functions are described below:

FILE SEGMENT: A section of the cutting face of a file band. The individual segments are attached to the file band with rivets.

BACK BAND: The long steel strip or loop upon which the file segments are mounted. Do not confuse this term with BAND BACK, which refers to a part of a saw band.

GATE CLIP: A steel strip at the leading end of the back band—a part of an adapter for



28.41X

Figure 5-10.—File band flexing principle and construction.

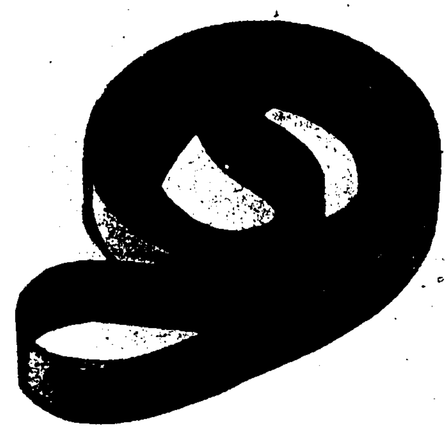
joining the back band ends to form the file band loop.

TAIL GATE: A steel strip at the other end of the back band. This is the other half of the adapter for joining the back band ends to form the file band loop.

SPACER: A small steel strip inserted between the file segment and the surface of the back band. There are as many spacers as there are file segments in each file band.

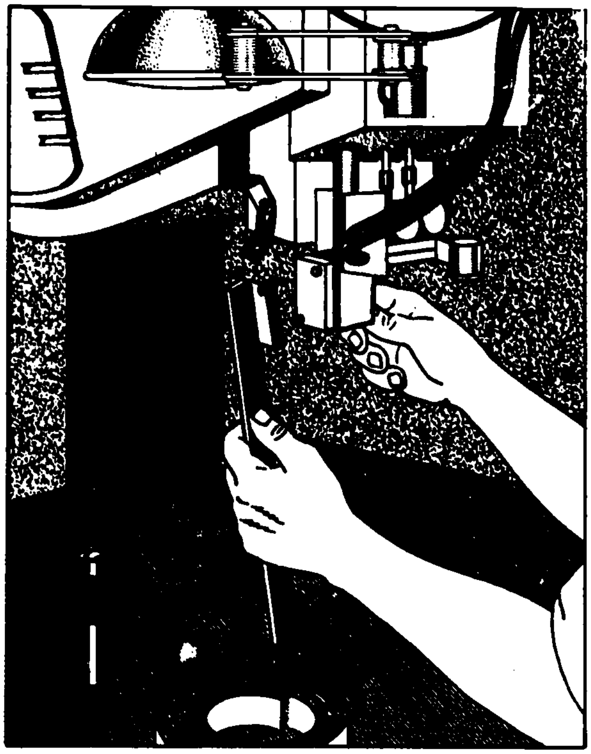
Polishing Bands

Abrasive coated fabric bands are used for grinding and polishing operations in a band tool machine. They are mounted in the same way as saw and file bands. Figure 5-11 shows a polishing band. Figure 5-12 shows a backup support strip being installed, before the polishing band is installed.



28.42X

Figure 5-11.—Polishing band.

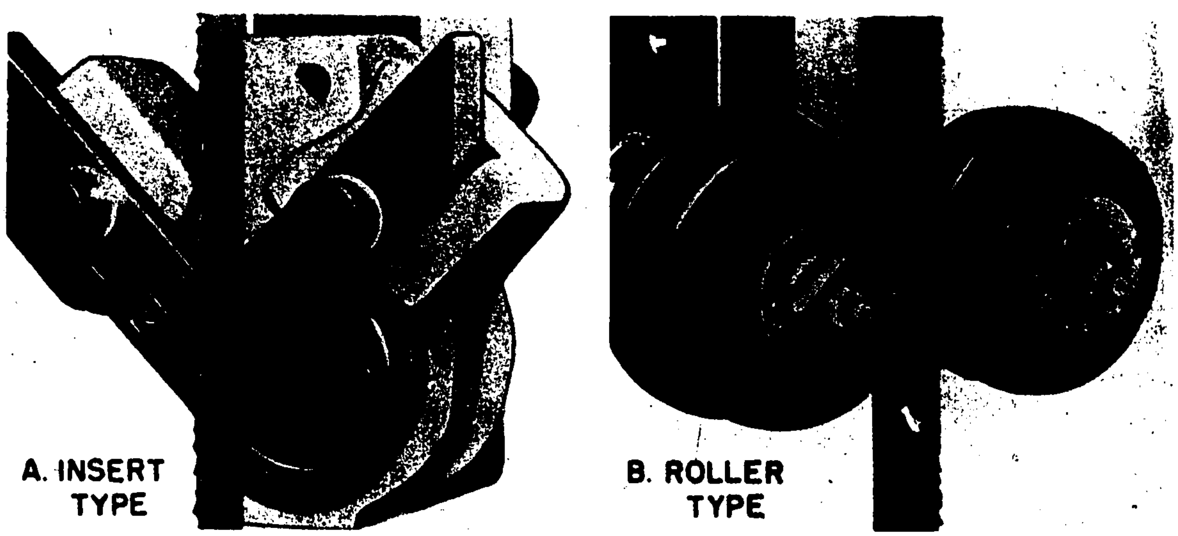


28.43X
Figure 5-12.–Installing a backup support strip for polishing band.

Band Tool Guides

SAW BAND GUIDES: The upper and lower guides keep the saw band in its normal track when work pressure is applied to the saw. The lower guide is in a fixed position under the work table, and the upper guide is attached to a vertically adjustable arm above the table which permits raising or lowering the guide to suit the height of work. To obtain adequate support for the band and yet not interfere with operation, place the upper guide so that it will clear the top of the workpiece by 3/8 to 1/2 inch. Figure 5-13 shows the two principal types of saw band guides: the insert type and the roller type. Note in both types the antifriction bearing surface for the band's relatively thin back edge; this feature allows the necessary work pressure to be placed on the saw without causing serious rubbing and wear. Be sure to lubricate the bearings of the guide rollers according to the manufacturer's recommendations.

FILE BAND AND POLISHING BAND GUIDES: For band filing operations, the regular saw band guide is replaced with a flat, smooth-surface metal backup support strip, as



28.44X

Figure 5-13.–Saw band guides.

shown in figure 5-14, which prevents sagging of the file band at the point of work. A similar support is used for a polishing band. This support has a graphite-impregnated fabric face that prevents undue wear on the back of the polishing band, which also is fabric.

SELECTION OF SAW BANDS, SPEEDS AND FEEDS

Saw bands are available in widths ranging from 1/16 to 1 inch; in various even-numbered pitches from 6 to 32; and in three gages—0.025, 0.032, and 0.035 inch. The gage of saw band that can be used in any particular machine, depends on the size of the band wheels. A thick saw band cannot be successfully used on a machine that has small diameter bandwheels; therefore, only one or two gages of blades may be available for some machines. Generally, only temper A, raker set, and wave set bands are used for metal cutting work. Another variable feature of saw bands is that they are furnished in readymade loops of correct length for some machines, and for others they come in stock form in coils of 100 feet or more from which a length can be cut and formed into a band loop by butt welding the ends together in a special machine. The process of joining the ends and installing bands will be described later in this chapter.

Band tool machines have a multitude of band speeds, ranging from about 50 feet per minute to about 1500 feet per minute. Most of these machines are equipped with a hydraulic feed which provides for three feeding pressures: low, medium, and heavy.

Success in your precision sawing with a metal cutting bandsaw depends to a large extent on your selecting the correct saw blade or band, running the saw band at the correct speed, and feeding the work to the saw at the correct rate. Many of the newer models of band tool machines have a JOB SELECTOR similar to the one shown in figure 5-15, which indicates the kind of saw band you should use for cutting any of a large variety of materials, the speed at which to operate the machine, and the power feed pressure to use.

Not all bandsaws have a job selector. You must know something about selecting the correct saw bands, speeds, and feeds to operate a bandsaw successfully. Table 5-1 will give you

28.45X

Figure 5-14.—File band guide.

28.46X

Figure 5-15.—Job selector.

Table 5-1.—Saw Pitch and Velocity for Various Metals

MATERIAL	SAW PITCH			SAW VELOCITY		
	Work Thickness			Work Thickness		
	1"	2"	Over 2"	1"	2"	Over 2"
FERROUS METALS						
Carbon Steel #1010-#1095*	14	10	6-8	175	150	125
Free Machining #X1112-#1340*	14	8	6-8	250	200	150
Nickel Chromium #2115-#3415*	14	10	6-8	100	85	60
Molybdenum #4023-#4820*	14	10	6-8	125	100	75
Chromium #5120-#52100*	14	10	8	100	75	50
Tungsten #7620-#71360*	14	10	6-8	85	60	50
Silicon Manganese #9255-#9260	14	10	6-8	100	75	50
* (SAE numbers)						
Armor Plate	14	12	6-8	100	75	50
Graphitic Steel	14	12	6-8	150	125	75
High Speed Steel	14	10	8	100	75	50
Stainless Steel	12	10	8	60	50	40
Angle Iron	14	14	10	190	175	150
Pipe	14	12	8	250	225	185
I Beams & Channels	14	14	10	250	200	175
Tubing (Thinwall)	14	14	14	250	200	200
Cast Steels	14	12	8	150	75	50
Cast Iron	12	10	8	200	185	160
NON-FERROUS METALS						
Aluminum (All Types)	8	6	6-8	250	250	250
Brass	8	8	8	250	250	250
Bronze (Cast)	10	8	8	175	125	50
Bronze (Rolled)	12	10	6-8	175	125	75
Beryllium Copper	10	8	6-8	175	150	125
Copper	10	8	6-8	250	225	225
Magnesium	8	8	6-8	250	250	250
Kirksite	10	8	6-8	200	175	150
Monel Metal	10	8	6-8	100	75	50
Zinc	8	8	6-8	250	225	200
NON-METALS						
Bakelite	10	8	6-8	250	250	250
Carbon	10	8	6-8	250	250	250
Plastics (All Types)	12	8	8	250	250	250
Wood	8	8	6-8	250	250	250

28.297

some of that information. Although this table does not cover all types and thicknesses of metals nor recommended feed pressures, it provides a basis on which you can build from your own experience.

Tooth Pitch

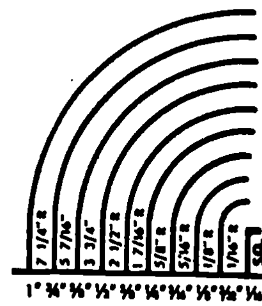
Tooth pitch is the primary consideration in selecting a saw band for any cutting job. For cutting thin materials, the pitch should be fine enough so that at least two teeth are in contact with the work; fewer than two will cause the teeth to tend to snag and tear loose from the band. For cutting thick material, the consideration is to not have too many teeth in contact with the work, because the greater the number of teeth in contact, the greater the feed pressure must be in order to force the teeth to cut into the material.

Excessive feed pressure puts severe strain on the band and the band guides, as well as causes sidewise wander of the band which results in off-line cutting. Another point to consider in selecting a saw band of proper pitch for a particular cutting job is the composition of the material to be cut, its hardness, and its toughness. Table 5-1 is a saw band pitch and velocity selection chart showing the pitch of saw band to use for cutting many commonly used metals.

Band Width and Gage

The general rule is to use the widest and thickest saw band consistent with the requirements of the job at hand. For example, a band of maximum width and thickness (if bands of different thickness are available) is used when the job calls for only straight cuts. On the other hand, when a layout requires radius cuts (curved cuts), the band selected must be capable of following the sharpest curve involved. Thus for curved work, select the widest band that will negotiate the smallest radius required. The saw band width selection guides, shown in figure 5-16, give the radius of the sharpest curve that can be cut with a particular width saw band. Note that the job selector illustrated in figure

WIDTH OF SAW BAND	MINIMUM RADII CUT
1/16"	SQ.
3/32"	1/16"
1/8"	1/8"
3/16"	5/16"
1/4"	5/8"
3/8"	1 7/16"
1/2"	2 1/2"
5/8"	3 3/4"
3/4"	5 7/16"
1"	7 1/4"



28.47X

Figure 5-16.—Saw band width selection guides.

5-15 contains a saw band radii cutting diagram similar to the one shown in figure 5-16.

Band Speeds

The rate at which the saw band travels in feet per minute from wheel to wheel is the saw band speed for velocity. Saw band velocity has considerable effect upon both the smoothness of the cut surfaces and the life of the band. The higher the band velocity, the smoother the cut; however, heat generated at the cutting point increases as band velocity increases. Too high a band velocity causes overheating and failure of the saw teeth. The band velocities given in Table 5-1 are based on manufacturers' recommendations, which in turn are based on data obtained from saw life tests and cutting experiments under various conditions. If you follow the recommendations given, you will be assured of the best band performance consistent with maximum band life.

Adjustment of the machine to obtain the proper band velocity cannot be covered in detail here because speed change is accomplished by different methods in different models of machines. Consult the manufacturer's technical manual for your particular machine and learn how to set up the various speeds available.

Feeds

Though manual feeding of the work to the saw is satisfactory for cutting metals up to 1 inch thick, power feeding generally provides better results and will be much safer for the operator. Regardless of whether power or manual feed is used, it is important not to crowd the saw because the band will tend to bend and twist. Feed pressure must not be so light that the teeth slip across the material instead of cutting through because this rapidly dulls the teeth. The job selector, shown in figure 5-15, shows the correct feed pressures for cutting any of the materials listed on the outer ring of the dial. In the absence of a job selector, table 5-2 can be used as a guide for selecting feed pressures for hard, medium hard, and soft metals.

The power feed controls vary with different makes of bandsaws and even with different models of the same make; therefore, no description of the physical arrangement of the power feed controls will be given here. Consult the manufacturer's technical manual and study the particular machine to learn its power feed arrangement and control.

SIZING, SPLICING, AND INSTALLING BANDS

Most contour cutting type bandsaws are provided with a buttwelder-grinder combination for joining saw bands that come in bulk stock coil form, and for joining broken band loops. The butt welder is usually attached to the saw machine, as shown in figure 5-17, although it may be portable or a pedestal-mounted accessory. The butt welder also makes inside cutting possible, since the saw band loop can be parted and rejoined after having been threaded through a starting hole in the work.

The following sections describe how to determine the length of the band, how to join the ends in the butt welder, and how to install a band tool in the machine.

Band Length

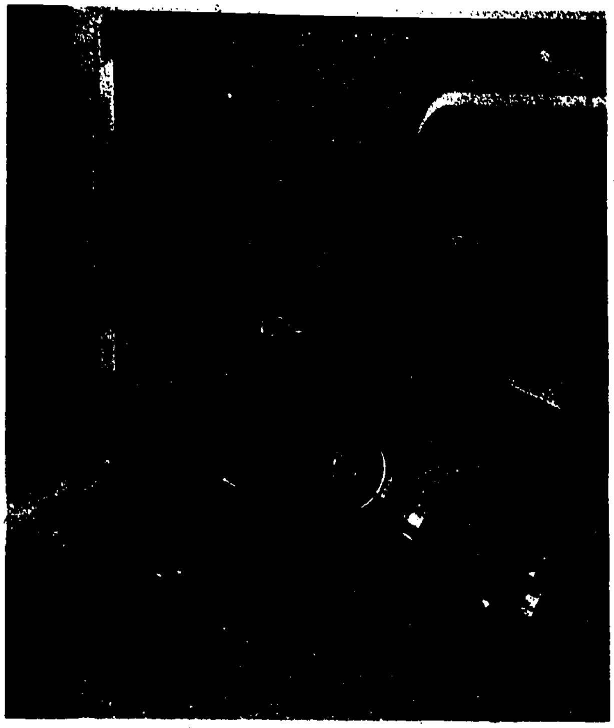
You can quickly determine the correct saw band length for any two-wheeled bandsaw by measuring the distance from the center of one wheel to the center of the other wheel,

Table 5-2.—Feed Pressures* for Hard, Medium Hard, and Soft Metal

Material	Work thickness				
	0-1/4"	1/4-1/2"	1/2-1"	1-3"	Over 3"
Tool Steel	M	M	H	H	H
Cast iron	M	M	M	H	H
Mild steel	L	M	H	H	H
Nickel-copper	L	M	H	H	H
Copper-nickel	L	L	M	H	H
Zinc	L	L	M	M	M
Lead	L	L	M	M	M

* L–light, M–medium, H–heavy.

28.298



28.48X
Figure 5-17.—Butt welder-grinder unit.

mulitplying by 2, and adding the circumference of one wheel.

The easiest way to determine the length of saw band for a three-wheeled machine is to take a steel tape, thread it through the machine over the wheels, and measure the distance.

Before measuring between the wheel centers, adjust the upper wheel so that it is approximately halfway between the upper and lower limits of vertical travel. This allows for taking up any band stretch resulting from operation.

Band Splicing

Figure 5-17 shows band ends being joined by using a butt welder. The procedure for joining is as follows:

1. Grind both ends of the band until they are square with the band back edge. If you do not do this carefully, the weld may not go completely across the ends of the band and as a result, the weld will not withstand the pressure of the cut when it is used. One easy method to ensure that the ends of the band will go together perfectly is to twist one end of the band until the teeth are on the opposite side of the other end. Place the band ends on top of each other, ensuring that the band back edge and the teeth are in a straight line on both sides. Carefully touch the ends of the band to the face of the grinding wheel and lightly grind until both ends have been ground completely across. Release the ends of the band so that they assume their normal position. Lay the back edge of the band on a flat surface and bring the ends together. If you did the grinding correctly, the ends will meet perfectly.

2. Set the controls of the butt welder to the weld position and adjust according to the width of band to be welded. The various models of butt welders that are found in many machine shops differ in the number of controls that must be set and the method of setting them. Most models have a lever that must be placed in the weld position so that the stationary and the movable clamping jaws are separated the correct distance. Some models have a resistance setting control which is set according to the width of band, while other models have a jaw pressure control knob that is also set according to band width. Read the manufacturer's instruction manual carefully before attempting welding.

3. Place the ends of the band in the jaws with the teeth of the band facing away from the welder. Push the back edge of the band firmly back toward the flat surfaces behind the clamping jaws to ensure proper alignment. Position the ends of the band so that they touch each other and are located in the center of the jaw opening. Some models of butt welders have interchangeable inserts for the clamping jaws to permit welding bands of different widths. This is done so that the teeth of the band are not damaged when the jaws are clamped tight.

4. You are now ready to weld the band. Some welders require that the weld button be fully depressed and held until the welding is

complete, while other welders required only that the button be fully depressed and then quickly released. There will be a shower of sparks from the welding action. Be sure that you are wearing either safety glasses or a face shield before welding and then stand back from the welder when you push the button.

5. When the welding is complete, release the jaw clamps and remove the band from the welder. Inspect the band to be sure it is straight and welded completely across. Do not bend or flex the band at this time to test the weld. The welding process has made the weld and the area near it hard and brittle and breakage will probably occur.

6. Place the lever that controls movement of the jaws in the anneal position. This should separate the jaws again. Set the control that regulates the anneal temperature to the setting for the width of the band.

7. Place the band in the clamping jaws with the teeth toward the welder and the welder section in the center of the jaw opening. Close the jaws.

8. The band is ready to be annealed. Push the anneal button and then quickly release repeatedly until the welded area becomes a dull cherry red. (Do NOT push and hold the anneal button. This will overheat and damage the band.) After the proper temperature is reached, push the anneal button and release it with increasingly longer intervals between the push cycle to allow the band to cool slowly.

9. The metal buildup resulting from the weld must be ground off. Using the attached grinding wheel, remove the weld buildup from both sides and the back until the band fits snugly into the correct slot on the saw band thickness gage mounted on the welder. You must do this grinding carefully to prevent looseness or binding between the saw guides and the band. Be careful not to grind on the teeth of the band.

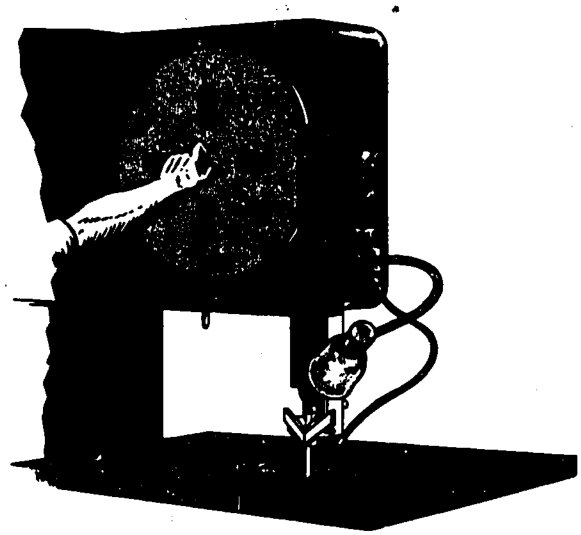
10. Repeat the procedure for annealing in step 8 after grinding the blade.

11. The welding process is complete. To test your weld, hold the band with both hands and form a radius in the band slightly smaller than the smallest wheel on the bandsaw by bringing your hands together. Move your hands up and down in opposite directions and observe the welded area as it rolls around the radius that you formed.

Installing Bands

Insert saw band or tool guides of correct size for the band you are going to install. Adjust the upper band wheel for a height that will allow the band to be looped easily around the wheels. Then place one end of the loop over the upper band wheel and the other end of the loop around the lower band wheel, being sure that the teeth are pointing downward on the cutting side of the band loop and that the band is properly located in the guides. Place a slight tension on the band by turning the upper wheel takeup hand wheel and revolve the upper band wheel by hand until the band has found its tracking position. If the band does not track on the center of the crowns of the wheels, use the upper wheel tilt control to correct the band track as illustrated in figure 5-18.

When the band is tracking properly, adjust the band guide rollers or inserts so that you have a total clearance of 0.001 to 0.002 inch between



28.49X

Figure 5-18.—Upper wheel tilt adjustment.

the sides of the band back and the guide rollers or inserts, and a slight contact between the back edge of the band back and the backup bearings of the guides. When band guide clearance has been set, increase the band tension. The amount of tension to be put on the band depends upon the width and gage of the band. A narrow, thin band will not stand as much tension as a wider or thicker band. Too much tension will cause a band to break; insufficient tension will cause the saw to run off the cutting line. The best way to obtain the proper tension is to start with a moderate tension; if the saw tends to run off the line when cutting, increase the tension slighty.

SAWING OPERATIONS

As previously mentioned, the types of sawing operations possible with a band tool machine are straight, angular, contour, inside, and disk cutting. The procedures for each of these cutting operations are described in the following paragraphs; but first, let us consider the general rules applicable to all sawing operations.

General Rules

- Check the level of the worktable and adjust the table, if necessary, to suit the angle of the cut.

- Use the proper blade and speed for each cutting operation. This ensures not only the fastest and most accurate work but also longer saw life.

- Always be sure the band guide inserts are the correct size for the width of the band installed and that they are properly adjusted.

- Before starting the machine, adjust the height of the upper band guide so that it will clear the work from 3/8 to 1/2 inch. The closer the guide is to the work, the greater the accuracy.

- When starting a cut, feed the work to the saw gradually. After the saw has started the kerf, increase the feed slowly to the recommended pressure. Do not make a sudden change in feed pressure because such a change may cause the band to break.

- Be sure the saw band and guides are properly lubricated.

- Use lubricants and cutting coolants as recommended by the manufacturer of your machine.

Straight Cuts with Power Feed

1. Change band guides as necessary. Select and install the proper band for the job and adjust the band guides.

2. Place the workpiece on the table of the machine and center the work in the work jaw.

3. Loop the feed chain around the work jaw, the chain roller guides, and the left-right guide sprocket, as shown in figure 5-19.

4. Determine the proper band speed and set the machine speed accordingly.

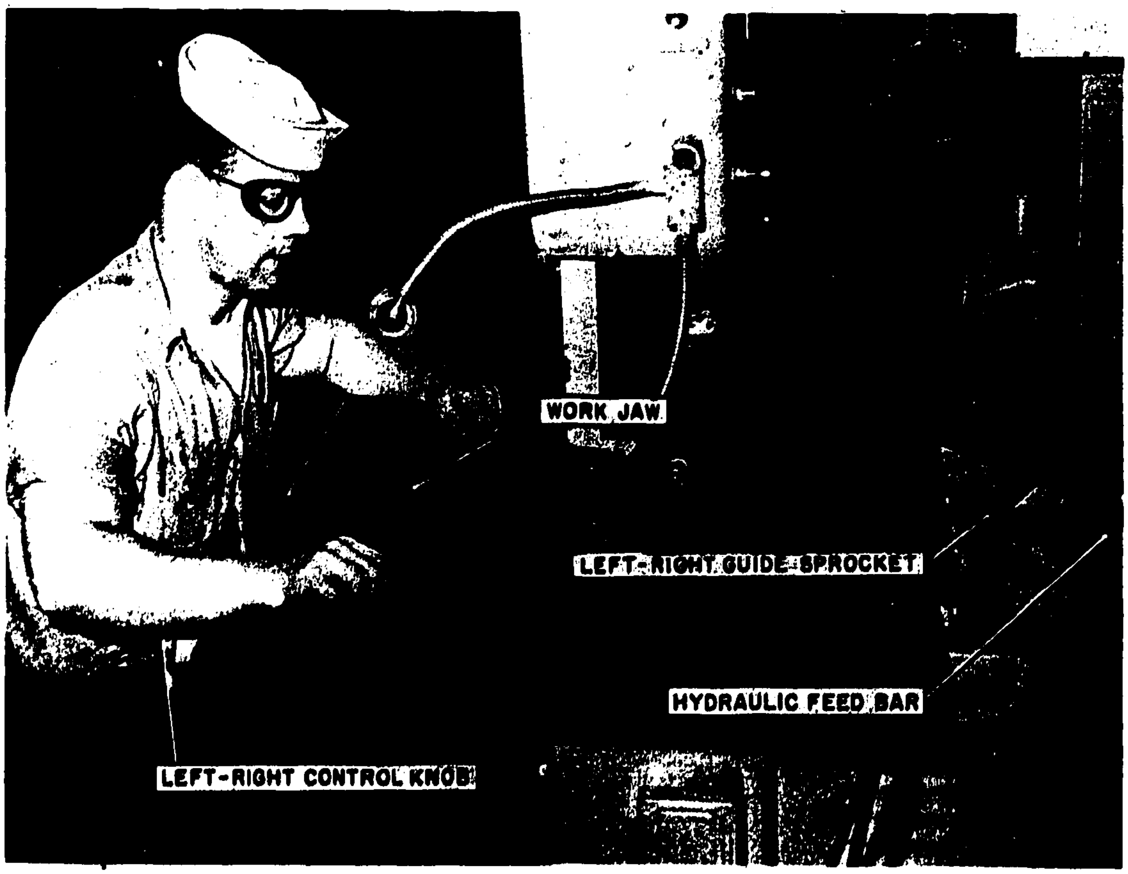
5. Start the machine and feed the work to the saw in the manner described in the general rules of operation given in the preceding section. Use the left-right control for guiding the work along the layout line.

Angular Cutting

Angular or bevel cuts on flat pieces are made in the same way as straight cuts except that the table is tilted to the desired angle of the cut as shown in figure 5-20.

Contour Cutting

Contour cutting, that is, following straight, angle, and curved layout lines, can be done offhand or with the power feed. The left-right chain control, shown in figure 5-19, is used for guiding the work along the layout line when power feed is used. A fingertip control for actuating the sprocket is located at the edge of the worktable. If there are square corners in the layout, drill a hole adjacent to each corner; this will permit the use of a wider band, greater feed pressure, and faster cutting.



28.50X

Figure 5-19.—Work jaw and feed chain adjustment.

28.50X

Figure 5-20.—Angular cutting.

Figure 5-21 shows the placement of corner holes on a contour cutting layout.

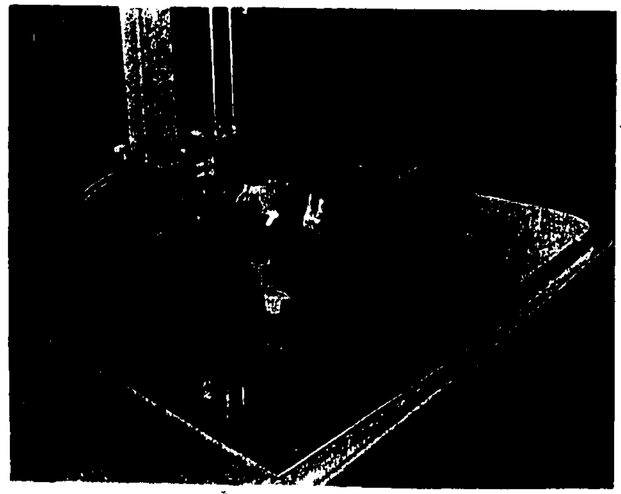
Inside Cutting

To make an inside cut, drill a starting hole slightly larger in diameter than the width of the band you are going to use. Remove the band from the machine. Shear the band; slip one end through the hole, and then splice the band. When the band has been spliced and reinstalled, the machine is ready for making the inside cut as illustrated in figure 5-22.

Disk Cutting

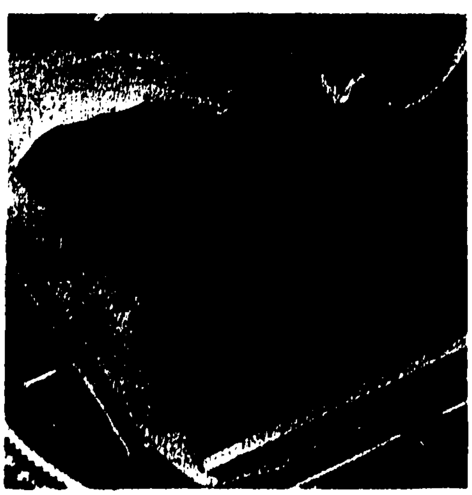
Disk cutting can be done either offhand by laying out the circle on the workpiece and following the layout circle or by using a disk cutting attachment which automatically guides

28.52X
Figure 5-21.—Sharp radii cutting eliminated by drilling corner holes.



28.54X
Figure 5-23.—Disk-cutting attachment.

the work so that a perfect circle is cut. Figure 5-23 shows a disk cutting attachment in use. The device consists of a radius arm, a movable pivot point, and a suitable clamp for attaching the assembly to the saw guidepost. To cut a disk using this device, lay out the circle and punch a center point. Clamp the radius arm to the guidepost. Position the workpiece (fig. 5-23) so that the saw teeth are tangent to the scribed circle. Adjust the pivot point radially and vertically so that it seats in the center-punch mark; then clamp the pivot point securely. Then rotate the work around the pivot point to cut the disk.



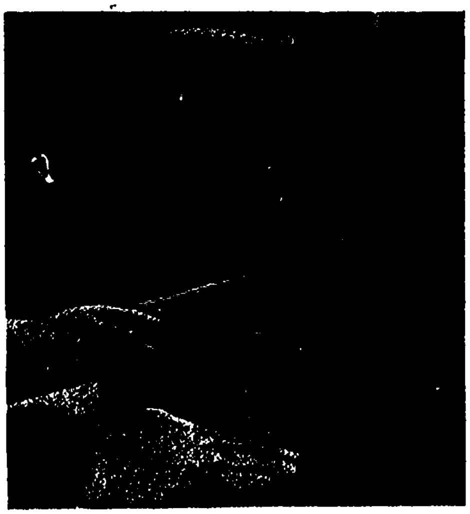
28.53X
Figure 5-22.—Inside cutting.

Filing and Polishing

In filing and polish finishing, the work is manually fed and guided to the band. Proper installation of the guides and backup support strips is very important if good results are to be obtained. A guide fence similar to the one shown in figure 5-24 is very helpful when working to close tolerances. Be sure to wear goggles or an eye protection shield when filing and polishing, and above all, be careful of your fingers. For proper band speeds and work pressures, consult the manufacturer's technical manual for the machine you are using.

DRILLING MACHINES AND DRILLS

Although drilling machines or drill presses are commonly used by untrained personnel, you cannot assume that operating these machines proficiently is simply a matter of inserting the



28.55X

Figure 5-24.—Polish finishing.

proper size drill and starting the machine. As a Machinery Repairman, you will be required to perform drilling operations with a great degree of accuracy. It is therefore necessary for you to be well acquainted with the types of machines and the methods and techniques of operation of drill presses and drills found in Navy machine shops.

DRILLING MACHINE SAFETY PRECAUTIONS

Because of the widespread use of the drill press by such a diverse group of people with different training and experience backgrounds, some unsafe operating practices have become rather routine in spite of the possibility of serious injury. The basic safety precautions for the use of a drill press are listed below:

- Always wear safety glasses or a face shield when operating a drill press.

- Keep loose clothing clear of rotating parts.

- NEVER attempt to hold a piece being drilled in your hand. Use a vise, hold-down bolts or other suitable clamping device.

- Check the twist drill to ensure that it is properly ground and is not damaged or bent.

- Make sure that the cutting tool is held tightly in the drill press spindle.

- Use the correct feeds and speeds.

- When feeding by hand, take care to prevent the drill from digging in and taking an uncontrolled depth of cut.

- Do NOT remove chips by hand. Use a brush.

TYPES OF MACHINES

The two types of drilling machines or drill presses common to the Navy machine shop are the upright drill press and the radial drill press. These machines have similar operational characteristics but differ in application in that the radial drill provides for positioning the drilling head rather than the workpiece.

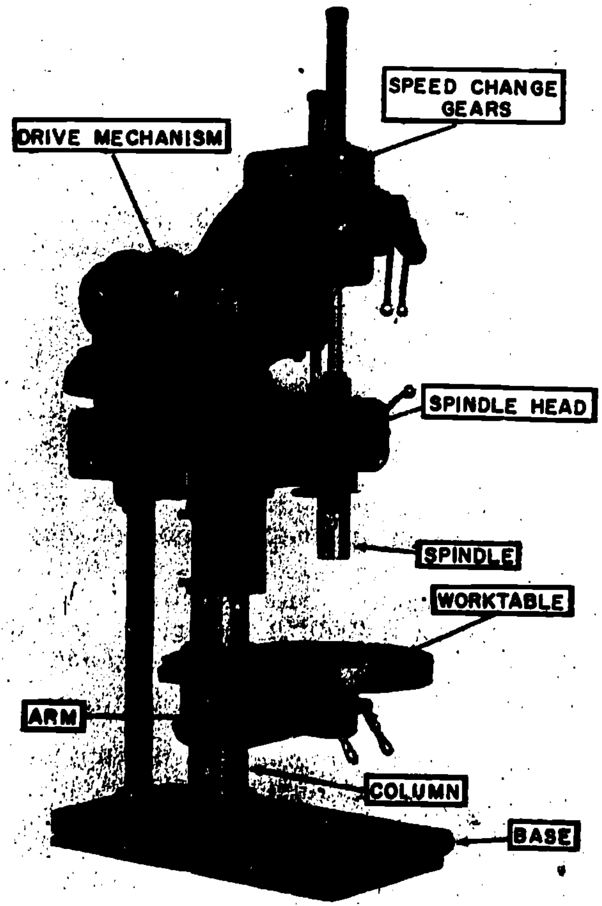
Upright drill presses discussed in this section will be the general purpose, the heavy duty, and the sensitive drill presses. One or more of these types will be found on practically all ships. They are classified primarily by the size of drill that can be used, and by the size of the work that can be set up.

The GENERAL PURPOSE DRILL PRESS (ROUND COLUMN), shown in figure 5-25, is perhaps the most common upright type of machine and has flexibility in operational characteristics. As you can see in the illustration the basic components of this machine are:

The BASE has a machined surface with T-slots for heavy or bulky work.

The COLUMN supports the worktable, the drive mechanism and spindle head.

The WORKTABLE and ARM can be swiveled around the column and can be moved up or down to adjust for height. In addition, the worktable may be rotated 360° about its own center.



11.9X
Figure 5-25.—General purpose drill press.

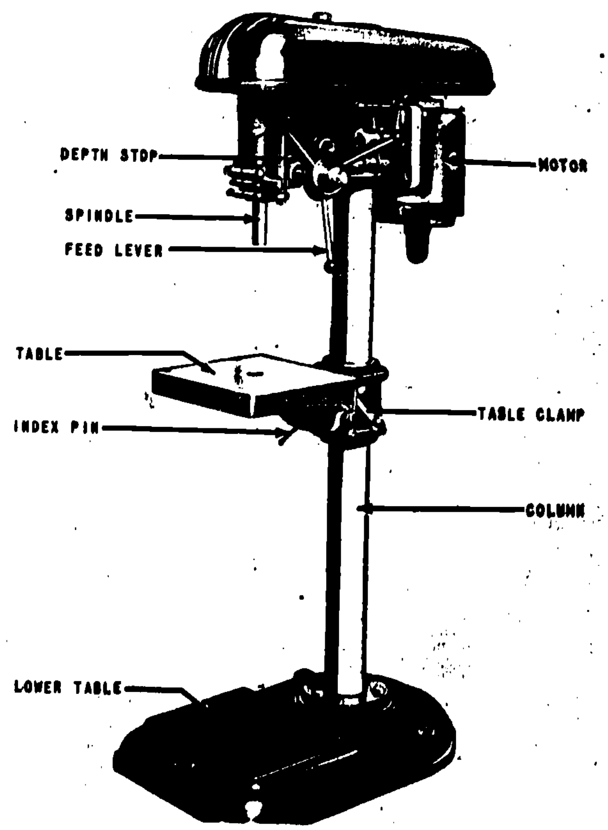
The SPINDLE HEAD guides and supports the spindle and can be adjusted vertically to provide maximum support near the spindle socket.

The SPINDLE is a splined shaft with a Morse taper socket for holding the drill. The spline permits vertical movement of the spindle while it is rotating.

The DRIVE MECHANISM includes the motor, speed and feed change gears, and mechanical controls.

HEAVY DUTY DRILL PRESSES (BOX COLUMNS) are normally used in drilling large holes. They differ from the general purpose drill presses in that the worktable moves vertically only. The worktable is firmly gibbed to vertical ways or tracks on the front of the column and is further supported by a heavy adjusting screw from the base to the bottom of the table. As the table can be moved vertically only, it is necessary to position the work for each hole.

The SENSITIVE DRILL PRESS shown in figure 5-26 is used for drilling small holes in work under conditions which make it necessary for the operator to "feel" what the cutting tool is doing. The tool is fed into the work by a very simple device—a lever, a pinion and shaft, and a rack which engages the pinion. These drills are nearly always belt-driven because the vibration caused by gearing would be undesirable. Sensitive drill presses are used in drilling holes less than one-half inch in diameter. The



11.10X
Figure 5-26.—Sensitive drill press.

high-speed range of these machines and the holding devices used, make them unsuitable for heavy work.

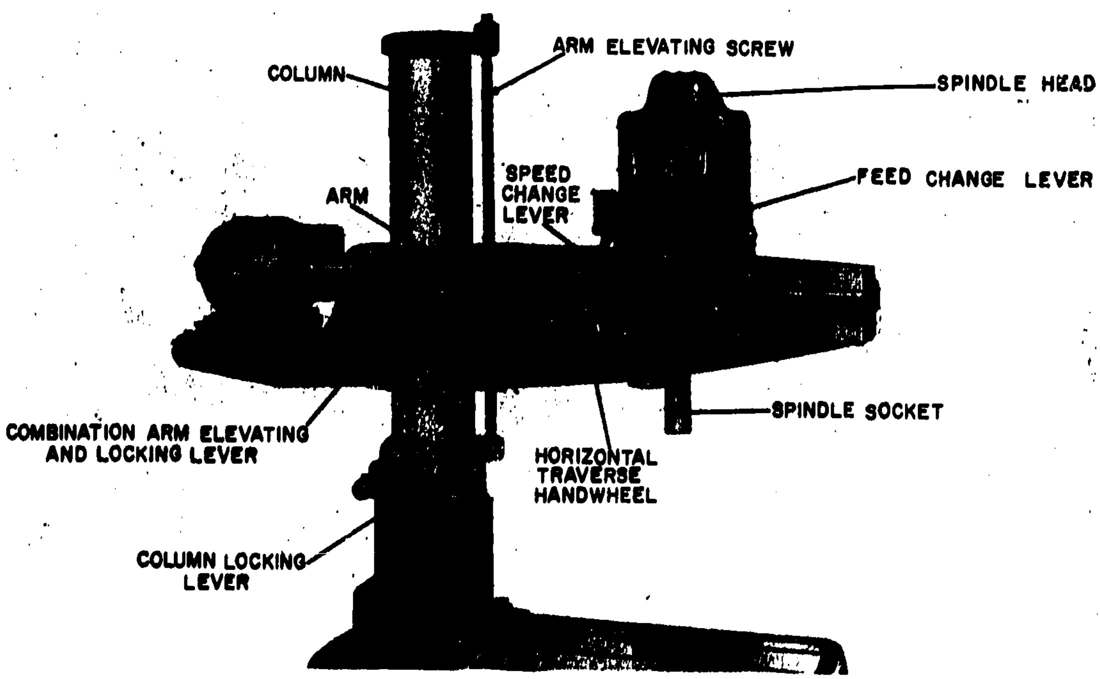
The RADIAL DRILL PRESS, shown in figure 5-27, has a spindle head on an arm that can be rotated axially on the column. The spindle head may be traversed horizontally along the ways of the arm, and the arm may be moved vertically in relation to the column. This machine is especially useful when the workpiece is bulky or heavy or when many holes can be drilled with one setup. The arm and spindle are designed so that the drill can be positioned easily over the layout of the workpiece.

Some operational features that are common to most drilling machines are: (1) high- and low-speed ranges provided from either a two-speed drive motor or a low-speed drive gear; (2) a reversing mechanism for changing the direction of rotation of the spindle by either a reversible motor or a reversing gear in the drive gear train; (3) automatic feed mechanisms which are driven from the spindle and feed the cutting tool at a selected rate per revolution of the spindle; (4) depth setting devices which permit the operator to preset the required depth of penetration of the cutting tool; and (5) coolant systems to provide lubrication and coolant to the cutting tool.

On other machines the control levers may be placed in different positions; however, they serve the same purposes as those shown. In using the locking clamps to lock or "dog down" the table or head of a drill after it is positioned over the work, make sure that the locking action does not cause the drill or work to move slightly out of position.

TWIST DRILL

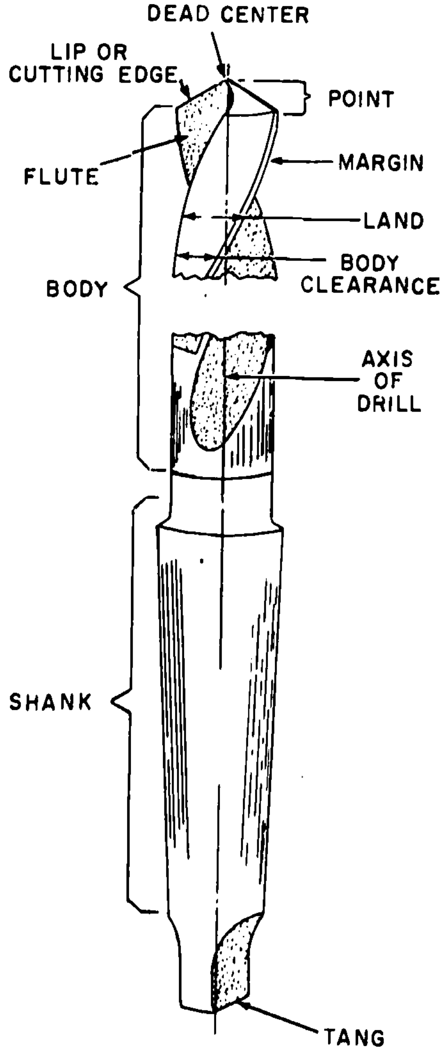
The twist drill is the tool generally used for drilling holes in metal. This drill is formed either by forging and twisting grooves in a flat strip of steel or by milling a cylindrical piece of steel.



11.11X

Figure 5-27.—Radial drill press.

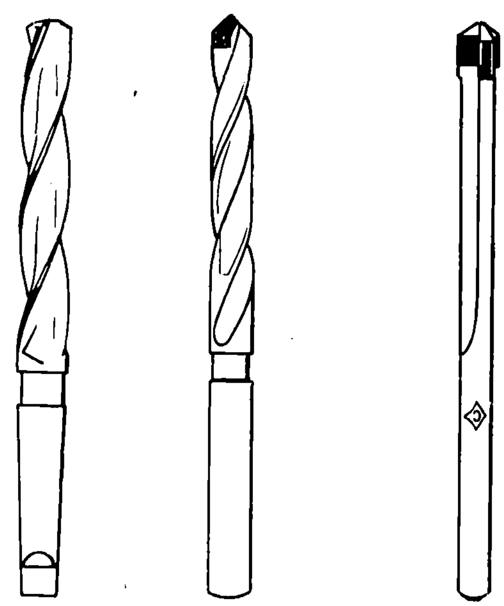
In figure 5-28 you see the principal parts of a twist drill: the BODY, the SHANK, and the POINT. The portion of the LAND behind the MARGIN is relieved to provide BODY CLEARANCE. The body clearance assists in reducing friction during drilling. The LIP is the cutting edge, and on the CONE of the drill is the area called the LIP CLEARANCE. DEAD CENTER is the sharp edge located at the tip end of the drill. It is formed by the intersection of the cone-shaped surfaces of the point and should always be in the exact center of the axis of the drill. The point of the drill should not be confused with the dead center. The point is the entire cone-shaped surface at the cutting end of the drill. The WEB of the drill is the metal column which separates the flutes. It runs the entire length of the body between the flutes and gradually increases in thickness toward the shank, giving additional rigidity to the drill.



44.20(11)X
Figure 5-28.—The parts of a twist drill.

The TANG is found only on tapered shank tools. It fits into a slot in the socket or spindle of the drill press and bears a portion of the driving strain. Its principal purpose is to make it easy to remove the drill from the socket with the aid of a drill drift. (NEVER use a file or screwdriver to do this job.)

The SHANK is the part of the drill which fits into the socket, spindle, or chuck of the drill press. The types of shanks that are most often found in Navy machine shops are the Morse taper shank, shown in figures 5-28 and 5-29A



28.318
Figure 5-29.—Twist drills: A. Three-fluted core drill; B. Carbide tipped drill with two helical flutes; C. Carbide tipped die drill with two flutes parallel to drill axis.

and the straight shank, shown in figures 5-29B and 5-29C.

Twist drills are made from several different materials. Drills made from high-carbon steel are available; however, the low cutting speed required to keep this type of drill from becoming permanently dull limits their use considerably. Most of the twist drills that you will use are made from high-speed steel and will have two flutes (fig. 5-28).

Core drills (fig. 5-29A) have three or more flutes and are used to enlarge a cast or previously drilled hole. Core drills are more efficient and more accurate when used to enlarge a hole than the standard two-fluted drill. Core drills are made from high-speed steel.

A carbide-tipped drill (fig. 5-29B), which is similar in appearance to a standard two-fluted drill with carbide inserts mounted along the lip or cutting edge, is used for drilling nonferrous metals, cast iron, and cast steel at high speeds. These drills are not designed for drilling steel and alloy metals.

A carbide-tipped die drill, or spade drill as it is often called (fig. 5-29C), has two flutes that run parallel to the axis of the drill as opposed to the helical flutes of the standard two-fluted drill. This drill can be used to drill holes in hardened steel.

A standard two-fluted drill made from cobalt high-speed steel is superior in cutting efficiency and wear resistance than the high-speed steel drill and is used at a cutting speed between the speed recommended for a high-speed steel drill and a carbide-tipped drill.

A solid carbide drill with two helical flutes is also available and can be used to drill holes in hard and abrasive metal where no sudden impact will be applied to the drill.

Drill sizes are indicated in three ways: by measurement, letter, and number. The nominal measurements range from 1/16 to 4 inches or larger, in 1/64-inch steps. The letter sizes run from "A" to "Z" (0.234 to 0.413 inch). The number sizes run from No. 80 to No. 1 (0.0135 to 0.228 inch).

Before putting a drill away, wipe it clean and then give it a light coating of oil. Do not leave drills in a place where they may be dropped or where heavy objects may fall on them. Do not place drills where they will rub against each other.

DRILLING OPERATIONS

Using the drill press is one of the first skills you will learn as a Machinery Repairman. Although a drill press is relatively simpler to operate and understand than other machine tools in the shop, the requirements for accuracy and efficiency in its use are no less strict. To achieve skill in drilling operations, you must have a knowledge of feeds and speeds, how the work is held, and how to ensure accuracy.

Speeds, Feeds, and Coolants

The cutting speed of a drill is expressed in feet per minute (fpm). This speed is computed by multiplying the circumference of the drill (in inches) by the revolutions per minute (rpm) of the drill. The result is then divided by 12. For example, a 1/2-inch drill, which has a circumference of approximately 1 1/2 inches, turned at 100 rpm has a surface speed of 150 inches per minute. To obtain fpm, divide this figure by 12 which results in a cutting speed of approximately 12 1/2 feet per minute.

The correct cutting speed for a job depends on many variable factors. The machinability of a metal, any heat treatment processes such as hardening, tempering, or normalizing, the type of drill used, the type and size of the drilling machine, the rigidity of the setup, the finish and accuracy required, and whether or not a cutting fluid is used are the main factors that you must consider when selecting a cutting speed for drilling. The following cutting speeds are recommended for using high-speed steel twist drills. Carbon steel drills should be run at one-half these speeds, while carbide may be run at two to three times these speeds. As you gain experience in using twist drills, you will be able to vary the speeds to suit the job you are doing.

Low carbon steel	80-110 fpm
Medium carbon steel	70-80 fpm
High carbon steel	50-60 fpm
Alloy steel	50-70 fpm
Corrosion-resistant steel (stainless)	30-40 fpm
Brass	200-300 fpm
Bronze	200-300 fpm
Monel	40-50 fpm
Aluminum	200-300 fpm
Cast iron	70-150 fpm

The speed of the drill press is given in rpm. Tables giving the proper rpm at which to run a drill press for a particular metal are usually available in the machine shop, or they may be found in machinists' handbooks. A formula may be used to determine the rpm required to give a specific rate of speed in fpm for a specific size drill. For example, if you wish to drill a hole 1 inch in diameter at the speed of 50 fpm, you would compute the rpm as follows:

$$\text{rpm} = \frac{\text{fpm} \times 12}{\pi \times D}$$

$$= \frac{50 \times 12}{3.1416 \times 1}$$

$$= \frac{600}{3.1416}$$

$$= 190$$

where

fpm = required speed in feet per minute
π = 3.1416
12 = constant
D = diameter of drill in inches

The feed of a drill is the rate of penetration into the work for each revolution. Feed is expressed in thousandths of an inch per revolution. In general, the larger the drill, the heavier the feed that may be used. Always decrease feed pressure as the drill breaks through the bottom of the work to prevent drill breakage and rough edges. The rate of feed depends on the size of the drill, the material being drilled, and the rigidity of the setup.

Use the following feed rates, given in thousandths of an inch per revolution (ipr), as a general guide until the experience that you will gain allows you to determine the most efficient feed rate for each different job.

Drill Diameter	IPR
No. 80 to 1/8 inch	0.001-0.002
1/8 inch to 1/4 inch	0.002-0.004
1/4 inch to 1/2 inch	0.004-0.007
1/2 inch to 1 inch	0.007-0.015
Greater than 1 inch	0.015-0.025

The lower feed rate given for each range of drill sizes should be used for the harder materials such as tool steel, corrosion-resistant steel and alloy steel, while the higher feed rate should be used for brass, bronze, aluminum, and other soft metals.

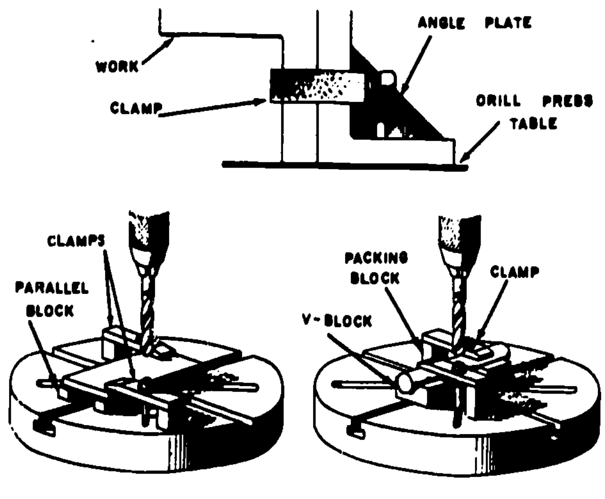
It is usually necessary to use a cutting oil or coolant for drilling carbon steel, alloy steel, corrosion-resistant steel and certain nonferrous metals such as Monel. Aluminum, brass, cast iron, bronze and similarly soft metals may be drilled dry unless a high drilling speed and feed are used. A mineral-lard oil should be used for the exceptionally hard metals; however, soluble oil can be used for most drilling operations.

Holding the Work

Before drilling, be sure your work is well clamped down. On a sensitive drill press you will probably have to use a drill vise and center the work by hand. Because the work done on this drill press is comparatively light, the weight of the vise is sufficient to hold the work in place.

The larger drill presses have slotted tables to which work of considerable weight can be bolted or clamped. T-bolts, which fit into the T-slots on the table, are used for securing the work. Various types of clamping straps, shown in figure 5-30, also can be used. (Clamping straps are also identified as clamps or dogs.) The U-strap is the most convenient for many setups because it has a larger range of adjustment.

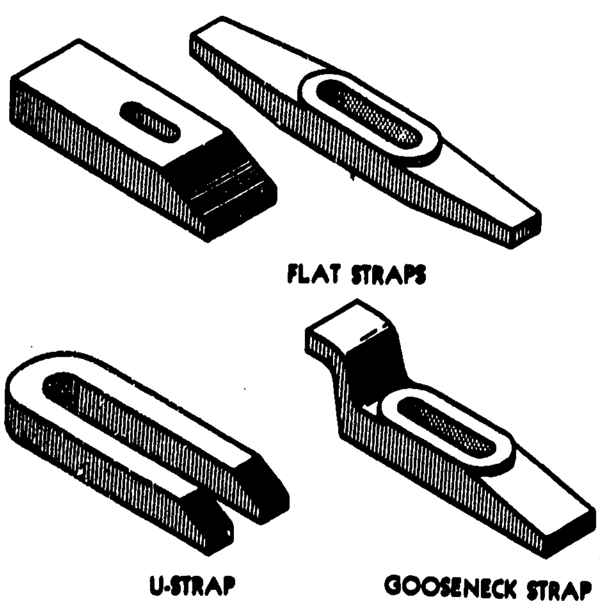
It is often necessary to use tools such as steel parallels, V-blocks, and angle plates for supporting and holding the work. Steel parallels are used to elevate the work above the table so that you can better see the progress of the drill. V-blocks are used for supporting round stock, and angle plates are used to support work where a hole is to be drilled at an angle to another surface. Some examples of setups are shown in figure 5-31.



11.16X
Figure 5-31.—Work mounted on table.

Drilling Hints

To ensure accuracy in drilling, position the work accurately under the drill, and use the proper techniques to prevent the drill from starting offcenter or from moving out of alignment during the cut. Here are some hints that will aid you in correctly starting and completing a drilling job.



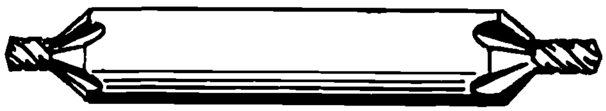
11.15X
Figure 5-30.—Common types of clamping straps.

1. Before setting up the machine, wipe all foreign matter from the spindle and table of the machine. A chip in the spindle socket will cause the drill to have a wobbling effect which tends to make the hole larger than the drill. Foreign matter on the work holding device under the workpiece tilts it in relation to the spindle, causing the hole to be out of alignment.

2. Center punch the work at the point to be drilled. Position the center-punched workpiece under the drill; use a dead center inserted in the spindle socket to align the center-punch mark on the workpiece directly under the axis of the spindle.

3. Bring the spindle with the inserted center down to the center-punch mark and hold it in place lightly while fastening locking clamps or dogs. This will prevent slight movement of the workpiece, table, or both when they are clamped in position.

4. Insert a center drill (fig. 5-32) in the spindle and make a center hole to aid in starting the drill. This is not necessary on small drills on which the dead center of the drill is smaller than



28.57X
Figure 5-32.—Combined drill and countersink (center drill).

the center-punch mark, but on large drills it will prevent the drill from "walking" away from the center-punch mark. This operation is especially important in drilling holes on curved surfaces.

5. Using a drill smaller than the required size to make a pilot hole will increase accuracy by eliminating the need of the dead center of the finishing drill to do any cutting, decreasing the pressure required for feeding the finishing drill and decreasing the width of cut taken by each drill. In drilling holes over 1 inch in diameter, you may need to use more than one size of pilot drill to increase the size of the hole by steps until the finished size is reached.

6. If the outer corners of the drill (margin) appear to be wearing too fast or have a burnt look, the drill is going too fast.

7. If the cutting edges (lip) chip during drilling, too much lip clearance has been ground into the drill, or too heavy a feed rate is being used.

8. A very small drill will break easily if the drill is not going fast enough.

9. When a hole being drilled is more than three or four times the drill diameter in depth, the drill should be backed out frequently to clear the chips from the flutes.

10. If the drill becomes hot quickly, is difficult to feed, squeals when being fed and produces a rough finish in the hole, it has become dull and requires resharpening.

11. If the drill has cutting edges of different angles or unequal length, the drill will cut with only one lip and will wobble in operation, resulting in an excessively oversized hole.

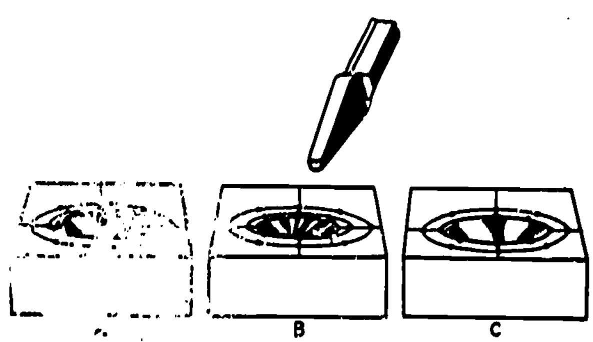
12. If the drill will not penetrate the work, insufficient or no lip clearance has been ground into the drill.

13. The majority of drilled holes will be oversized regardless of the care taken to ensure a good setup. Generally, you can expect the oversize to average an amount equal to 0.004 inch times the drill diameter plus 0.003 inch. For example, a 1/2-inch drill can be expected to result in a hole approximately 0.505 in diameter ([0.004 X 0.500] + 0.003). This amount can vary up or down depending on the condition of the drilling machine and the twist drill.

Correcting Offcenter Starts

A drill may start offcenter because of improper center drilling, careless starting of the drill, improper grinding of the drill point, or hard spots in the metal. To correct this condition, take a half-round chisel and cut a groove on the side of the hole toward which the center is to be drawn. (See fig. 5-33.) The depth of this groove depends upon the eccentricity (deviation from center) of the partially drilled hole with the hole to be drilled. When the groove is drilled out, lift the drill from the work and check the hole for concentricity with the layout line. Repeat the operation until the edge of the hole and the layout line are concentric. When the drill begins to cut its full diameter, the prick-punch marks on the layout should be evenly cut at the centers.

When using this method to correct an offcenter condition, you must be very careful that the cutting edge or lip of the drill does not grab in the chisel groove. Generally, you should use very light feeds until the new center point is established. (Heavy feeds cause a sudden bite in the groove which may result in the work being



11.17X
Figure 5-33.—Using a half-round chisel to guide a drill to the correct center.

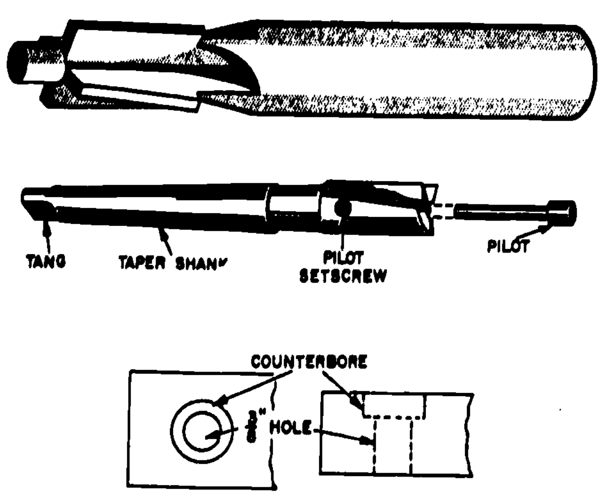
pulled out of the holding device, or the drill being broken.)

Counterboring, Countersinking, and Spotfacing

A counterbore is a drilling tool used in the drill press to enlarge portions of previously drilled holes to allow the heads of fastening devices to be flush with or below the surface of the workpiece. The parts of a counterbore that distinguish it from a regular drill are a pilot, which aligns the tool in the hole to be counterbored, and the cutting edge of the counterbore, which is flat so that a flat surface is left at the bottom of the cut, enabling fastening devices to seat flat against the bottom of the counterbored hole.

Figure 5-34 shows two types of counterbores and an example of a counterbored hole. The basic difference between the counterbores illustrated is that one has a removable pilot and the other does not. A counterbore with provisions for a removable pilot can be used in counterboring a range of hole sizes by simply using the appropriate size pilot. The use of the counterbore with a fixed pilot is limited to holes of the same dimensions as the pilot.

Countersinks are used for seating flathead screws flush with the surface. The basic difference in countersinking and counterboring is that a countersink makes an angular sided recess, while the counterbore forms straight sides. The angular point of the countersink acts as a guide to center the tool in the hole being countersunk. Figure 5-35 shows two common types of countersinks.



28.58

Figure 5-34.—One type of counterbore.

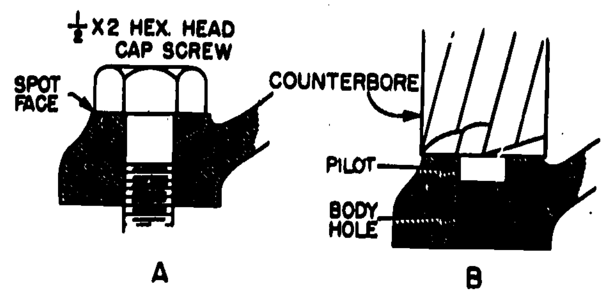
28.59X

Figure 5-35.—Countersinks.

Spotfacing is an operation that cleans up the surface around a hole so that a fastening device can be seated flat on the surface. This operation is commonly required on rough surfaces that have not been machined and on the circumference of concave or convex workpieces. Figure 5-36 shows an example of spotfacing and the application of spotfacing in using fastening devices. This operation is commonly accomplished by using a counterbore.

Reaming

In addition to drilling holes, the drill press may be used for reaming. For example, when specifications call for close tolerances, the hole must be drilled slightly undersize and then reamed to the exact dimension. Reaming is also done to remove burrs in a drilled hole or to



28.60

Figure 5-36.—Examples of spotfacing.

enlarge a previously used hole for new applications.

Machine reamers have tapered shanks so that they fit the drilling machine spindle. Be sure not to confuse them with hand reamers, which have straight shanks. Hand reamers will be ruined if used in a machine.

There are many types of reamers, but the ones used most extensively are the straight fluted, the taper, and the expansion types. They are illustrated in figure 5-37.

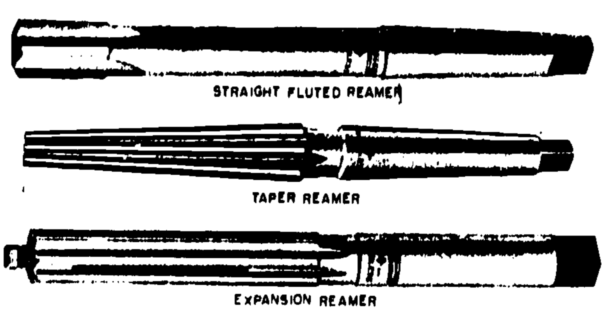
The STRAIGHT FLUTED REAMER is made to remove small portions of metal and to cut along the edges to bring a hole to close tolerance. Each tooth has a rake angle which is comparable to that on a lathe tool.

The TAPER PIN REAMER has a tapered body and is used to smooth and true tapered holes and recesses. The taper pin reamer is tapered at 1/4 inch per foot.

The EXPANSION REAMER is especially useful in enlarging reamed holes by a few thousandths of an inch. It has a threaded plug in the lower end which expands the reamer to various sizes.

The steps outlined below should be followed in reaming:

1. Drill the hole about 1/64 inch less than the reamer size.
2. Substitute the reamer in the drill press without removing the work or changing the position of the work.



5.10X

Figure 5-37.—Reamers.

3. Adjust the machine for the proper spindle speed. (Reamers should turn at about one-half the speed of the twist drill.)

4. Use a cutting oil to ream. Use just enough pressure to keep the reamer feeding into the work; excessive feed may cause the reamer to dig in and break.

5. The starting end of a reamer is slightly tapered; always run it all the way through the hole. NEVER RUN A REAMER BACKWARD because the edges are likely to break.

Tapping

Special attachments that permit cutting internal screw threads with a tap driven by the drilling machine spindle can save considerable time when a number of identically sized holes must be threaded. The attachment is equipped with a reversing device that automatically changes the direction of rotation of the tap when either the tap strikes the bottom of the hole or a slight upward pressure is applied to the spindle down-feed lever. The reversing action takes place rapidly, permitting accurate control over the depth of the threads being cut. A spiral-fluted tap should be used to tap a through hole while a standard straight-fluted plug tap can be used in a blind hole. A good cutting oil should always be used in tapping with a machine.

DRILLING ANGULAR HOLES

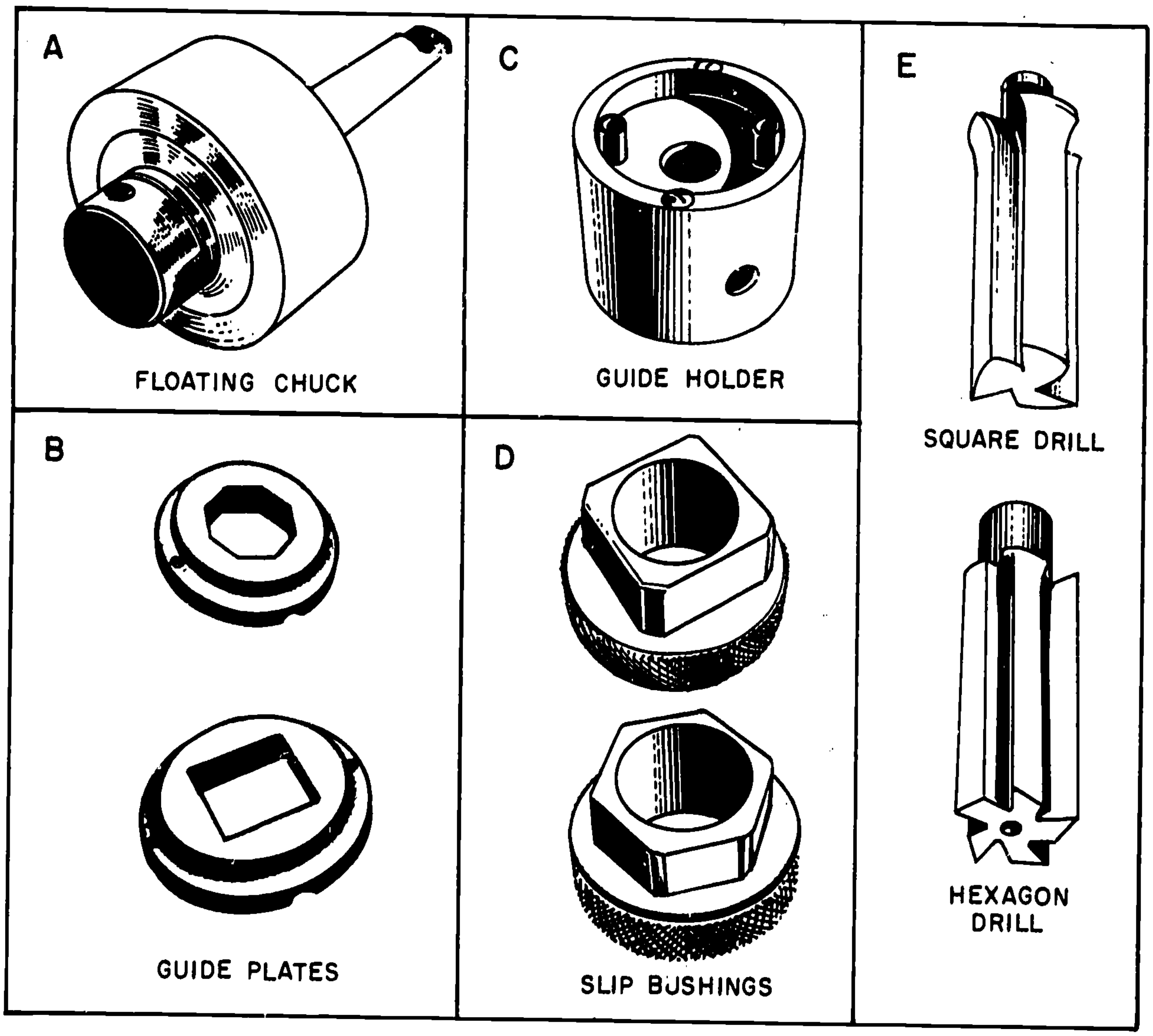
An angular hole is a hole having a series of straight sides of equal length. A square (4-sided), a hexagon (6-sided), a pentagon (5-sided), and an octagon (8-sided) are examples of angular holes. An angular hole that goes all the way through a part can be made easily by using a broach; however, a blind hole, one in which the angular hole does not go all the way through the part, cannot be made with a broach. There are two methods available to you for machining a blind angular hole. One method, the shaper, will be covered later in Chapter 12. The second method, drilling the angular hole in a drill press or on a lathe, will be briefly described in the following paragraphs.

EQUIPMENT

The equipment required to drill angular holes is specialized and is designed to accomplish only this particular operation. The machining process, known as the WATTS METHOD, was developed by the Watts Bros. Tool Works, Incorporated and the required equipment is patented and manufactured exclusively by that company. A brief description of the equipment is included in the following paragraphs. A complete description of the equipment and its use is available from the manufacturer when the equipment is ordered.

Chuck

The chuck (fig. 5-38A) used in drilling angular holes is of an unusual design in that while it holds the drill in a position parallel to the



28.319
Figure 5-38.—Equipment for drilling angular holes. A. Chuck; B. Guide plate; C. Guide holder; D. Slip bushing; E. Angular drill

spindle of the lathe or drill press and prevents it from revolving, it allows the drill to float freely so that the flutes can follow the sides of the angular hole in the guide plate. The chuck is available with a Morse taper shank to fit most lathes and drill presses. There are several different sizes of chucks, each capable of accepting drills for a given range of hole sizes.

Guide Plates

The guide plate (fig. 5-38B) is the device that causes the drill to make an angular hole. The free-floating action of the chuck allows the drill to randomly follow the straight sides and corners of the guide plate as it is fed into the work. Attach the guide plate to a guide holder when using a lathe and directly to the work when using a drill press. A separate guide plate is required for each different shape and size hole.

Guide Holder

The guide holder (fig. 5-38C), as previously stated, holds the guide plate and is placed over the outside diameter of the work and locked in place with a setscrew. The guide holder is used when the work is being done in a lathe and is not required for drill press operations.

Slip Bushings

Prior to actually drilling with the angular hole drill, you must drill a normal round hole in the center of the location where the angular hole will be located. This pilot hole reduces the pressure that would otherwise be required to feed the angular drill and ensures that the angular drill will accurately follow the guide plate. In a lathe, you need only to drill a hole using the tailstock since it and the chuck will automatically center the pilot hole. In a drill press, a method must be devised to assist you in this alignment of the pilot hole. A slip bushing will accomplish the job quickly and accurately. The slip bushing (fig. 5-38D) fits into the guide plate and has a center hole which is the correct size for the pilot hole of the particular size angular hole being drilled. Then, position the correct drill so that it enters the hole in the slip bushing and drill the pilot hole.

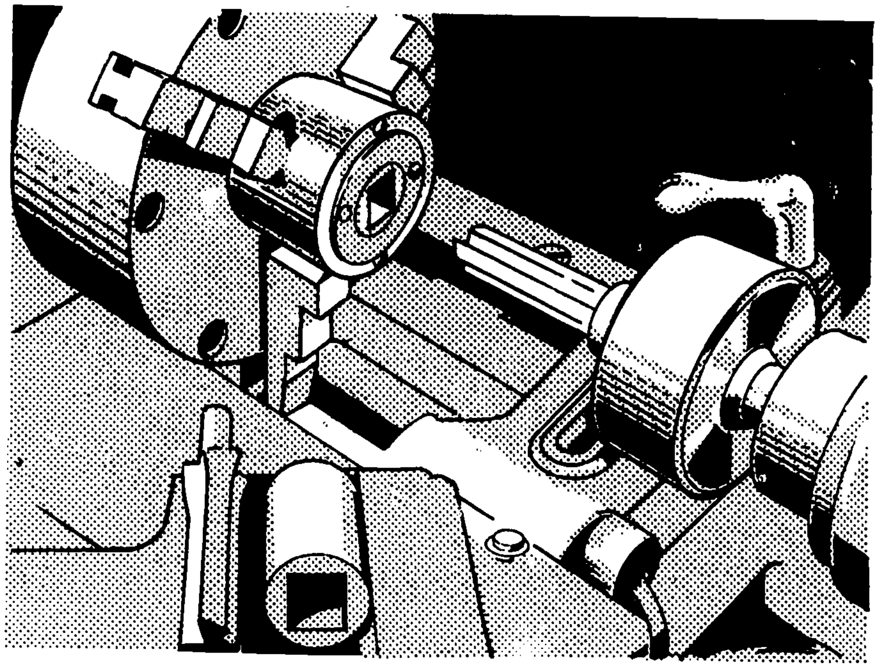
Angular Drill

The angular drills (fig. 5-38E) are straight fluted and have one less flute or cutting lip than the number of sides in the angular hole it is designed to drill. The drills have straight shanks with flats machined on them to permit securing them in the floating chuck with setscrews. The cutting action of the drill is made by the cutting lips or edges on the front of the drill.

OPERATION

The procedure for drilling an angular hole is similar to drilling a normal hole, differing only in the preliminary steps required in setting the job up. The feeds and speeds for drilling angular holes should be slower than those recommended for drilling a round hole the same size. Obtain specific recommendations concerning feeds and speeds from the information provided by the manufacturer. Use a coolant to keep the drill cool and help flush away the chips. The following procedures apply when the work is being done on a lathe. See figure 5-39 for an example of a lathe setup.

1. Place the work to be drilled in the lathe chuck. The work must have a cylindrical outside diameter and the intended location of the angular hole must be in the center of the work.
2. Place the guide holder over the outside diameter of the work and tighten the setscrew. If the bore in the back of the guide holder is larger than the diameter of the work, make a sleeve to adapt the two together. If the part to be drilled is short, place it in the guide holder and place the guide holder in the chuck.
3. Drill the pilot hole at this time. The size of the pilot hole should be slightly smaller than the distance across the flats of the angular hole. The manufacturer makes specific recommendations on pilot hole sizes.
4. Attach the guide plate to the guide holder.
5. Mount the floating chuck in the lathe tailstock spindle and place the drill in the chuck. Tighten the setscrews to hold the drill securely.
6. You are now ready to drill the angular hole. Do not force the drill into the work too rapidly, and use plenty of coolant.



28.320

Figure 5-39.—Lathe setup for drilling an angular hole.

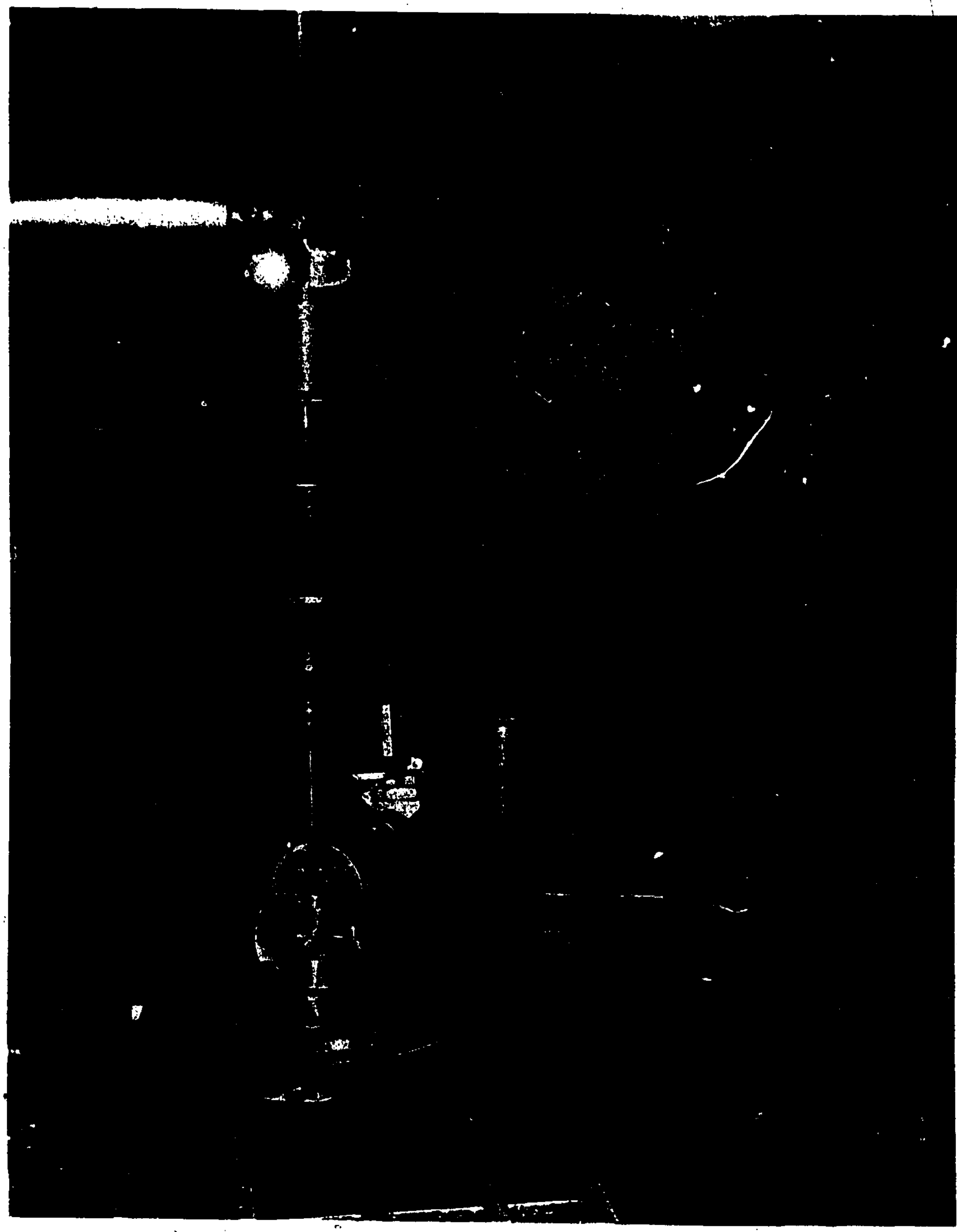
The setup for drilling an angular hole using a drill press differs in that instead of using a guide holder, clamp the guide plate directly to the work and drill the pilot hole by using a slip bushing placed in the guide plate to ensure alignment. Once you have positioned the work under the drill press spindle and drilled the pilot hole, do not move the setup. Any movement will result in misalignment of the work and the angular drill.

METAL DISINTEGRATORS

There are occasions when a broken tap or a broken hardened stud cannot be removed by the usual removal methods previously covered. To accomplish removal without damaging the part, use a metal disintegrator. This machine disintegrates a hole through the broken tap or stud by the use of an electrically charged electrode that vibrates as it is fed into the work. The part that is to be disintegrated and the mating part that it is screwed into must be made from a material that will conduct electricity. Figure 5-40 shows a disintegrator removing a broken stud.

Obtain the specific operating procedures for the metal disintegrator from the reference material furnished by the manufacturer; however, there are several steps involved in setting up for a disintegrating job that are common to most of the models of disintegrators found aboard Navy ships.

Setting up the part to be disintegrated is the first step that must be done. Some disintegrator models have a built-in table with the disintegrating head mounted above it in a fashion similar to a drill press. On a machine such as this, you need only to bolt the part securely to the table, ensuring that the part makes good contact so that an electrical ground is provided. Align the tap or stud to be removed square with the table so that the electrode will follow the center of the hole correctly. Misalignment could result in the electrode leaving the tap or stud and damaging the part. A machinist's square laid on the table or a dial indicator mounted on the disintegrating head



28.321

Figure 5-40.—Metal disintegrator removing a broken stud.

can be used to help align the part. If the part will not make an electrical ground to the table or if the model of machine being used is designed as an attachment to be mounted in a drill press spindle, the disintegrator's auxiliary ground cable must be attached to the part.

Selection of the correct electrode depends on the diameter and length of the part to be removed. As a general rule, the electrode should be large enough in diameter to equal the smallest diameter of a tap (the distance between the bottom of opposite flutes). To remove a stud, the electrode must not be so large that it could burn or damage the part if a slight misalignment is present. A scribe and a small magnet can be used to remove any of the stud material not disintegrated.

A free-flowing supply of clean soluble oil is an essential part of the disintegrating operation. The coolant is pumped from a sump to the disintegrating head and then through the electrode, which is hollow, to the exact point of the disintegrating action.

The specific controls which must be set may vary among the different machines; however, most have a control to start the disintegrating head vibrating and a selector switch for the heat or power setting. The position of this switch is dependent on the diameter of the electrode being used. Some models have an automatic feed control that regulates the speed that the electrode penetrates the part to be removed. Regardless of whether the feed is automatic or manual, it must not be advanced so fast that it stops the disintegrating head and the electrode from vibrating. If this happens, the disintegrating action will stop and the electrode could be bent or broken.

CHAPTER 6

OFFHAND GRINDING OF TOOLS

One requirement for advancement in the MR rating is to demonstrate ability to grind and sharpen some of the tools used in the machine shop. Equipment used for this purpose includes bench, pedestal, carbide, and chip breaker grinders and precision grinding machines. This chapter contains information on the use of these grinders and the grinding of small tools while using the offhand grinding technique. (Precision grinding machines will be discussed in a later chapter.)

Grinding is the removal of metal by the cutting action of an abrasive. In offhand grinding you hold the workpiece in your hand and position it as needed while grinding. To grind accurately and safely, using the offhand method, you must have experience and practice. In addition, you must know how to install grinding wheels on pedestal and bench grinders, how to sharpen or dress them, and you must know the safety precautions concerning grinding.

To properly grind small handtools, single-edged cutting tools, and twist drills, you must know the terms used to describe the angles and surfaces of the tools. You must also know the operations for which each tool is used, and you must know the composition of the tool material.

GRINDING SAFETY

The grinding wheel is a fragile cutting tool which operates at high speeds. Therefore, the safe operation of bench and pedestal grinders is as important to you as learning grinding techniques. Observance of safety precautions, posted on or near all grinders used by the Navy, is mandatory for the safety of the operator and the safety of personnel in the nearby vicinity.

What are the most common sources of injury during grinding operation? Hazards leading to eye injury caused by grit generated by the grinding process are the most common and the most serious. Abrasions caused by bodily contact with the wheel are quite painful and can be serious. Cuts and bruises caused by segments of an exploded wheel, or a tool "kicked" away from the wheel are other sources of injury. Cuts and abrasions can become infected if not protected from grit and dust produced during grinding.

Safety in using bench and pedestal grinders is primarily a matter of using common sense and concentrating on the job at hand. Each time you start to grind a tool, stop briefly to consider how observance of safety precautions and the use of safeguards protect you from injury. Consider the complications that could be caused by loss of your sight, or loss or mutilation of an arm or hand.

Some guidelines for safe grinding practices are:

- Read posted safety precautions before starting to use a machine. In addition to refreshing your memory about safe grinding practices, this gets your mind on the job at hand.

- Secure all loose clothing and remove rings or other jewelry.

- Inspect the grinding wheel, wheel guards, the toolrest, and other safety devices to ensure they are in good condition and positioned properly. Set the toolrest so that it is within 1/8 inch of the wheel face and level with the center of the wheel.

- Transparent shields, if installed, should be clean and properly adjusted. Transparent shields do not protect against dust and grit that may get around a shield. You must ALWAYS wear goggles while grinding. Goggles with side shield give the best eye protection.

- Stand aside when starting the grinder motor until it has run for 1 minute. This prevents injury in case the wheel explodes from a defect that you did not notice.

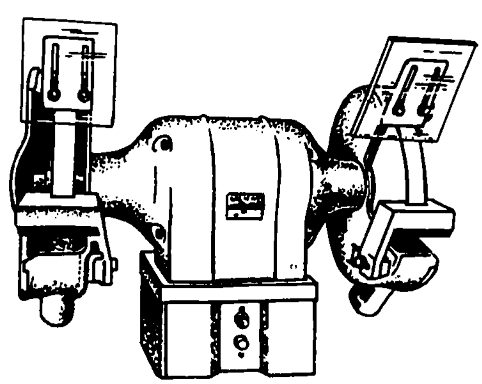
- Use light pressure when starting grinding; too much pressure on a cold wheel may cause failure.

- On bench and pedestal grinders, grind only on the face or periphery of a grinding wheel unless the grinding wheel is specifically designed for side grinding.

- Use a coolant to prevent the work from overheating.

BENCH AND PEDESTAL GRINDERS

Bench grinders (fig. 6-1) are small self-contained grinders which are usually mounted on a workbench. They are used for grinding and sharpening small tools such as lathe, planer, and shaper cutting tools; twist drills; and handtools such as chisels and center punches. These grinders do not have installed coolant systems; however, a container of water is usually mounted on the front of the grinder.



28.328

Figure 6-1.—Bench grinder.

Grinding wheels up to 8 inches in diameter and 1 inch in thickness are normally used on bench grinders. A wheel guard encircles the grinding wheel except for the work area. An adjustable toolrest, steadies the workpiece and can be moved in or out or swiveled to adjust to grinding wheels of different diameters. An adjustable eye shield made of safety glass should be installed on the upper part of the wheel guard and positioned to deflect the grinding wheel particles away from the machine operator.

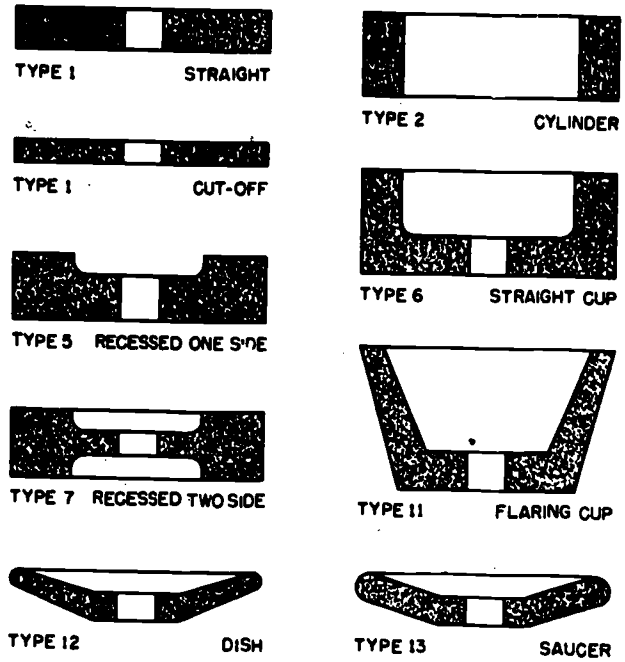
Pedestal grinders are usually heavy duty bench grinders which are mounted on a pedestal fastened to the deck. In addition to the features of the bench grinder, pedestal grinders normally have a coolant system which includes a pump, storage sump, and a hose and fittings to regulate and carry the coolant to the wheel surface. Pedestal grinders are particularly useful for rough grinding such as "snagging" castings. Figure 6-2 shows a pedestal grinder in use.

GRINDING WHEELS

A grinding wheel is composed of two basic elements: (1) the abrasive grains, and (2) the bonding agent. The abrasive grains may be compared to many single point tools embedded in a toolholder or bonding agent. Each of these grains removes a very small chip from the workpiece as it makes contact on each revolution of the grinding wheel.

An ideal cutting tool is one that will sharpen itself when it becomes dull. This, in effect, is what happens to the abrasive grains. As the individual grains become dull, the pressure that is generated on them causes them to fracture and present new sharp cutting edges to the work. When the grains can fracture no more, the pressure becomes too great and they are released

28.61
Figure 6-2.–Grinding on a pedestal grinder.



28.309
Figure 6-3.–Grinding wheel shapes.

rom the bond, allowing new sharp grains to ontact the work.

IZES AND SHAPES

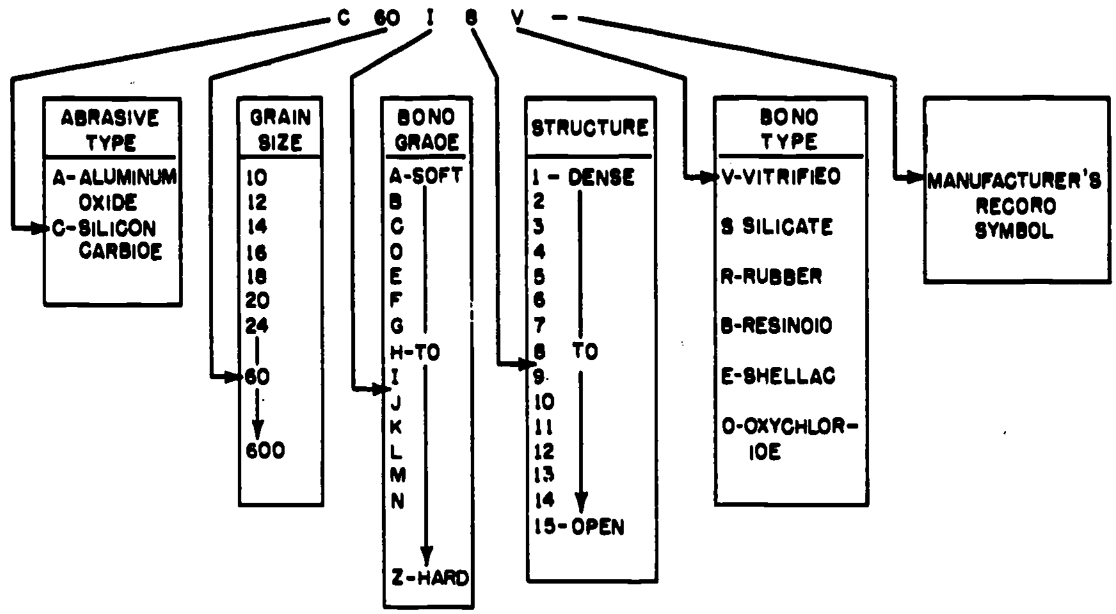
Grinding wheels come in various sizes and hapes. The size of a grinding wheel is etermined by its diameter in inches, the iameter of the spindle hole, and the width of he face of the wheel. All the shapes of grinding heels are too numerous to list in this manual, ut figure 6-3 shows most of the frequently used heel shapes. The type numbers are standard nd are used by all manufacturers. The shapes re shown in cross-sectional views. The specific b will dictate the shape of the wheel to be sed.

HEEL MARKINGS AND COMPOSITION

Grinding wheel markings are comprised of x stations. Figure 6-4 illustrates the standard arking. The following information breaks down the marking and explains each station–type of abrasive, grain size, bond grade, structure, type of bond and the manufacturer's record symbol. Study this information carefully as it will be invaluable to you in making the proper wheel selection for each grinding job you attempt.

Type of Abrasive

The first station of the wheel marking is the abrasive type. There are two types of abrasives: natural and manufactured. Natural abrasives, such as emery, corundum, and diamond, are used only in honing stones and in special types of grinding wheels. The common manufactured abrasives are aluminum oxide and silicon carbide. They have superior qualities and are more economical than natural abrasives. Aluminum oxide (designated by the letter A) is used for grinding steel and steel alloys and for heavy duty work such as cleaning up steel



28.310

Figure 6-4.—Standard marking system for grinding wheels (except diamond).

castings. Silicon carbide (designated by the letter C), which is harder but not as tough as aluminum oxide, is used mostly for grinding nonferrous metals and carbide tools. The abrasive in a grinding wheel comprises about 40% of the wheel.

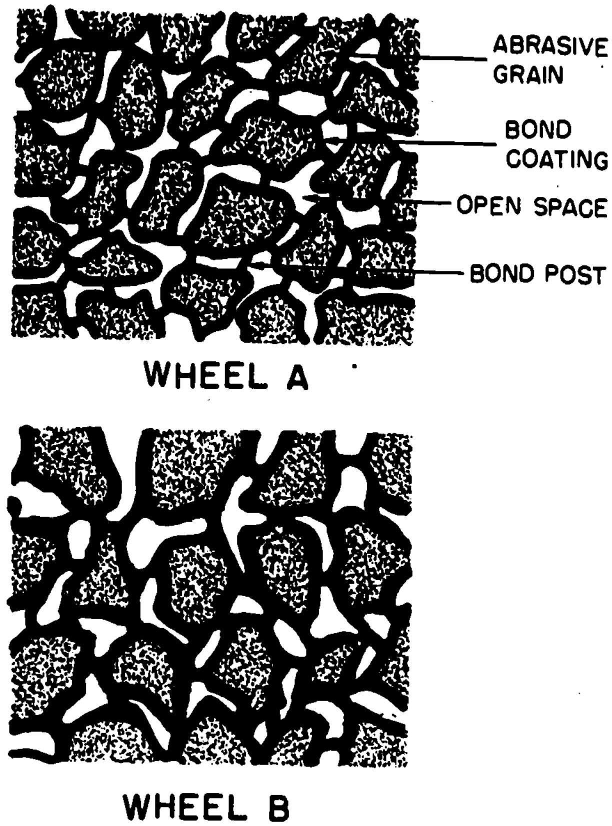
Grain Size

The second station on the grinding wheel marking is the grain size. Grain sizes range from 10 to 600. The size is determined by the size of mesh of a sieve through which the grains can pass. Grain size is rated as follows: Coarse: 10,12, 14, 16, 18, 20, 24; Medium: 30, 36, 46, 54, 60; Fine: 70, 80 90, 100, 120, 150, 180; and Very Fine: 220, 240, 280, 320, 400, 500, 600. Grain sizes finer than 240 are generally considered flour. Fine grain wheels are preferred for grinding hard materials, as they have more cutting edges and will cut faster than coarse grain wheels. Coarse grain wheels are generally preferred for rapid metal removal on softer materials.

Bond Grade (Hardness)

Station three of the wheel marking is the grade or hardness of the wheel. As shown in figure 6-4, the grade is designated by a letter of the alphabet; grades run from A to Z, or soft to hard.

The grade of a grinding wheel is a measure of the bond's ability to retain the abrasive grains in the wheel. Grinding wheels have a soft to a hard grade. This does not mean that the bond or the abrasive is soft or hard; it means that the wheel has a large amount of bond (hard grade) or a small amount of bond (soft grade). Figure 6-5 shows a magnified portion of a soft grade and a hard grade wheel. You can see by the illustration that a part of the bond surrounds the abrasive grains, and the remainder of the bond forms into posts which both hold the grains to the wheel and hold them apart from each other. The wheel with the larger amount of bonding material has thick bond posts and will offer great resistance to pressures generated in grinding. The wheel with the least amount of bond will offer less resistance to the grinding pressures. In other words, the wheel with a large amount of bond is



28.311
Figure 6-5.–How bond affects the grade of the wheel Wheel A, softer; wheel B, harder.

a hard grade and the wheel with a small amount of bond is a soft grade.

Structure

The fourth station of the grinding wheel marking is the structure. The structure is designated by numbers from 1 to 15, as illustrated in figure 6-4. The structure of a grinding wheel refers to the open space between the grains, as shown in figure 6-5. Grains that are very closely spaced are said to be dense; when grains are wider apart, they are said to be open. The metal removal will be greater for open-grain wheels than for close-grain wheels. Also dense, or close grain, wheels will normally produce a finer finish. The structure of a grinding wheel comprises about 20% of the grinding wheel.

Bond Type

The fifth station on the grinding wheel marking is the bond type. The bond comprises the remaining 40% of the grinding wheel and is one of the most important parts of the wheel. The bond determines the strength of the wheel. The six basic types of bonds are vitrified, silicate, rubber, resinoid, shellac, and oxychloride.

VITRIFIED BOND.–Designated by the letter V, this is the most common bond used today in grinding wheels. Approximately 75% of all grinding wheels are made with vitrified bond. This bond is not affected by oil, acid, or water. Vitrified bonded wheels are strong and porous, and rapid temperature changes have little or no effect on them. Vitrified bond is composed of special clays. When heated to approximately 2300°F the clays form a glass-like cement. Vitrified wheels should not be run faster than 6500 surface feet per minute.

SILICATE BOND.–Silicate bonded wheels are designated by the letter S. The bond is made of silicate of soda. Silicate bonded wheels are used mainly for large, slow rpm machines where a cooler cutting action is desired. Silicate bonded wheels are softer than vitrified wheels; they release the grains more readily than vitrified bonded wheels. Silicate bonded wheels are heated to approximately 500°F when they are made. This wheel, like the vitrified bonded wheel, is not to be run at a speed greater than 6500 feet per minute.

RUBBER BOND.–Rubber bonded wheels are designated by the letter R. The bond consists of rubber with sulphur added as a vulcanizing agent. The bond is made into a sheet into which the grains are rolled. The wheel is stamped out of this sheet and heated in a pressurized mold until the vulcanizing action is completed. Rubber bonded wheels are very strong and are elastic. They are used in making thin cutoff wheels and are used extensively for regulating wheels on centerless grinders. Rubber bonded wheels produce a high finish and can be run at speeds between 9,500 and 16,000 surface feet per minute.

RESINOID BOND.—Resinoid bonded wheels are designated by the letter B. Resinoid bond is made from powdered or liquid resin with a plasticizer added. The wheels are pressed and molded to size and fired at approximately 320°F. Resinoid wheels are shock resistant and very strong. They are used for rough grinding and as cutoff wheels. Resinoid wheels like rubber bonded wheels can be run between the speed of 9,500 to 16,000 surface feet per minute.

SHELLAC BOND.—Shellac bonded wheels are designated by the letter E. Wheels of this type are made from a secretion from Lac bugs. The abrasive and bond are mixed and molded to shape and are baked at approximately 300°F. Shellac bonded wheels give a high finish and have a cool cutting action when used as cutoff wheels. Shellac bonded wheels can be run between 9,500 and 12,500 surface feet per minute.

OXYCHLORIDE BOND.—Oxychloride bonded wheels are designated by the letter O. Oxychloride bond is made from chemicals and is a form of cold-setting cement. This bond is seldom, if ever, used in grinding wheels but is used extensively to hold abrasives on sanding disks. Oxychloride bonded wheels can be run at speeds between 5,000 and 6,500 surface feet per minute.

Manufacturer's Record Symbol

The sixth station on the grinding wheel marking is the manufacturer's record. This may be a letter or number, or both. It is used by the manufacturer to designate bond modifications or wheel characteristics.

DIAMOND WHEELS

Diamond grinding wheels are classed by themselves. Wheels of this type are very expensive and should be used with care and only for grinding carbide cutting tools. Diamond wheels can be made from natural or manufactured diamonds. They are marked similarly to aluminum-oxide and silicon-carbide wheels, although there is not a standard system. The first station is the type of abrasive, designated D for natural and SD for manufactured. The second station is the grit size which can range from 24 to 500. A 100-grain size might be used for rough work, and a 220 for finish work. In a Navy machine shop, you might find a 150-grain wheel and use it for both rough and finish grinding. The third station is the grade, designated by letters of the alphabet. The fourth station is concentration, designated by numbers. The concentration or proportion of diamonds to bond might be numbered 25, 50, 75 and 100 going from low to high. The fifth station is the bond type, designated B for resinoid, M for metal, and V for vitrified. The sixth station may or may not be used; when used it identifies bond modification. The seventh station is the depth of diamond section. This is the thickness of the abrasive layer and ranges from 1/32 to 1/4 inch. Cutting speeds from 4,500 to 6,000 surface feet per minute should be used.

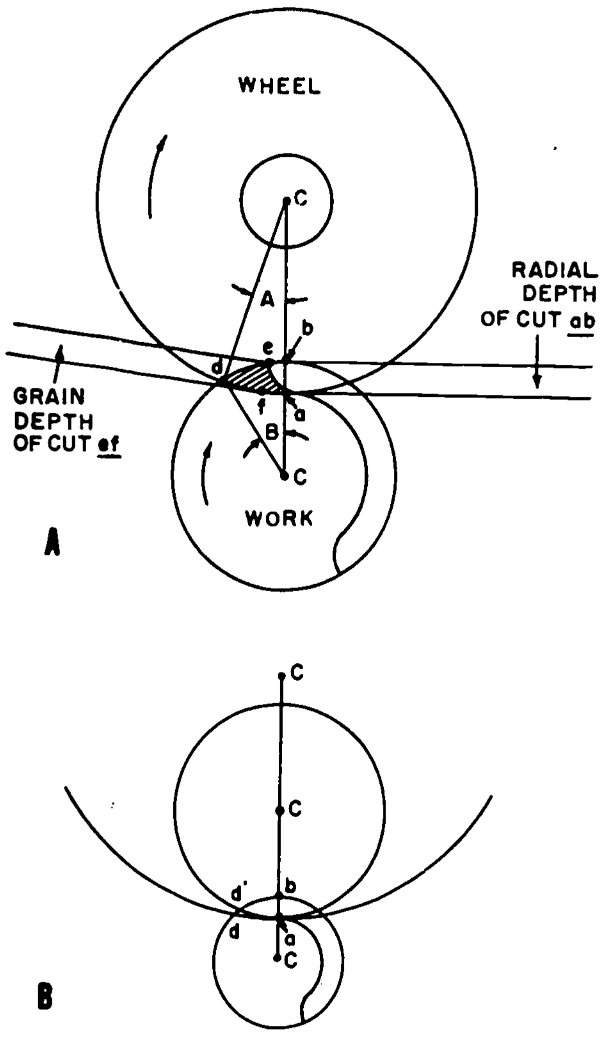
GRAIN DEPTH OF CUT

On most ships, stowage space is limited. Consequently, the inventory of grinding wheels must be kept to a minimum. It would be impracticable and unnecessary to keep on hand a wheel for each and every grinding job. With a knowledge of the theory of grain depth of cut, you can vary the cutting action of the various wheels and with a small inventory can perform practically any grinding operation that may be necessary.

For ease in understanding this theory, assume that a grinding wheel has a single grain. When the grain reaches the point of contact with the work, the depth of cut is zero. As the wheel and the work revolve, the depth of cut will increase to a maximum at some point along the arc of contact. Since the wheel rotates much faster than the work, the greatest depth of cut will be near the spot where the grain loses contact with the work. This greatest depth is called the grain depth of cut.

Part A of figure 6-6 illustrates a grinding wheel and a workpiece; ab is the radial depth of cut, ad is the arc of contact, and ef is the grain depth of cut. As the wheel rotates, the grain moves from the point of contact to point d in a given amount of time, and at the same time point d on the workpiece moves to point e. The workpiece generally rotates at a much slower speed than the wheel; consequently, length de will be less than ad. During a given amount of time the grain will remove a chip represented by ade.

Referring to part B of figure 6-6, assume that the wheel has worn down to a much smaller size. The speed of both the work and the wheel remains the same. The length of the arc of contact ad is greater than the arc of contact ad' so that the chips cut by both the original size wheel and the smaller size wheel are of the same volume, but since the arc of contact of the smaller wheel is less, the thickness of the chip is greater. From this theory you can see that the grain depth of cut increases as the diameter of the wheel decreases, and the wheel will act softer. This is true because the area of contact between the wheel and the work decreases, but the pressure exerted on the grains increases. Consequently, the grains are torn from the wheel sooner.



126.41X
Figure 6-6.—Grain depth of cut; center type machine.

Based on this theory, increasing the grain depth of cut applies more pressure per grain and causes the wheel to act softer; decreasing the grain depth of cut decreases the pressure per grain and causes the wheel to act harder. The following actions show the changes that may be made to make the wheel act a certain way.

INCREASE THE GRAIN DEPTH OF CUT (SOFTER WHEEL)

Increase the work speed
Decrease the wheel speed
Reduce the diameter of the wheel

DECREASE THE GRAIN DEPTH OF CUT (HARDER WHEEL)

Decrease the work speed
Increase the wheel speed
Increase the diameter of the wheel

GRINDING WHEEL SELECTION AND USE

The selection of grinding wheels for precision grinding can be discussed generally in terms of such factors as the physical properties of the material to be ground, the amount of stock to be removed (depth of cut), the wheel speed and work speed, and the finish required. Selection of a grinding wheel that has the proper abrasive, grain, grade, and bond is determined by one or more of these factors.

An aluminum oxide abrasive is most suitable for grinding carbon and alloy steel, high-speed

Table 6-1.—Recommendations for Selecting Grinding Wheels

OPERATION	WHEEL DESIGNATION						MATERIAL
	Abrasive	Grain size	Grade	Structure	Bond	Mfg. Symbol	
Cylindrical grinding	A	60	K	8	V	------	High-speed steel
	A	60	L	5	V	------	Hardened steel
	A	54	M	5	V	------	Soft steel
	A	36	G	12	V	------	Stainless steel
	C	36	K	5	V	------	Cast iron, brass, aluminum
	A	60	G	12	V	------	Nickel copper (Monel)
	A	54	L	5	V	------	General purpose
Surface grinding	A	46	H	8	V	------	High-speed steel
	A	60	F	12	V	------	Hardened steel
	A	46	J	5	V	------	Soft steel
	A	36	G	12	V	------	Stainless steel
	C	36	J	8	V	------	Cast iron and bronze
	A	60	G	12	V	------	Nickel copper (Monel)
	A	24	H	8	V	------	General purpose
Tool and cutter grinding	A	46	K	8	V	------	High-speed steel or cast alloy milling cutter
	A	54	L	5	V	------	Reamers
	A	60	K	8	V	------	Taps

28.253.0

steel, cast alloys and malleable iron. A silicon carbide abrasive is most suitable for grinding nonferrous metals, nonmetallic materials, and cemented carbides.

Generally, the softer and more ductile the material being ground, the coarser should be the grain selected. Also, if a large amount of material is to be removed, a coarse grain wheel is recommended (except on very hard materials). If a good finish is required, a fine grain wheel should be used. If the machine you are using is worn, a harder grade may be necessary to help offset the effects of wear on the machine. Using a coolant also permits the use of a harder grade of wheel. Table 6-1 lists recommended grinding wheels for various operations.

Figure 6-7 shows the type of grinding wheel used on bench and pedestal grinders. When you replace the wheel be sure that the physical dimensions of the new wheel are correct for the grinder on which it will be used. The outisde diameter, the thickness, and the spindle hole size are the three dimensions that must be checked. An adapter (bushing) may be used to decrease the size of the spindle hole, so that it fits your grinder.

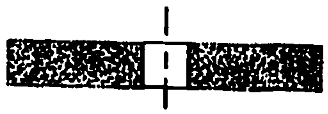
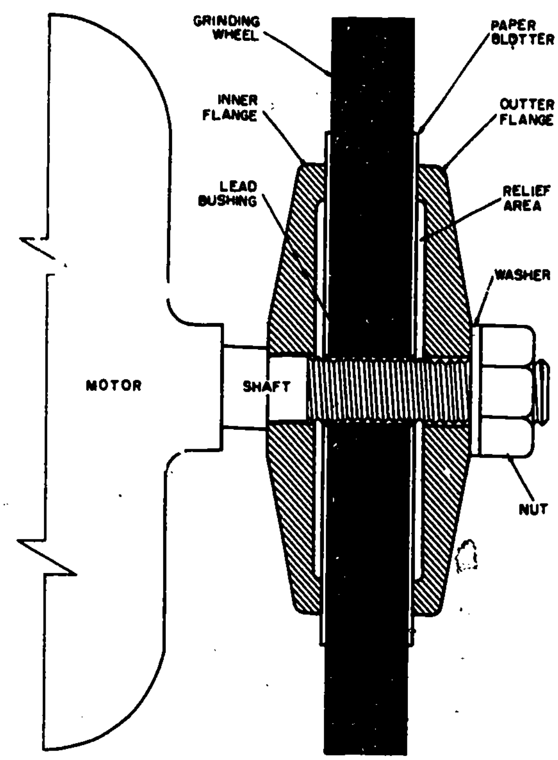


Figure 6-7.–Grinding wheel for bench and pedestal grinders.

The wheels recommended for grinding and sharpening single point (lather, planer, shaper, etc) tool bits made from high-carbon steel or high-speed steel are A3605V (coarse wheel) and A60M5V (fine or finish wheel). Stellite tools should be ground on a wheel designated A46N5V. These grinding wheels, which have aluminum oxide as an abrasive material, should be used to grind steel and steel alloys only. Grinding cast iron, nonferrous metal or nonmetallic materials with these grinding wheels will result in loading or pinning of the wheel as the particles of the material being ground become imbedded in the pores of the grinding wheel. This condition puts a strain on the wheel and could cause it to fail with possible injury to someone nearby.



28.62(28D)
Figure 6-8.–Method of mounting a grinding wheel.

WHEEL INSTALLATION

The wheel of a bench or pedestal grinder must be properly installed; otherwise, the wheel will not operate properly and accidents may occur. Before a wheel is installed, it should be inspected for visible defects and "sounded" to determine whether it has invisible cracks. To properly sound a wheel, hold it up by placing a hammer handle or a short piece of cord through the spindle hole. Using a nonmetallic object such as a screwdriver handle or small wooden mallet, tap the wheel lightly on the side. Rotate the wheel 1/4 of a turn (90°) and repeat. A good wheel gives out a clear ringing sound when tapped, but if the wheel is cracked, a dull thud is produced.

You will find it easier to understand the following information on mounting the wheel if you refer to figure 6-8. Ensure that the shaft and flanges are clean and free of grit and old blotter material. Place the inner flange in place and follow it with a blotter. NOTE: The blotter thickness for paper must be no thicker than 0.025 inch and no thicker than 0.125 inch for leather or rubber. The blotter is used to ensure even pressure on the wheel and to dampen the vibration between the wheel and shaft when the grinder is in operation.

Next, mount the wheel, and ensure that it fits on the shaft without play. There should be a 0.002- to 0.005-inch clearance. You may need to scrape or ream the lead bushing in the center of the wheel to obtain this clearance. NEVER FORCE THE WHEEL ON THE SHAFT. Forcing the wheel on the shaft may cause the wheel to crack when placed in operation, or cause the wheel to be slightly out of axial alignment.

The next item is another blotter and then the outer flange. NOTE: the flanges are recessed so they provide an even pressure on the wheel.

The flanges should be at least one-third the diameter of the wheel.

Next, install the washer and secure the nut. Tighten the securing nut sufficiently to hold the wheel firmly; tightening too much may damage the wheel.

TRUING AND DRESSING THE WHEEL

Grinding wheels, like other cutting tools require frequent reconditioning of cutting surfaces to perform efficiently. Dressing is the process of cleaning the cutting face of grinding wheels. This cleaning breaks away dull abrasive grains and smooths the surface so that there are no grooves. Truing is the removal of material from the cutting face of the wheel so that the resulting surface runs absolutely true to some other surface such as the grinding wheel shaft.

The wheel dresser shown in figure 6-9 is used for dressing grinding wheels on bench and pedestal grinders. To dress a wheel with this tool, start the grinder and let it come up to speed. Set the wheel dresser on the rest as shown in figure 6-9 and bring it in firm contact with the wheel. Move the wheel dresser across the periphery of the wheel until the surface is clean and approximately square with the sides of the wheel.

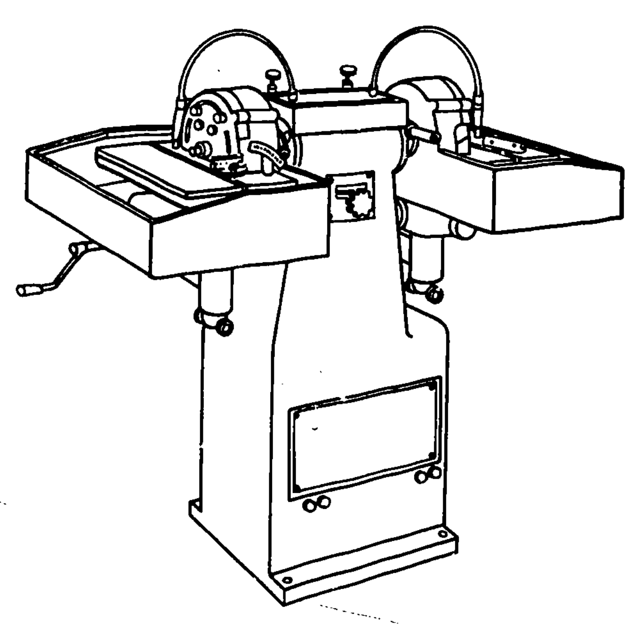
If grinding wheels get out of balance because of out-of-roundness, dressing the wheel will usually remedy the condition. A grinding wheel can get out of balance if part of the wheel is immersed in coolant. If this happens, remove the wheel and dry it out by baking. If the wheel gets out of balance axially, it probably will not affect the efficiency of the wheel on bench and pedestal grinders. This unbalance may be remedied simply by removing the wheel and cleaning the shaft spindle and spindle hole in the wheel and the flanges.

28.63

Figure 6-9.—Using a grinding wheel dresser.

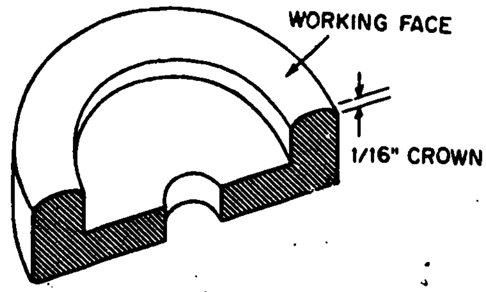
CARBIDE TOOL GRINDER

The carbide tool grinder (fig. 6-10) looks much like a pedestal grinder with the toolrest on the s... instead of on the front. The main components of the carbide tool grinder are: a motor with the shaft extended at each end for mounting the grinding wheels; the pedestal which supports the motor and is fastened to the deck; wheel guards which are mounted around the circumference and back of the grinding wheels as a safety device; and an adjustable toolrest mounted in front of each wheel for supporting the tool bits while they are being ground.



28.329

Figure 6-10.—Carbide tool grinder.



28.284
Figure 6-11.—Crown on the working face of a wheel for a carbide tool bit grinder.

Unlike the pedestal grinder where the grinding is done on the periphery of the wheel, the grinding wheels used on carbide tool bit grinders have the cutting face on the side of the wheel. The straight cup wheel (fig. 6-11) is similar to the wheels used on most carbide tool bit grinders. Some carbide tool grinders have a straight cup wheel on one side of the grinder and a straight wheel, such as the type used on a pedestal or bench grinder, on the other side.

The adjustable toolrest has an accurately ground groove or keyway across the top of its table. This groove is for holding a protractor attachment which can be set to the desired cutting edge angle. The toolrest will also adjust to permit grinding the relief angle.

Some carbide tool grinders have a coolant system. When coolant is available, the tool should have an ample, steady stream of coolant directed at the point of grinding wheel contact. An irregular flow of coolant may allow the tool to heat up and then be quenched quickly, resulting in cracks to the carbide. If no coolant system is available, do NOT dip the carbide into a container of water when it becomes hot. Allow it to air cool.

Carbide tipped tool bits may be (1) the disposable tip type where the tip has three or more pre-ground cutting edges or (2) the brazed tip type where the cutting edges must be ground into the tip. The disposable tip type tool bit needs no sharpening; the tips are disposed of as their cutting edges become dull. The brazed tip type tool bit is sharpened on the carbide tool bit grinder.

For best results in sharpening carbide tipped tool bits, use a silicon carbide wheel for roughing and a diamond impregnated wheel for finishing.

WHEEL SELECTION

The best results are obtained from a carbide tipped tool by using four different grinding wheels to sharpen them. The aluminum oxide wheel recommended for grinding high-speed steel tools should be used to grind the steel shank beneath the carbide tip to the desired end and side cutting edge angles with a relief angle of approximately 15°. This angle is approximately double the clearance angle ground on the carbide tip. The wheels for the carbide tool grinder should have silicon carbide as the abrasive material. The wheel used to rough grind should be designated C6018V. The wheel for semifinish grinding should be C100H8V. A diamond impregnated grinding wheel with the designation SD 220-P50V is recommended for the finish grinding of carbide tipped tools.

OPERATION OF THE CARBIDE TOOL GRINDER

The sharpening of a carbide tipped tool bit is accomplished in the following manner.

- Using a grinder with an ALUMINUM OXIDE wheel, grind side relief and end relief angles on the STEEL shanks. Caution: NEVER grind steel shanks with silicon carbide wheels.

- Dress the silicon carbide wheel with a star type wheel dresser. Form a 1/16-inch crown on the working face of the wheel to minimize the amount of contact between the tip and the wheel (fig. 6-11).

- Using the graduated dial on the side of the toolrest, adjust the toolrest to the desired side clearance angle.

- Place the protractor on the toolrest with the protractor key in the keyway. Set the protractor to the proper side cutting edge angle.

- Hold the shank of the tool bit firmly against the side of the protractor; move the tool bit back and forth across the wheel, keeping a steady, even pressure against the wheel. To prevent burning the carbide tip, keep the tool bit continually in motion while grinding it.

Carbide tipped cutters may be rough ground and finish ground with silicon carbide wheels. Generally, when a carbide tool chip grinder is available, the finish grinding operation is performed on this machine with a diamond wheel. The chip grinder is very similar to the carbide tool bit grinder except that the wheels are smaller and diamond impregnated.

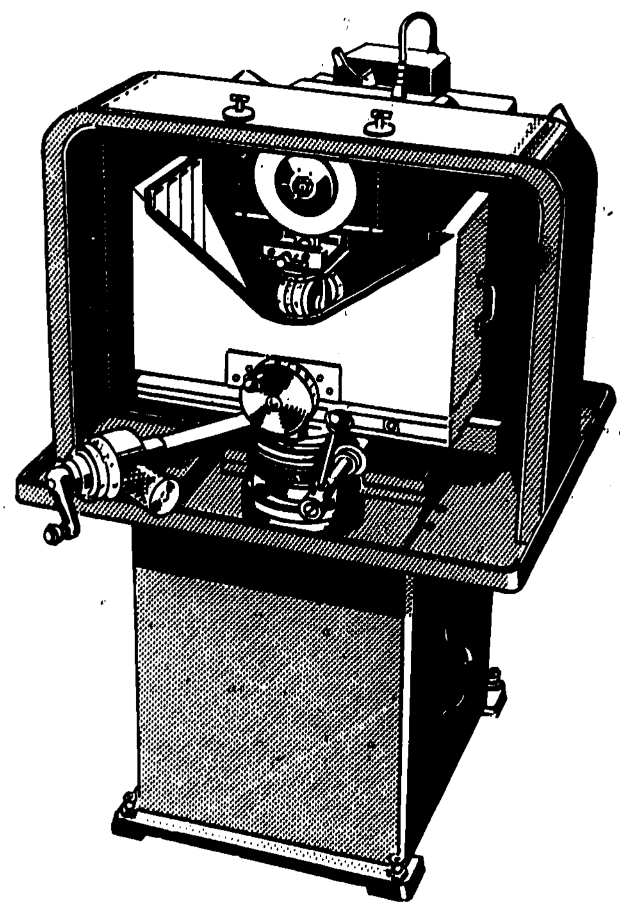
Dry grinding is generally done when silicon carbide wheels are used. When using diamond wheels, be sure to use coolant on the tool and wheel face. NEVER allow the steel shank to come into contact with a diamond wheel as this will immediately load the wheel.

CHIP BREAKER GRINDER

A chip breaker grinder (fig. 6-12) is a specialized grinding machine. It is designed to permit accurate grinding of grooves or indentations on the top surface of carbide tools, so that the direction and length of the chips produced in cutting metal can be controlled. A description of the various types of chip breakers that are commonly ground on carbide tools will be presented later in this chapter.

The chip breaker grinder has a vise, which can be adjusted to four different angles, to hold the tool to be ground. These angles—the side cutting edge, back rake, side rake, and the chip breaker—are explained later in this chapter. The vise is mounted so it can be moved back and forth under the grinding wheel. Both the cross feed, for positioning the tool under the grinding wheel, and the vertical feed, for controlling the depth of the chip breaker, are graduated in increments of 0.001 inch.

A diamond wheel is used on the chip breaker grinder. The wheel is usually a type 1 straight



28.330

Figure 6-12.—Chip breaker grinder.

wheel but differs from other type 1 wheels that you will use in that it is normally less than 1/4 inch thick. An SD 150 R 100B grinding wheel is normally recommended.

Chip breaker grinders have a coolant system that either floods or slowly drips coolant on the tool being ground. The main objective in using coolant is to prevent the grinding wheel from loading up or glazing over from the grinding operation.

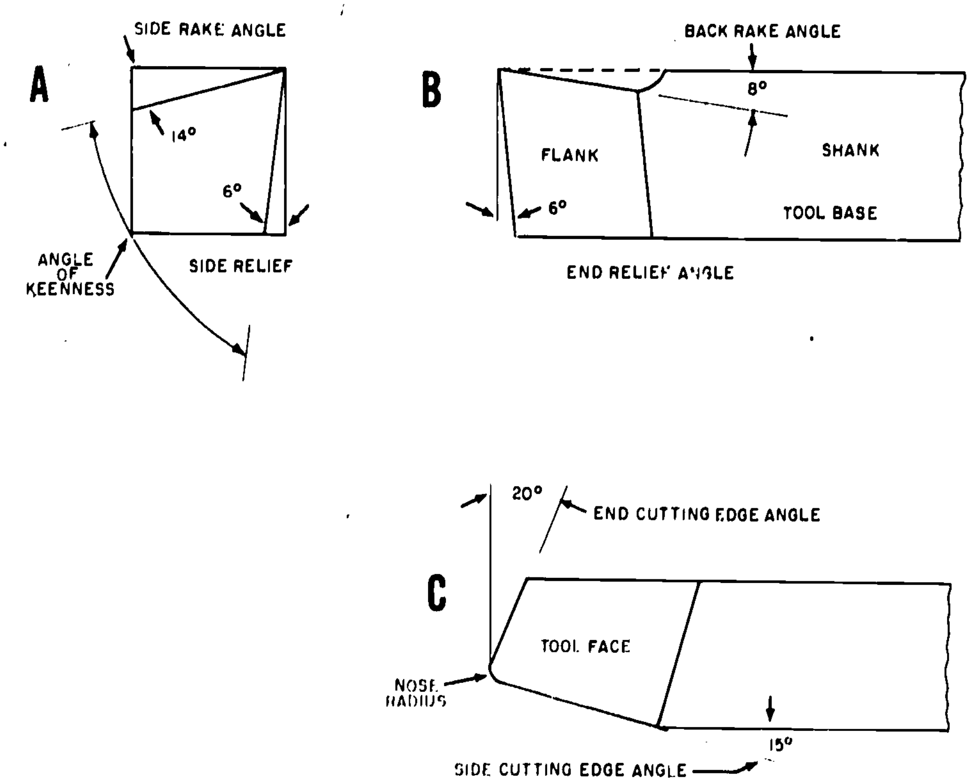
SINGLE-POINT CUTTING TOOLS

A single-point or single-edged cutting tool is a tool which has only one cutting edge as opposed to two or more cutting edges. Drill bits are multiple-edged cutters, most lathe tools are single edged. To properly grind a single-point cutting tool, you must know the relief angles, the rake angles, and the cutting edge angles that are required for specific machines and materials. You must know also what materials are generally used for cutting tools and how tools for various machines differ.

Cutting Tool Terminology

Figure 6-13 shows the application of the angles and surfaces we use in discussing single-point cutting tools. Notice that there are two relief angles and two rake angles and that the angle of keenness is formed by a rake and a relief angle.

SIDE RAKE.–Side rake (fig. 6-13A) is the angle at which the top surface of the tool is ground away with respect to the bottom surface of the tool bit and either away from or toward the side cutting edge. The amount of side rake influences to some extent the size of the angle of keenness. It causes the chip to "flow" to the side of the tool away from the side cutting edge. The side rake is positive if the angle slopes downward from the side cutting edge toward the opposite side of the tool, and negative if it slopes upward. A positive side rake is most often used on ground single-point tools. Generally, the side rake angle will be steeper (in the positive direction) for cutting the safety metals and will decrease as the hardness of the metal increases. A steep side rake angle in the positive direction causes the chip produced in cutting to be long and stringy. Decreasing the angle will cause the chip to curl up and break more quickly. A negative side rake is recommended when the tool will be subjected to shock, such as an interrupted cut or when the metal being cut is extremely hard.



28.64

Figure 6-13.–Applications of tool terminology.

166

BACK RAKE.—The back rake (fig. 6-13B) is the angle at which the top surface to the tool is ground away with respect to the bottom surface of the tool and perpendicular to the side rake angle. It is ground primarily to cause the chip cut by the tool to "flow" back toward the shank of the tool. Back rake may be positive or negative; it is positive if it slopes downward from the nose of the tool toward the shank, or negative if a reverse angle is ground. The rake angles aid in forming the angle of keenness and in directing the chip flow away from the point of cutting. The same general recommendations concerning positive or negative side rake angles apply to the back rake angle.

SIDE RELIEF.—The side relief (fig. 6-13A) is the angle at which the side or flank of the tool is ground so that the side cutting edge leads the flank surface when cutting. The side relief angle, like the side rake angle, influences the angle of keenness. The total of the side rake and side relief subtracted from 90° equals the angle of keenness. A tool with proper side relief causes the side thrust to be concentrated on the cutting edge rather than rubbing on the flank of the tool.

END RELIEF.—The end relief (fig. 6-13B) is the angle at which the end surface of the tool is ground so that the front face edge of the tool leads the front surface.

ANGLE OF KEENNESS.—The angle of keenness or wedge angle (fig. 6-13A) is formed by the side rake and the side relief ground in a tool. Generally, for cutting soft materials this angle is smaller than for cutting hard materials.

SIDE CUTTING EDGE.—The side cutting edge angle (fig. 6-13C) is ground on the side of the tool that is fed into the work. This angle can be anywhere from 0° for cutting to a shoulder, up to 30° for straight turning. An angle of 15° is recommended for most rough turning operations. In turning long slender shifts, a side cutting edge angle that is too large can cause chatter. Since the pressure on the cutting edge and the heat generated by the cutting action decrease as the side cutting edge angle increases, the angle should be as large as the machining operation will allow.

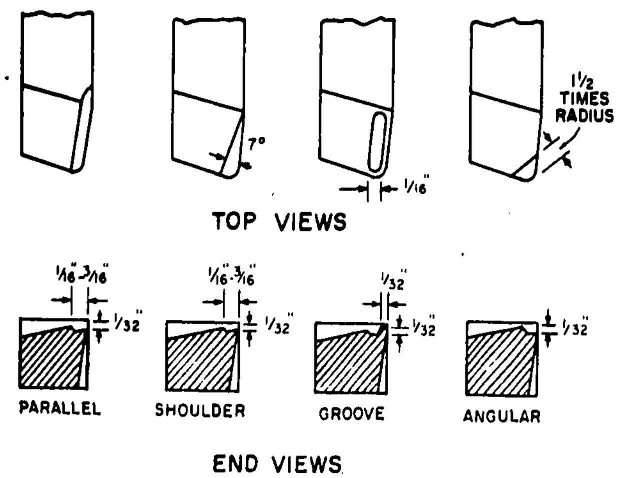
END CUTTING EDGE.—The end cutting edge angle (fig. 6-13C) is ground on the end of the tool to permit the nose to make contact with the work without the tool dragging the surface. An angle of from 8° to 30° is commonly used with approximately 15° recommended for rough turning operations. Finish operations can be made with the end cutting edge angle slightly larger. Too large an end cutting edge angle will reduce the support given the nose of the tool and could cause premature failure of the cutting edge.

NOSE.—The nose (fig. 6-13C) strengthens the tip of the tool, helps to carry away the heat generated by the cutting action and helps to obtain a good finish. A radius (rounded end) of from 1/64 to 1/32 inch is normally used for turning operations. A tool that is used with the nose ground to a straight point will fail much more rapidly than one which has had a slight radius ground or honed on it. However, too large a radius will cause chatter because of excessive tool contact with the work

GROUND-IN CHIP BREAKERS

Chip breakers are indentations ground on the top surface of the tool that help reduce or prevent the formation of long and dangerous chips. The chip breaker will cause the chips to curl up and break into short, safe, manageable chips. Chip breakers are ground mostly on roughing tools, but they can be ground on finishing tools used to machine soft ductile metals. Figure 6-14 shows four of the several types of chip breakers that can be ground onto the cutting tool.

The dimensions given are general and can be modified to compensate for the various feed rates, depths of cut, and types of material being machined. The groove type chip breaker must be carefully ground to prevent it from coming too close to the cutting edge which reduces the life of the tool due to decreased support of the cutting edge. Chip breakers on carbide tipped



28.333

Figure 6-14.—Chip breakers.

tools can be ground with the diamond wheel on the chip breaker grinder. High-speed steel tools must be ground with an aluminum oxide grinding wheel. This can be done on a bench grinder by dressing the wheel until it has a sharp edge or by using a universal vise which can be set to compound angles on a surface or tool end cutter grinder.

CUTTING TOOL MATERIALS

The materials used to make machine cutting tools must have the hardness necessary to cut other metals, be wear resistant, have impact strength to resist fracture, and be able to retain their hardness and cutting edge at high temperatures. There are several different materials which are used for cutting tools and each one has properties different from the others. Selection of a specific cutting tool material depends on the metal being cut and conditions under which the cutting is being done.

CARBON TOOL STEEL

The carbon steel used to make cutting tools usually contains from 0.90% to 1.40% carbon. Some types contain small amounts of chrome or vanadium to increase the degree of hardness or toughness. Carbon steel is limited in its use as a cutting tool material because of its low tolerance to the high temperatures generated during the cutting process. Jobs made from carbon steel will begin to lose their hardness, 50 to 64 Rockwell "C," at a tempering range of approximately 350° to 650°F. Carbon steel tools perform best as lathe cutting tools when used to take light or finishing cuts on relatively soft materials such as brass, aluminum, and unhardened low carbon steels. The cutting speed for carbon steel tools should be approximately 50% of the speeds recommended for high-speed steel tools.

HIGH-SPEED STEEL

High-speed steel is probably the most common cutting tool material used in Navy machine shops. Unlike carbon steel tools, high-speed steel tools are capable of maintaining their hardness and abrasion resistance under the high temperatures and pressures generated during the general cutting process. Although the hardness of the high-speed tool (60 to 70 Rockwell "C") is not much greater than the carbon steel tools, the tempering temperature at which high-speed steel begins to lose its hardness is 1000° to 1100°F. There are two types of high speed steel tools which are generally used in machine shops. They are tungsten high-speed steel and molybdenum high-speed steel. These designations are used to indicate the major alloying element in each of the two types. Both types have similar characteristics in their ability to resist abrasive wear, remain hard at high temperatures, and in their degree of hardness. The molybdenum type high-speed steel is tougher than the tungsten type and is more effective in machinery operations where interrupted cuts are present.

During interrupted cuts, such as cutting out-of-round or slotted material, the cutter contacts the material many times in a short period of time. This "hammering" effect dulls or breaks cutters which are not tough enough to withstand the shock effect.

CAST ALLOYS

Cast alloy tool steel usually contains varying amounts of cobalt, chrome, tungsten, and molybdenum. Tools made from these steels are generally more efficient than high-speed steel, providing a tool that retains its hardness up to an operating temperature of approximately 1400°F. This characteristic allows cutting speeds approximately 60% greater than high-speed steel. However, cast alloy tools are not as tough as the high-speed steel tools and therefore cannot be subjected to the same cutting stresses, such as interrupted cuts. Clearances that are ground on cast alloy cutting tools are less than those ground on high-speed steel tools because of the lower degree of toughness. Tools made from this metal are generally known under trade names such as Stellite, Rexalloy, and Fantung.

CEMENTED CARBIDE

Cemented carbides, or sintered carbides as they are sometimes called, can be used at cutting speeds of two to four times those listed for high-speed steel. The softest carbide grade is equal in hardness to the hardest tool steel and is capable of maintaining its hardness and abrasive resistance up to approximately 1700°F. Carbide is much more brittle than any of the other cutting tool materials previously described in this chapter. Because of this, interrupted cuts should be avoided and the machine setup should be as rigid and vibration free as possible. There are many different grades of carbides, each grade being more suited for a particular machining operation and metal than the others. Carbide manufacturers normally have available charts that match the correct grade for any given cutting application. Due to the brittleness of carbide, it is seldom used in a solid form as a cutting tool. The most common usage is as a tip on a steel shank or on the cutting edge of a twist drill. Carbide tipped lathe cutting tools are usually in the form of carbide tips brazed onto the end of a steel shank or as small variously shaped inserts, mechanically held on the end of a steel shank. A brief description of these two types of cutters is included in the following paragraphs.

BRAZED ON TIP

The brazed on carbide tip cutting tool was the first carbide cutting tool developed and made available to the metal cutting industry. The insert type of carbide tip has become more widely used because of the ease in changing cutting edges. There are some jobs which have shapes or radii that cannot be readily formed with a standard shaped insert, while a brazed on tip is easily ground to machine them. The various styles of tools required in machinery, such as turning, facing, threading, and grooving in are available with different grades of carbide tips already brazed onto steel shanks. Small carbide blanks are also available that can be brazed onto a shank by the machinist.

Brazing on a carbide tip is a relatively simple operation that can be performed by anyone qualified to operate an oxyacetylene torch. The first step is to thoroughly clean the steel shank by grinding or sandblasting and degreasing it with an approved solvent. The steel shanks and the carbide tip should be completely coated with a flux to further remove any contamination and to prevent oxidation during brazing. A thin shim-like brazing alloy is available that can be cut to the size needed and placed between the shank and the carbide tip. This type of bronze alloy is better than the rod type because it results in a more uniform and stronger bronze. Heating of the tool should be started from the bottom of the shank. Raise the temperature slowly until the bronze alloy melts. Tap the carbide tip gently to ensure a firm seat onto the shank and then let the tool cool in the air. Quenching the tool in water will either cause the carbide tip to crack or prevent the bronze band from holding the tip in place. After the tool is cooled, it can be ground to the shape desired.

Chip control, when cutting tools with brazed-on carbide tips are used, may be provided by either feeds and speeds or by chip breaker grooves ground into the top of the carbide tip. Figure 6-14 shows examples of the various types of chip breakers that are used. Using a chip breaker grinder with a diamond impregnated wheel is the best way to grind a chip breaker. However, it is possible to use a carbide tool grinder or a pedestal grinder wheel dressed so that it has a sharp edge. The depth of the chip breakers averages about 1/32 inch, while the width varies with the feed rate, depth c cut and material being cut. Grind the chip breaker narrow at first and widen it if the chip does not curl and break quickly enough. These same types of chip breakers also may be used on high-speed steel cutters.

CARBIDE CUTTING TOOLS—INSERT TYPE

Mechanically held carbide inserts are available in several different shapes—round, square, triangular, diamond threading, and grooving—and in different thicknesses, sizes, and nose radii. The inserts may have either a positive, a neutral, or a negative rake attitude to the part being cut. The rake attitude is a combination of the back rake oı the toolholder, the amount of clearance ground along the edge of the insert beneath the cutting edge, and the ground-in chip breaker.

An insert and its toolholder must have the same direction of rake. For instance, a negative rake toolholder requires a negative rake insert. Whenever possible, select the negative rake setup because both sides of the insert can be used, thus doubling the number of cutting edges available on positive or neutral inserts. Be sure to place a specially made shim, having the same shape as the insert, into the toolholder pocket beneath the insert to provide a smooth and firm support for the insert. Metho. s of holding the insert in the toolholder vary from one manufacturer to another. Some inserts are held in place by the cam-lock action of a screw positioned through a hole in their centers, while others are held against the toolholder by a clamp.

Chip control for carbide insert tooling is provided by two different methods. Some inserts have a groove ground into their cutting surfaces. Other inserts have a chip breaker plate held by a clamp on top of their cutting surfaces.

Other than diamond tools, ceramic cutting tools are the hardest and most heat resistant cutting tools available to the machinist. A ceramic cutting tool is capable of machining metals that are too hard for carbide tools to cut. Additionally, ceramic can sustain cutting temperatures of up to 2000°F. Therefore, ceramic tools can be operated at cutting speeds two to four times greater than cemented carbide tools.

Ceramic cutting tools are available as either solid ceramic or as ceramic coated carbide in several of the insert shapes available in cemented carbides and are secured in the toolholder by a clamp.

Whenever you handle ceramic cutting tools, be very careful because they are very brittle and will not tolerate shock or vibration. Be sure your lathe setup is very rigid and do not try to take any interrupted cuts. Also ensure that the lathe feed rate does not exceed 0.015 to 0.020 inch per revolution, as any rate exceeding this will subject the insert to excessive forces and may result in fracturing the insert.

Engine Lathe Tools

Figure 6-15 shows the most popular shapes of ground lathe tool cutter bits and their applications. In the following paragraphs each of the types shown is described.

LEFT-HAND TURNING TOOL.—This tool is ground for machining work when fed from left to right, as indicated in figure 6-15A. The cutting edge is on the right side of the tool and the top of the tool slopes down away from the cutting edge.

ROUND-NOSE TURNING TOOL.—This tool is for general all-round machine work and is used for taking light roughing cuts and finishing cuts. Usually, the top of the cutter bit is ground with side rake so that the tool may be fed from right to left. Sometimes this cutter bit is ground flat on top so that the tool may be fed in either direction (fig. 6-15B).

RIGHT-HAND TURNING TOOL.—This is just the opposite of the left-hand turning tool and is designed to cut when fed from right to left (fig. 6-15C). The cutting edge is on the left side. This is an ideal tool for taking roughing cuts and for general all-round machine work.

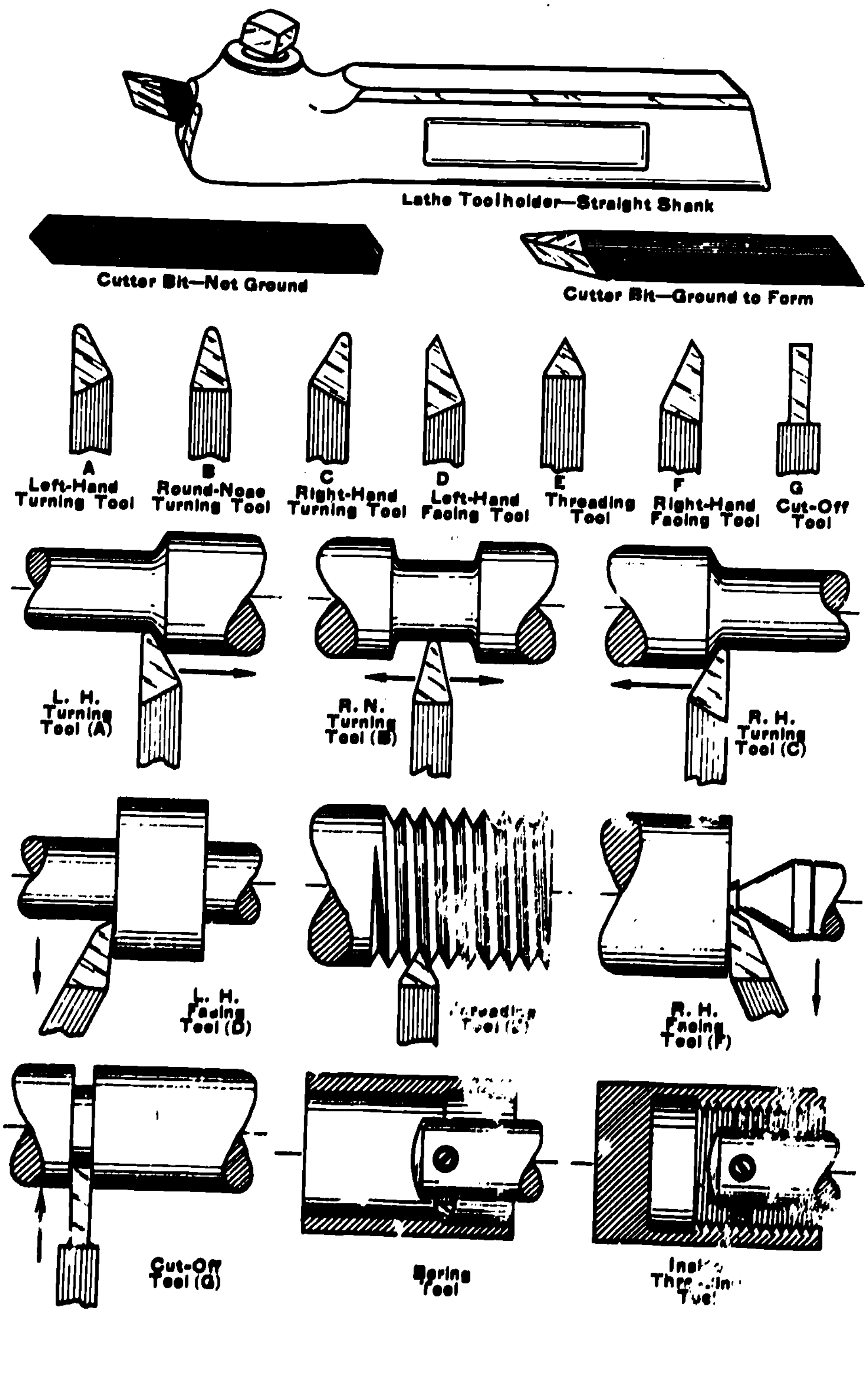


Figure 6-15.—Lathe tools and their application. 28.66

LEFT-HAND FACING TOOL.–This tool is intended for facing on the left-hand side of the work, as shown in figure 6-15D. The direction of feed is away from the lathe center. The cutting edge is on the right-hand side of the tool and the point of the tool is sharp to permit machining a square corner.

THREADING TOOL.–The point of the threading tool is ground to a 60° included angle for machining V-form screw threads (fig. 6-15E). Usually, the top of the tool is ground flat and there is clearance on both sides of the tool so that it will cut on both sides.

RIGHT-HAND FACING TOOL.–This tool is just the opposite of the left-hand facing tool and is intended for facing the right end of the work and for machining the right side of a shoulder. (See fig. 6-15F.)

SQUARE-NOSED PARTING (CUT-OFF) TOOL.–The principal cutting edge of this tool is on the front. (See fig. 6-15G.) Both sides of the tool must have sufficient clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge. This tool is convenient for machining necks, grooves, squaring corners, and for cutting off.

BORING TOOL.–The boring tool is usually ground the same shape as the left-hand turning tool so that the cutting edge is on the front side of the cutter bit and may be fed in toward the headstock.

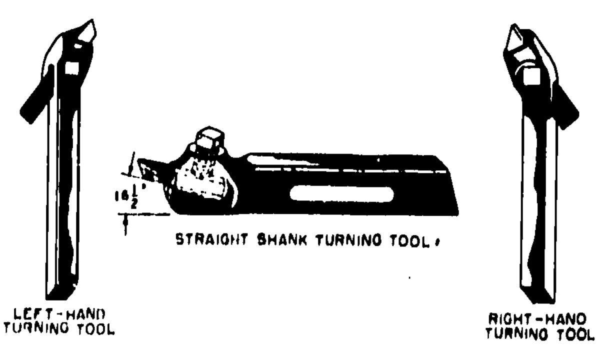
INTERNAL-THREADING TOOL.–The internal-threading (INSIDE-THREADING) tool is the same as the threading tool in figure 6-15E, except that it is usually much smaller. Boring and internal-threading tools may require larger relief angles when used in small diameter holes.

GRINDING ENGINE LATHE CUTTING TOOLS

The materials being machined and the machining techniques used limit the angles of a tool bit. When grinding the angles, however, you must also consider the type of toolholder and the position of the tool with respect to the axis of the workpiece. The angular offset and the angular vertical rise of the tool seat in a standard lathe toolholder affects the cutting edge angle and the end clearance angle of a tool when it is set up for machining. The position of the point of the tool bit with respect to the axis of the workpiece, whether higher, lower, or on center, changes the amount of front clearance.

Figure 6-16 shows some of the standard toolholders used in lathe work. Notice the angles at which the tool bits set in the various holders. These angles must be considered with respect to the angles ground in the tools and the angle that the toolholder is set with respect to the axis of the work. Also notice that a right-hand toolholder is offset to the LEFT and a left-hand toolholder is offset to the RIGHT. For most machining operations, a right-hand toolholder utilizes a left-hand turning tool and a left-hand toolholder a right-hand turning tool. Study figures 6-15 and 6-16 carefully to clearly understand this apparent contradiction. (Carbide tipped cutting tools should be held directly in the toolpost or in heavy duty holders similar to those used on turret lathes.)

The contour of a cutting tool is formed by the side cutting edge angle and the end cutting edge angle of the tool. (Parts A through G of fig. 6-15 illustrate the recommended contour of several types of tools.) There are no definite guidelines on either the form or the included angle of the contour of pointed tool bits. Each



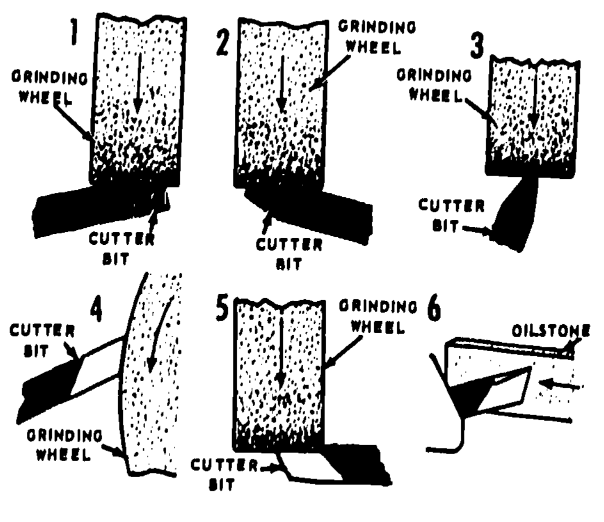
28.67(28D)
Figure 6-16.–Standard lathe toolholders.

machinist usually forms the contour as he prefers. For roughing cuts, it is recommended that the included angle of the contour of pointed bits be made as large as possible and still provide clearance on the trailing side or end edge. Tools for threading, facing between centers, and parting have specific shapes because of the form of the machined cut or the setup used.

STEPS IN GRINDING A TOOL BIT

The basic steps are similar for grinding a single-edged tool bit for any machine. The difference lies in shapes and angles. Use a coolant when grinding tool bits. Finish the cutting edge by honing on an oilstone. The basic steps for grinding a round nose turning tool are illustrated in figure 6-17. A description of each step follows:

1. Grind the left side of the tool, holding it at the correct angle against the wheel to form the necessary side clearance. Use the coarse grinding wheel to remove most of the metal, and then finish on the fine grinding wheel. (If ground on the periphery of a wheel less than 6 inches in diameter, the cutting edge will be undercut and will not have the correct angle.) Keep the tool cool while grinding.



28.65
Figure 6-17.—Grinding and honing a lathe cutter bit.

2. Grind the right side of the tool, holding it at the required angle to form the right side.

3. Grind the radius on the end of the tool. A small radius (approximately 1/32 inch) is preferable, as a large radius may cause chatter. Hold the tool lightly against the wheel and turn from side to side to produce the desired radius.

4. Grind the front of the tool to the desired front clearance angle.

5. Grind the top of the tool, holding it at the required angle to obtain the necessary side rake and back rake. Try not to remove too much of the metal in grinding, as the more metal left on the tool, the better it absorbs the heat produced during cutting.

6. Hone the cutting edge all around and on top with an oilstone until you have a keen cutting edge. Use a few drops of oil on the oilstone when honing. Honing will not only improve the cutting quality of the tool, but it will also produce a better finish on the work, and the cutting edge of the tool will stand up much longer than if it is not honed. The cutting edge should be sharp in order to shear off the metal instead of tearing it off.

GRINDING TOOLS FOR ROUGHING CUTS

A single edged cutting tool used for roughing cuts (relatively heavy depth of cut and heavy feed) can be modified slightly and used for finishing operations. In finishing, lighter feed and less depth of cut are normally used to get a smooth surface. To grind a finishing tool from a roughing tool, it is usually necessary only to increase the back rake angle, decrease the side rake and side clearance angles, and grind a radius on the point or nose of the tool. The only portion of a tool ground in this manner that will be cutting is the nose or point. Grinding a larger back rake angle makes a more acute, chisel-type nose or point. Decreasing side rake and side clearance provides more support for the cutting edge. By increasing the radius of the nose of the tool, more of the cutting edge is in contact with the work during the cut; and thus, by decreasing the feed rate of the tool, you will have a finer cut (similar to a scraping) which ensures a good finish.

In general machining work, you will find that it is easy to grind a tool which can be used for both roughing and finishing. To do this you grind a roughing tool to increase the nose or point radius a little more than usual. When you take the finish cut, decrease the feed rate until you obtain the required finish.

Table 6-2 gives recommended angles of tools for roughing and finishing cuts of various materials. The values provided in table 6-2 are somewhat arbitrarily selected as the most appropriate so that you can grind a minimum number of tools for maximum use, with respect to materials commonly machined in the shop. The angles given in table 6-2 and other tables in this chapter are intended as guidelines for the beginner. As you gain experience, you will find that you can grind tools that cut efficiently even though the angles do not conform exactly to the angles prescribed.

In table 6-2 you will note that the front clearance angles are practically standard for commonly used materials. The angle of side clearance within the tolerance given is based on the fact that small angles are necessary when a light feed rate is used and a larger angle is necessary when a higher feed rate is used. The front clearance angle should generally be increased in proportion to the increase in the diameter of the workpiece.

TURRET LATHE TOOLS

The angles of cutting tools for turret lathes are quite similar to those for engine lathe tool bits. However, the cutters themselves are usually much larger than those used on an engine lathe because the turret lathe is designed to remove large quantities of metal rapidly.

The relative merits, limitations, and applications, as well as the grinding of carbon tool steel, high-speed steel, Stellite, and carbide tool bits have been discussed in relation to

Table 6-2.—Angles for Grinding Engine Lathe Tools

Material	Operation	Angle (Degrees)			
		Back Rake	Side Rake	Side Relief	End Relief
Mild steel	Roughing Finishing	6-10 14-22	14-22 0	5-9 0	5-9 5-9
Hard steel and cast iron	Roughing Finishing	6-8 6-10	12-14 0	5-9 0	5-9 5-9
Brass and bronze	Roughing Finishing	6-8 14-22	4-10 0	5-9 0	5-9 5-9
Copper and aluminum	Roughing Finishing	8-10 8	16-24 16-24	5-9 0	5-9 5-9
Monel	Roughing Finishing	4-8 14-22	10-14 0	5-9 0	5-9 5-9

28.273

engine lathe tools. That information is applicable to turret lathe cutters, with a few exceptions which will be discussed here.

The turret lathe cutter must withstand heavy cutting pressures; therefore, its cutting edge must be well supported. The amount of support depends upon the amount of side clearance and side rake, and front clearance and back rake, given the tool. The clearance and rake angles prescribed in table 6-2 for tool bits are given in ranges, but a turret lathe cutter clearance and rake angles must be more specifically controlled. You must know the exact tool angles and grind the cutter to those angles. Table 6-3 lists the angles to which high-speed and carbon steel cutters should be ground for cutting particular kinds of metals. Additional information on turret lathe cutting tools will be given in chapter 10.

As carbide tips cannot tolerate bending but are otherwise capable of withstanding heavy cutting pressures, the tool angles prescribed for them are somewhat different. Table 6-4 lists the clearance and rake angles for carbide-tipped cutters. Notice that the side and front clearance angles differ only slightly from those prescribed for high-speed steel cutters but that the rake angles differ considerably. The reduction in back rake and side rake angles for carbide-tipped tools provides a bigger included angle for the cutting edge and, therefore, greater resistance against bending stress.

Before a carbide tip is ground, a clearance angle is ground on the shank with a conventional

Table 6-3.—Angles for Grinding Turret Lathe Tools (High Speed and Carbon Steel)

Material	Angle (Degrees)			
	Side Clearance	Front Clearance	Back Rake	Side Rake
Cast Iron	8	8	8	14
Copper	8	8	10	25
Brass, Soft	8	8	0	0
Hard Bronze	8	8	6	5
Aluminum	8	8	8	18
Steels:				
SAE X1112 Spec. Screw Stock	8	8	15	20
SAE X1315 Screw Stock	8	8	15	20
SAE 1020 Carbon Steel	8	8	15	15
SAE 1035 Carbon Steel	8	8	15	15
SAE 1045 Carbon Steel	8	8	10	12
SAE 1095 High Carbon Steel	8	8	5	10
SAE 2315 Nickel Alloy	8	8	15	15
SAE 2335 Nickel Alloy (Annealed)	8	8	15	15
SAE 2350 Nickel Steel (Annealed)	8	8	10	12
SAE 3115 Nickel-Chromium Alloy	8	8	15	15
SAE 3140 Nickel-Chromium (Annealed)	8	8	10	12
SAE 3250 Nickel-Chromium (Annealed)	8	8	8	12
SAE 4140 Chromium-Molybdenum	8	8	10	12
SAE 4615 Nickel-Molybdenum	8	8	15	15
SAE 6145 Chromium-Vanadium	8	8	8	12

28.274

Table 6-4.—Angles for Grinding Turret Lathe Tools (Carbide)

Material	Angle (Degrees)			
	Side Clearance	Front Clearance	Back Rake	Side Rake
Cast Iron	4-6	4-6	0-4	10-12
Aluminum	8-10	8-10	25	15
Copper	8-10	8-10	4	20
Brass	6	6	0	4
Bronze	6	6	0	4
Low carbon steel up to 0.20% carbon	8-10	8-10	4-6	10-12
Carbon steel up to 0.60% carbon	8-10	8-10	4-6	10-12
Tool steel over 0.60% carbon, and tough alloys	8-10	8-10	4-6	6-10

NOTE: Keep back rake angle as small as possible for greatest strength.

28.275

grinding wheel. This clearance angle must be slightly larger than the angle to be ground on the carbide tip. The clearance prevents loading the grinding wheel with the soft material of the shank when the clearance angles are ground on the tip.

Stellite cutters should be given tool angles that lie approximately midway between those prescribed for the high-speed steel and carbide-tipped types.

Another problem in connection with grinding turret lathe cutters is that of providing the cutter directional control for its chips, especially when the cutter is to machine a tough ductile metal from which the chip peels off in a continuous stream. A long, hot chip, in addition to being hazardous to you, will often interfere with the operation of the other cutters or with the operation of the lathe itself unless the direction of its run-off is controlled. As some other factors are involved, chip control will be discussed after the setting of cutters have been taken up in chapter 10.

SHAPER AND PLANER TOOLS

Shaper and planer cutting tools are similar in shape to lathe tools but differ mainly in the relief angles. As these cutting tools are held practically square with the work and do not feed during the cut, relief angles are much less than those required for turning operations. Nomenclature used for shaper and planer tools is the same as that for lathe tools; and the elements of the tool, such as relief and rake angles, are in the same relative position as shown in figure 6-13. Both carbon and high-speed steel are used for these tools.

Several types of tools are required for the various operations that may be performed on the shaper or planer. Although the types differ considerably as to shape, the same general rules govern the grinding of each type. Hand forging of shaper and planer tools is a thing of the past. Toolholders and interchangeable tool bits have replaced forged tools; this practice greatly reduces the amount of tool steel required for each tool.

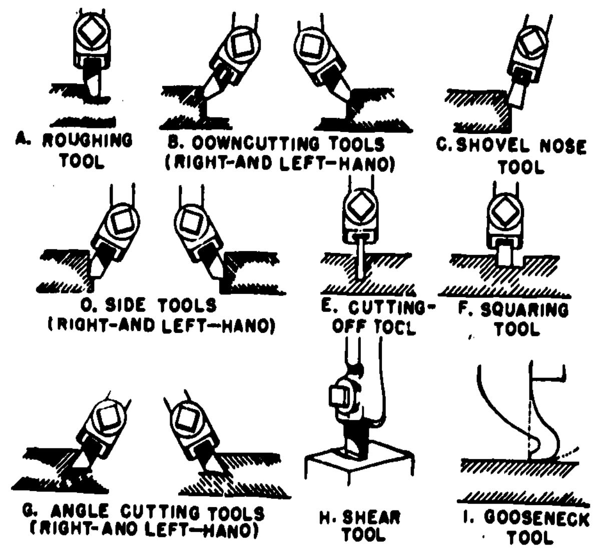
For an efficient cutting tool, the side and end of the tool must be ground to give a projecting cutting edge; this is called side and end relief. If the clearance is insufficient, the tool bit will rub the work, causing excessive heat and producing a rough surface on the work. If too much relief is given the tool, the cutting edge will be weak and tend to break during the cut. The front and side clearance angles seldom exceed 3° to 5°.

In addition to having the relief angles, the tool bit must slope away from the cutting edge. This slope is know as side rake and reduces the power required to force the cutting edge into the work. The side rake angle is usually 10° or more, depending upon the type of tool and the metal being machined. Roughing tools are given no back rake although a small amount is generally required for finishing operations.

The shape and use of various standard cutting tools are illustrated in figure 6-18 and may be outlined as follows:

ROUGHING TOOL (fig. 6-18A): This tool is very efficient for general use and is designed to take heavy cuts in cast iron or steel. The roughing tool is generally ground for left-hand operation as illustrated; for special applications, the angles may be reversed for right-hand cuts. No back rake is given this tool although the side rake may be as much as 20° for soft metals. Finishing operations on small flat pieces may be performed with the roughing tool if a fine feed is used.

DOWNCUTTING TOOL (fig. 6-18B): The downcutting tool may be ground and set for either right- or left-hand operation and is used for making vertical cuts on edges, sides, and ends. The tool is substantially the same as the roughing tool described, with the exception of its position in the toolholder.



28.68

Figure 6-18.—Standard shaper and planer tools.

SHOVEL NOSE TOOL (fig. 6-18C): This tool may be used for downcutting in either a right- or left-hand direction. A small amount of back rake is required, and the cutting edge is made the widest part of the tool. The corners are slightly rounded to give them longer life.

SIDE TOOL (fig. 6-18D): Both right- and left-hand side tools are required for finishing vertical cuts. These tools may also be used for cutting or finishing small horizontal shoulders after a vertical cut has been made in order to avoid changing tools.

CUTTING-OFF TOOL (fig. 6-18E): This tool is given relief on both sides to allow free cutting action as the depth of cut is increased.

SQUARING TOOL (fig. 6-18F): This tool is similar to the cutting-off tool and may be made in any desired width. The squaring tool is used chiefly for finishing the bottom and sides of shoulder cuts, keyways, and grooves.

ANGLE CUTTING TOOL (fig. 6-18G): The angle cutting tool is adapted for finishing operations and is generally used following a roughing operation made with the downcutting tool. The tool may be ground for either right- or left-hand operation.

SHEAR TOOL (fig. 6-18H): This tool is used to produce a high finish on steel and should be operated with a fine feed. The cutting edge is ground to form a radius of 3 to 4 inches, twisted to a 20° to 30° angle, and given a back rake in the form of a small radius.

GOOSENECK TOOL (fig. 6-18I): This tool is used for finishing cast iron and must be forged so that the cutting edge is behind the backside of the tool shank. This feature allows the tool to spring away from the work slightly, reducing the tendency for gouging or chattering. The cutting edge is rounded at the corners and given a small amount of back rake.

GRINDING HANDTOOLS AND DRILLS

Tools and Their Uses, NAVPERS 10085-B, contains detailed descriptions of offhand grinding of twist drills and handtools. Therefore, these subjects are not discussed here. You should study NAVPERS 10085-B so that you can accurately grind these tools that you will often use in your work.

WHEEL CARE AND STORAGE

All grinding wheels can be broken or damaged due to mishandling and improper storage. Whenever you handle grinding wheels, take care not to drop them or bump them against other hard objects such as the grinder or other wheels.

Grinding wheels should be stored in a cabinet or on shelves large enough to allow selection of a wheel without disturbing the other wheels. The storage space should provide protection against high humidity, contact with liquids, freezing temperatures, and extreme temperature changes. Also, provisions must be made to secure grinding wheels aboard ship to prevent them from being damaged when the ship is at sea. Thin cutoff wheels should be stacked flat on a rigid surface without any separators or blotters between them. Flaring cup wheels should be stacked flat with the small ends together. All other types of wheels may be stored upright on their rims with blotters placed between them. A sheet metal cabinet, lined with felt or corrugated cardboard to prevent wheel chipping, is acceptable for storage.

CHAPTER 7

LATHES AND ATTACHMENTS

There are several types of lathes installed in shipboard machine shops including the engine lathe, horizontal turret lathe, vertical turret lathe, and several variations of the basic engine lathe, such as bench, toolroom, and gap lathes. All lathes, except the vertical turret type, have one thing in common for all usual machining operations--the workpiece is held and rotated around a horizontal axis while being formed to size and shape by a cutting tool. In a vertical turret lathe, the workpiece is rotated around a vertical axis.

All of the lathes mentioned above, as well as many of their attachments, are described in this and the next three chapters. Engine lathe operations and turret lathes and their operations are covered later in this manual.

ENGINE LATHE

An engine lathe similar to the one shown in figure 7-1 is found in every machine shop. It is used mainly for turning, boring, facing, and screw cutting, but it may also be used for drilling, reaming, knurling, grinding, spinning, and spring winding. The work held in an engine lathe can be rotated at any one of a number of different speeds. The cutting tool can be accurately controlled by hand or power for longitudinal feed and crossfeed. (Longitudinal feed is the movement of the cutting tool parallel to the axis of the lathe; crossfeed is the movement of the cutting tool perpendicular to the axis of the lathe.)

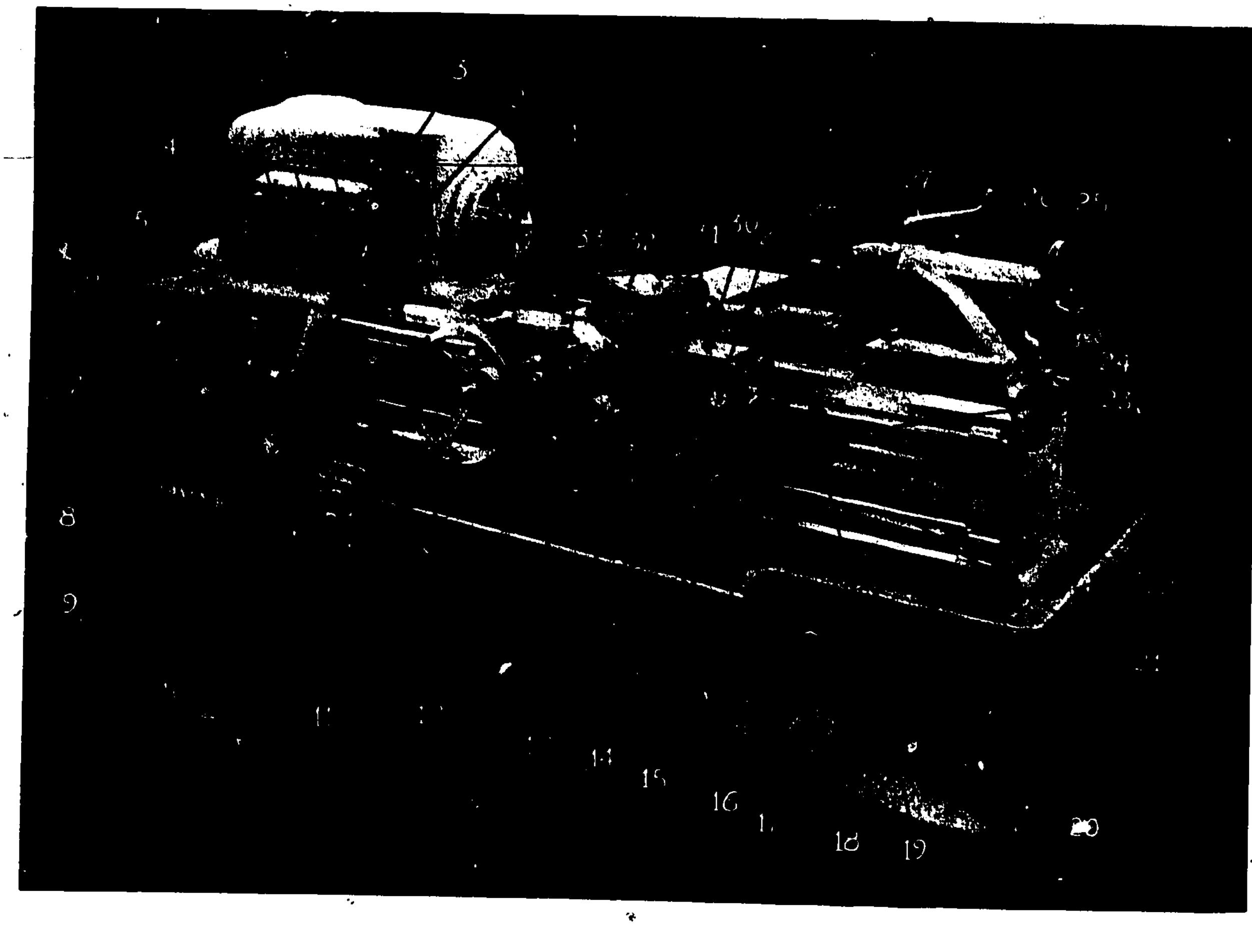
Lathe size is determined by various methods depending upon the manufacturer. Generally, the size is determined by two measurements: (1a) either the diameter of work it will swing over the bed or (1b) the diameter of work it will swing over the cross-slide and (2a) either the length of the bed or (2b) the maximum distance between centers. For example, using methods 1a and 2a, a 14-inch X 6-foot lathe has a bed that is 6 feet long and will swing work (over the bed) up to 14 inches in diameter.

Engine lathes range in size from small bench lathes with a swing of 9 inches to very large lathes for turning work of large diameters, such as low-pressure turbine rotors. A 16-inch swing lathe is a good, average size for general purposes and is usually the size installed in ships that have only one lathe.

To learn the operation of a lathe, you must be familiar with the names and functions of the principal parts. In studying the principal parts in detail, remember that lathes all provide the same general functions even though the design may differ among manufacturers. As you read the description of each part, find its location on the lathe pictured in figure 7-1. For specific details on a given lathe, refer to the manufacturer's technical manual for that machine.

BED AND WAYS

The bed is the base for the working parts of the lathe. The main feature of the bed is the ways which are formed on its upper surface and run the full length of the bed. The tailstock and carriage slide on the ways in alignment with the headstock. The headstock is permanently bolted at one end (at operator's left).



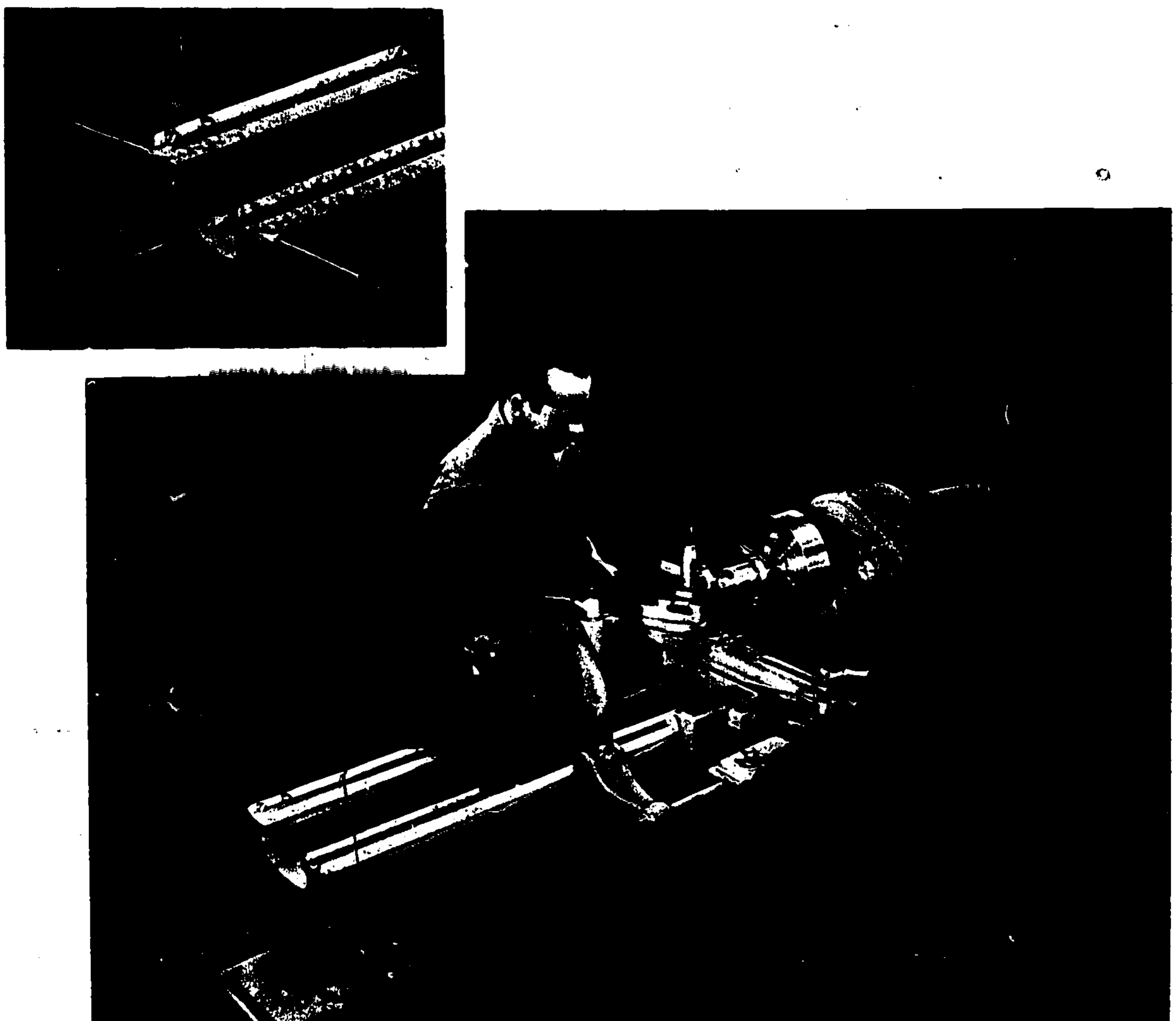
1. Headstock spindle
2. Identification plate
3. Spindle speed index plate
4. Headstock spindle speed change levers
5. Upper compound lever
6. Lower compound lever
7. Tumbler lever
8. Feed-thread index plate
9. Feed-thread lever
10. Spindle control lever
11. Electrical switch grouping
12. Apron handwheel
13. Longitudinal friction lever
14. Cross-feed friction lever
15. Feed directional control lever
16. Half nut closure lever
17. Spindle control lever
18. Leadscrew reverse lever
19. Reverse rod stop dog
20. Control rod
21. Feed rod
22. Lead screw
23. Reverse rod
24. Tailstock setover screw
25. Tailstock handwheel
26. Tailstock clamping lever
27. Tailstock spindle binder lever
28. Tailstock spindle
29. Chasing dial
30. Carriage binder clamp
31. Compound rest dial and handle
32. Thread chasing stop
33. Cross-feed dial and handle

28.69X

Figure 7-1.—Gear-head engine lathe.

Figure 7-2 shows the ways of a typical lathe. The inset shows the inverted V-shaped ways (1, 3, and 4) and the flat way (2). The ways are accurately machined parallel to the axis of the spindle and to each other. The V-ways are guides that allow the carriage and tailstock to move over them only in their longitudinal direction. The flat way, number 2, takes most of the downward thrust. The carriage slides on the outboard V-ways (1 and 4), which, because they are parallel to number 3, keep it in alignment with the headstock and tailstock at all times—an absolute necessity if accurate lathe work is to be done. Some lathe beds have two V-ways and two flat ways, while others have four V-ways.

For satisfactory performance of a lathe, the ways must be kept in good condition. A common fault of careless machinists is to use the bed as an anvil for driving arbors or as a shelf for hammers, wrenches, and chucks. Never allow anything to strike a hard blow on the ways or damage their finished surfaces in any way. Keep them clean and free of chips. Wipe them off



28.70X

Figure 7-2.—Rear view of lathe.

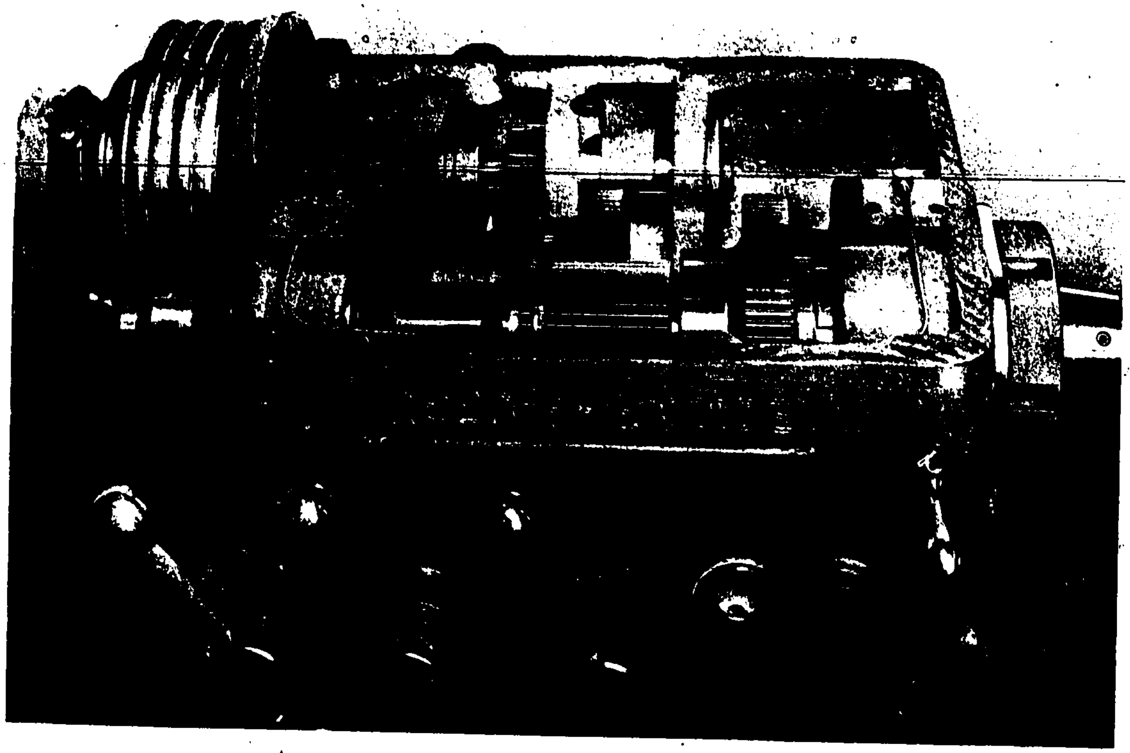


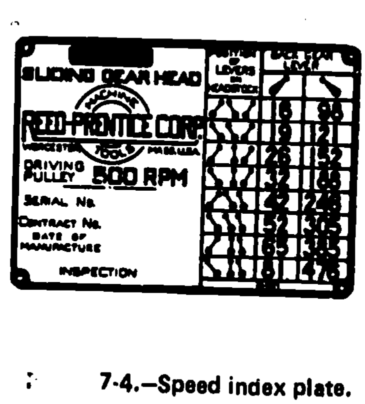
Figure 7-3.—Sliding gear type headstock. 28.72X

daily with an oiled rag to help preserve their polished surface.

HEADSTOCK

The headstock carries the headstock spindle and the mechanism for driving it. In the belt-driven type the driving mechanism consists merely of a cone pulley that drives the spindle directly or through back gears. When the spindle is driven directly, it rotates with the cone pulley; when the spindle is driven through the back gears, it rotates more slowly than the cone pulley, which in this case turns freely on the spindle. Thus two speeds are available with each position of the belt on the cone; if the cone pulley has four steps, eight spindle speeds are available.

The geared headstock shown in figure 7-3 is more complicated but more convenient to operate because speed is changed by shifting gears. This headstock is similar to an automobile transmission except that it has more gear-shift combinations and therefore has a greater number of speed changes. A speed index plate, attached to the headstock, indicates the lever positions for the different spindle speeds. Figure 7-4 shows this plate for the geared headstock in



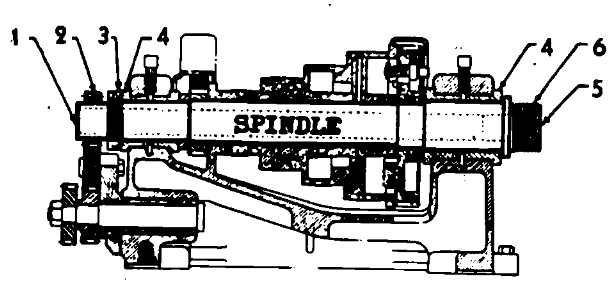
7-4.—Speed index plate. 28.73X

figure 7-3. Always stop the lathe when you shift gears to avoid possible damage to gear teeth.

Figure 7-3 shows the interior of a typical geared headstock that has 16 different spindle speeds. The driving pulley at the left is driven at a constant speed by a motor located under the headstock. Various combinations of gears in the headstock transmit the power from the drive shaft to the spindle through an intermediate shaft. Use the speed-change levers to shift the sliding gears on the drive shaft and intermediate shaft to line up the gears in different combinations. This produces the gear ratios you need to obtain the various spindle speeds. Note that the back gear lever has a high and low speed for each combination of the other gears (figure 7-4).

The headstock casing is filled with oil to lubricate the gears and the shifting mechanism contained within it. Parts not immersed in the oil are lubricated by either the splash produced by the revolving gears or by an oil pump. Be sure to keep the oil to the oil level indicated on the oil gage, and drain and replace the oil when it becomes dirty or gummy.

The headstock spindle (fig. 7-5) is the main rotating element of the lathe and is directly connected to the work which revolves with it. The spindle is supported in bearings (4) at each end of the headstock through which it projects. The section of the spindle between the bearings carries the pulleys or gears that turn the spindle. The nose of the spindle holds the driving plate, the faceplate, or a chuck. The spindle is hollow throughout its length so that bars or rods can be passed through it from the left (1) and held in a chuck at the nose. The chuck end of the spindle (5) is bored to a Morse taper to receive the LIVE center. The hollow spindle also permits the use of the draw-in collet chuck which is discussed later in this chapter. At the other end of the spindle is the gear (2) by which the spindle drives the feed and screw-cutting mechanism through a gear train located on the left end of the lathe. A collar (3) is used to adjust end play of the spindle.



28.74X

Figure 7-5.—Cross section of a belt-driven headstock.

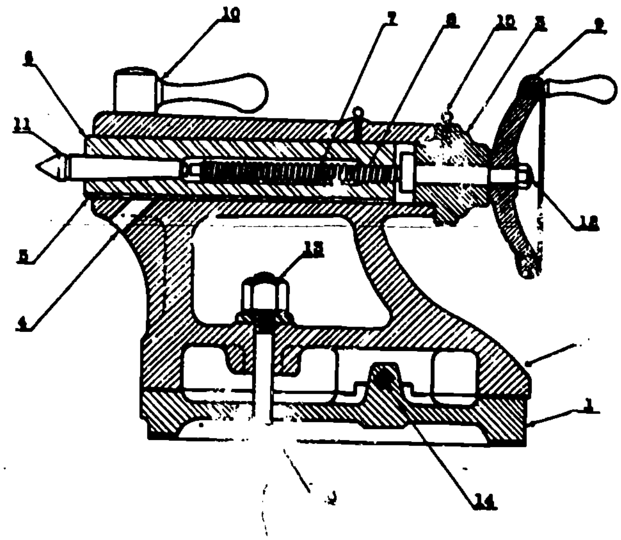
The spindle is subjected to considerable torque because it drives the work against the resistance of the cutting tool as well as drives the carriage that feeds the tool into the work. For this reason adequate lubrication and accurately adjusted bearings are absolutely necessary. (Bearing adjustment should be done only by an experienced lathe repairman.)

TAILSTOCK

The primary purpose of the tailstock (fig. 7-6) is to hold the DEAD center to support one end of work being machined on centers. However, it can also be used to hold tapered shank drills, reamers, and drill chucks. The tailstock moves on the ways along the length of the bed to accommodate work of varying lengths. It can be clamped in the desired position by the tailstock clamping nut (13).

The dead center (11) is held in a tapered hole (bored to a Morse taper) in the tailstock spindle (6). To move the spindle back and forth in the tailstock barrel for longitudinal adjustment, turn the handwheel (9) which turns the spindle-adjusting screw (7) in a tapped hole in the spindle at (8). The spindle is kept from revolving by a key (4) that fits a spline, or keyway, (5) cut along the bottom of the spindle as shown. After making the final adjustment, use the binding clamp (10) to lock the spindle in place.

The tailstock body is made in two parts. The bottom, or base (1), is fitted to the ways; the top (2) can move laterally on its base. The lateral movement can be closely adjusted by setscrews. Zero marks inscribed on the base and top indicate the center position and provide a way to measure setover for taper turning.



1. Tailstock base.
2. Tailstock top.
3. Tailstock nut.
4. Key.
5. Keyway (in spindle).
6. Spindle.
7. Tailstock screw.
8. Internal threads in spindle.
9. Handwheel.
10. Spindle binding clamp
11. Dead center.
12. End of tailstock screw.
13. Tailstock clamp nut.
14. Tailstock set-over.
15. For oiling.
16. Tailstock clamp bolt.

28.75X

Figure 7-6.—Cross section of a tailstock.

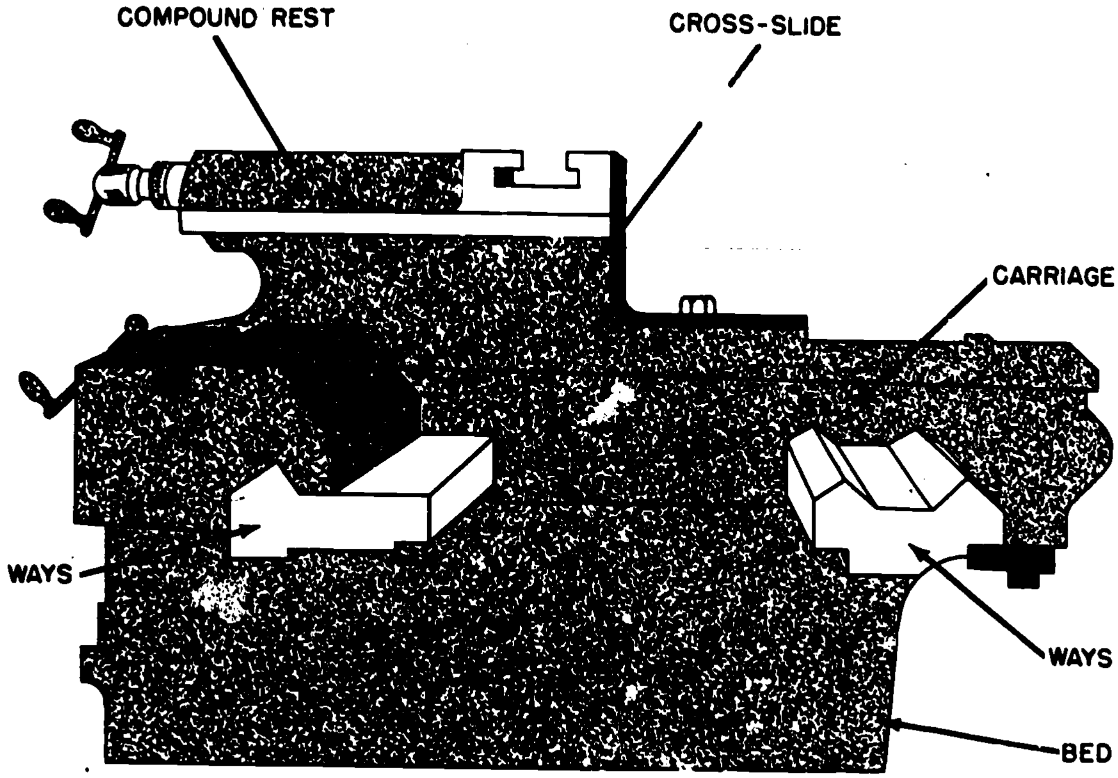
Setover of the tailstock for taper turning is described in a later chapter.

Before you insert a dead center, a drill, or a reamer into the spindle, carefully clean the tapered shank and wipe out the tapered hole of the spindle. After you put a drill or a reamer into the tapered hole of the spindle, be sure to tighten the spindle so that the tool will not revolve. If the drill or reamer is allowed to revolve, it will score the tapered hole and destroy its accuracy. The spindle of the tailstock is engraved with graduations which help in determining the depth of a cut when you drill or ream.

CARRIAGE

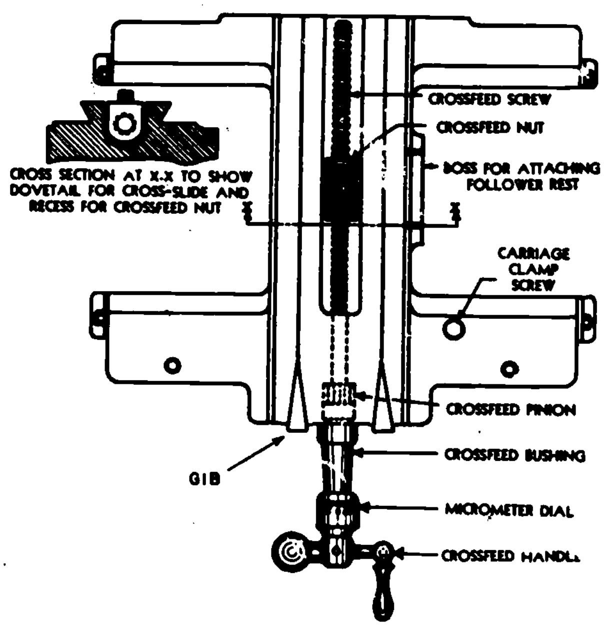
The carriage carries the crossfeed slide and the compound rest which in turn carries the cutting tool in the toolpost. The carriage slides on the ways along the bed (fig. 7-7).

Figure 7-8 shows a top view of the carriage. The wings of the H-shaped saddle contain the bearing surfaces which are fitted to the V-ways of the bed. The crosspiece is machined to form a dovetail for the crossfeed slide. The crossfeed slide is closely fitted to the dovetail and has a tapered gib which fits between the carriage dovetail and the matching dovetail of the crossfeed slide. The gib permits small adjustments to remove any looseness between the two parts. The slide is securely bolted to the crossfeed nut which moves back and forth when the crossfeed screw is turned by the handle. The micrometer dial on the crossfeed handle is graduated to permit accurate infeed. Depending on the manufacturer of the lathe, the dial may be graduated so that each division represents a 1



28.76(28D)

Figure 7-7.—Side view of a carriage mounted on the bed.



28.77X

Figure 7-8.—Carriage (top view).

to 1 ratio or a 2 to 1 ratio. The compound rest is mounted on top of the crossfeed slide.

The carriage has T-slots or tapped holes for clamping work for boring or milling. When the lathe is used in this manner, the carriage movement feeds the work to the cutting tool which is revolved by the headstock spindle.

You can lock the carriage in any position on the bed by tightening the carriage clamp screw. Use the clamp screw only when doing such work as facing or cutting-off for which longitudinal feed is not required. Normally, keep the carriage clamp in the released position. Always move the carriage by hand to be sure it is free before you apply the automatic feed.

APRON

The apron is attached to the front of the carriage. It contains the mechanism that controls the movement of the carriage for longitudinal feed and thread cutting and controls the lateral movement of the cross-slide. You should thoroughly understand the construction and operation of the apron before you attempt to operate the lathe.

In general, a lathe apron contains the following mechanical parts:

1. A longitudinal feed HANDWHEEL for moving the carriage by hand along the bed. This handwheel turns a pinion that meshes with a rack gear secured to the lathe bed.

2. GEAR TRAINS driven by the feed rod. These gear trains transmit power from the feed rod to move the carriage along the ways and to move the cross-slide across the ways, thus providing powered longitudinal feed and crossfeed.

3. FRICTION CLUTCHES operated by knobs on the apron to engage or disengage the power-feed mechanism. (Some lathes have a separate clutch for longitudinal feed and crossfeed; others have a single clutch for both.) NOTE: The power feeds are usually driven through a friction clutch to prevent damage to gears if excessive strain is put on the feed mechanism. If clutches are not provided, there is some form of safety device that operates to disconnect the feed rod from its driving mechanism.

4. A selective FEED LEVER or knob for engaging the longitudinal feed or crossfeed as desired.

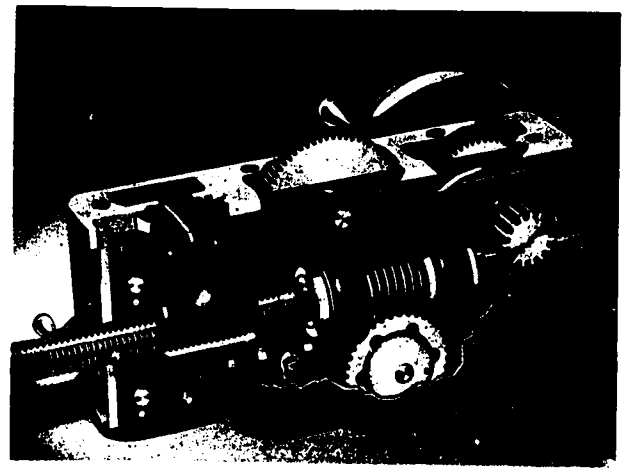
5. HALF-NUTS that engage and disengage the lead screw when the lathe is used to cut threads. They are opened or closed by a lever located on the right side of the apron. The half-nuts fit the thread of the lead screw which turns in them like a bolt in a nut when they are clamped over it. The carriage is then moved by the thread of the lead screw instead of by the gears of the apron feed mechanisms. (The half-nuts are engaged only when the lathe is used to cut threads, at which time the feed mechanism must be disengaged. An interlocking device that prevents the half-nuts and the feed mechanism from engaging at the same time is usually provided as a safety feature.)

Aprons on lathes made by different manufacturers differ somewhat in construction and in the location of controlling levers and knobs. But, they all are designed to perform the same functions. The principal difference is in the arrangement of gear trains for driving the automatic feeds. For example, in some aprons there are two separate gear trains with separate operating levers for longitudinal feed and crossfeed. In others, both feeds are driven from the same driving gear on the feed rod through a common clutch and have a selective lever for connecting the drive to either longitudinal feed or crossfeed. The apron shown in figure 7-9 is of the latter type.

FEED ROD

The feed rod transmits power to the apron to drive the longitudinal feed and crossfeed mechanisms. The feed rod is driven by the spindle through a train of gears, and the ratio of its speed to that of the spindle can be varied by changing gears to produce various rates of feed. The rotating feed rod drives gears in the apron; these gears in turn drive the longitudinal feed and crossfeed mechanisms through friction clutches, as explained in the description of the apron.

Lathes which do not have a separate feed rod have a spline in the lead screw to serve the



28.79X

Figure 7-9.—Rear view of a lathe apron.

same purpose. The apron shown in figure 7-9 belongs to a lathe of this type and shows clearly how the worm which drives the feed mechanism is driven by the spline in the lead screw. If a separate feed rod were used, it would drive the feed worm in the same manner, that is, by means of a spline. The spline permits the worm, which is keyed to it, to slide freely along its length to conform with the movement of the carriage apron.

LEAD SCREW

The lead screw is used for thread cutting. Along its length are accurately cut Acme threads which engage the threads of the half-nuts in the apron when half-nuts are clamped over it. When the lead screw turns in the closed half-nuts, the carriage moves along the ways a distance equal to the lead of the thread in each revolution of the lead screw. Since the lead screw is connected to the spindle through a gear train (discussed later in the section on quick-change gear mechanism), the lead screw rotates with the spindle. Therefore, whenever the half-nuts are engaged, the longitudinal movement of the carriage is directly controlled by the spindle rotation. The cutting tool is moved a definite distance along the work for each revolution that the spindle makes.

The ratio of the threads per inch of the thread being cut and the thread of the lead screw is the same as the ratio of the speeds of the spindle and the lead screw. For example: If the lead screw and spindle turn at the same speed, the number of threads per inch being cut is the same as the number of threads per inch of the lead screw. If the spindle turns twice as fast as the lead screw, the number of threads being cut is twice the number of threads per inch of the lead screw.

You can cut any number of threads by merely changing gears in the connecting gear train to get the desired ratio of spindle and lead screw speeds.

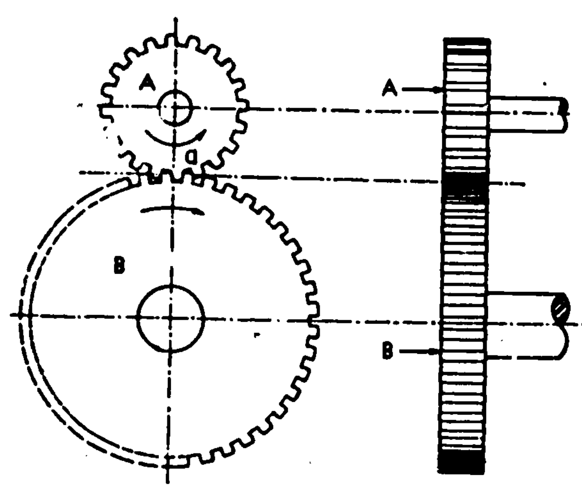
GEARING

First, consider the simplest possible arrangement of gearing between the spindle and the lead screw—a gear on the end of the spindle meshed with a gear on the end of the lead screw, as shown in figure 7-10. Let a be point of contact between the spindle gear A and the screw gear B. As each tooth on gear A passes point a, it causes a tooth on B to pass this same point. Suppose gear A has 20 teeth and gear B has 40 teeth. Then when A makes one complete turn, 20 teeth will have passed point a. Since B has 40 teeth around its rim, only half of them will have passed point a. Gear B has made just one-half of a revolution while gear A has made one revolution. In other words, gear B with 40 teeth will turn half as fast as gear A with 20 teeth, or their speeds are inversely proportional to their size. This relation may be expressed as follows:

$$\frac{\text{rpm of B}}{\text{rpm of A}} = \frac{\text{number of teeth on A}}{\text{number of teeth on B}}$$

or

$$\frac{\text{rpm of lead screw}}{\text{rpm of spindle}} = \frac{\text{number of teeth on spindle gear A}}{\text{number of teeth on screw gear B}}$$



28.81X

Figure 7-10.—A simple gear arrangement.

By using this formula, you can change the speed of the screw relative to that of the spindle by changing the gears to get the ratio desired.

In figure 7-10, the ratio is 20:40 or 1:2. Any combination of gears that has a ratio of 1:2, such as 30 and 60 or 35 and 70, will cause the lead screw to turn half as fast as the spindle.

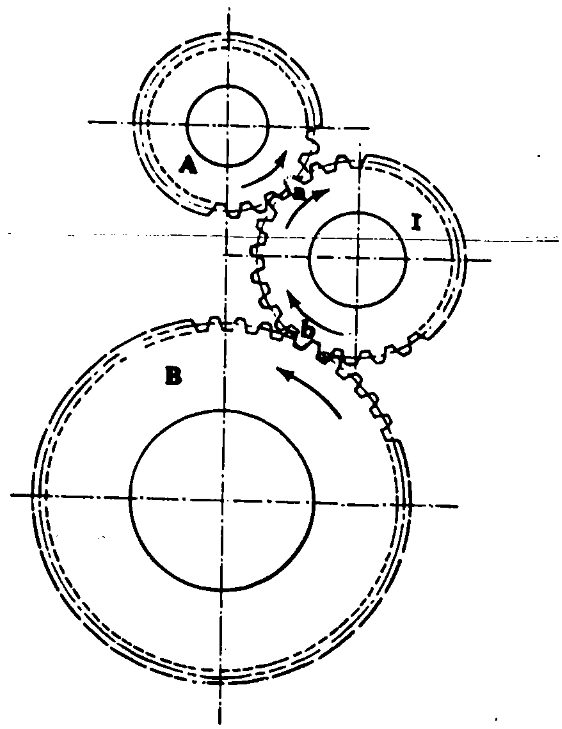
Suppose you want to cut 8 threads per inch on a lathe that has a lead screw with 6 threads per inch. The carriage must carry the thread-cutting tool 1 inch along the work whi' the work makes eight complete revolutions. Since the lead screw has 6 threads per inch, it must revolve six times in the half-nuts to move the carriage 1 inch. Therefore, you must gear the lathe to cause the lead screw to make six revolutions while the spindle makes eight revolutions. In other words, the lead screw must turn 6/8 or 3/4 as fast as the spindle. Since the speeds will be proportional to the size of the gears, you can use any two gears having this ratio, such as 30 and 40, 33 and 44, 45 and 60, the smaller of the two going on the spindle (use the inverse ratio formula).

By now you should have discovered that the ratio in threads per inch of the thread to be cut and the lead screw is identical to the ratio of the number of teeth of the change gears. If the spindle gear is smaller than the screw gear, the thread cut will be finer (more threads per inch) than the lead screw and vice versa.

Idler Gears

It is obviously impracticable to have the spindle gear mesh directly with the screw gear because, for one thing, the distance between them is so great that the gears required would be too large. Therefore, smaller gears of the desired ratio are used, and idler gears bridge the gap between them. You can place any number of idler gears between the driving gear and the driven gear without changing the original gear ratio. The idler gears allow the lead screw and spindle gears to rotate as if they were in direct contact.

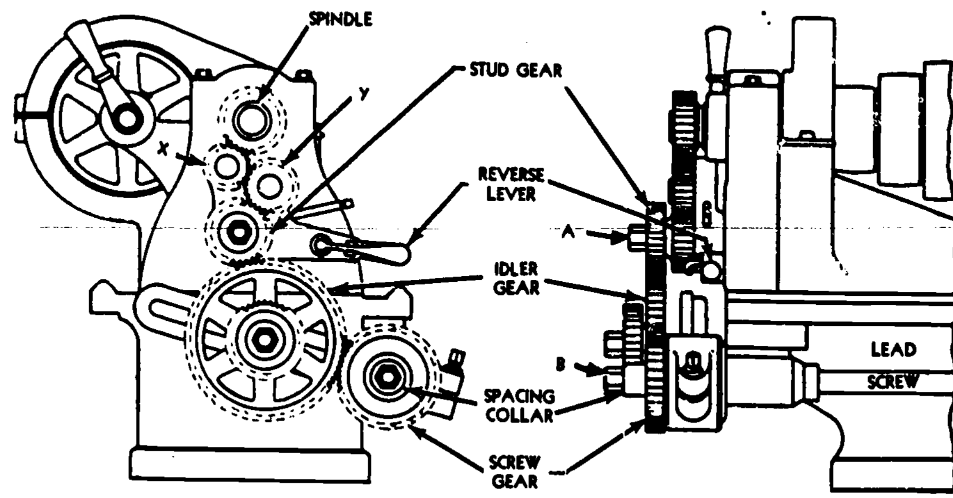
In figure 7-11, I is an idler gear inserted between the driving gear A and the driven gear



28.82X
Figure 7-11.—Idler gear inserted between a driving gear and a driven gear.

B. Suppose that A has 20 teeth. In making one complete revolution, all of these 20 teeth will pass a given point a and cause 20 teeth on I to pass this same point. If 20 teeth on I pass point a, an equal number of teeth on I will pass point b where gear B meshes with it. Gear B will be moved the same distance as it would if it were directly meshed with A; so the ratio between their speeds remains the same, but the direction of rotation of B is reversed. Idler gears, then, are used for two purposes: (1) to connect gears in a gear train and (2) to reverse the direction of rotation of a gear-driven mechanism.

Figure 7-12 is an example of simple gearing used on a change gear lathe. The gear on the spindle drives the stud gear shaft A at a fixed ratio, usually 1:1, in which the stud gear revolves at the same speed as the spindle. Between the spindle and the stud are the idler gears X and Y mounted on the movable bracket



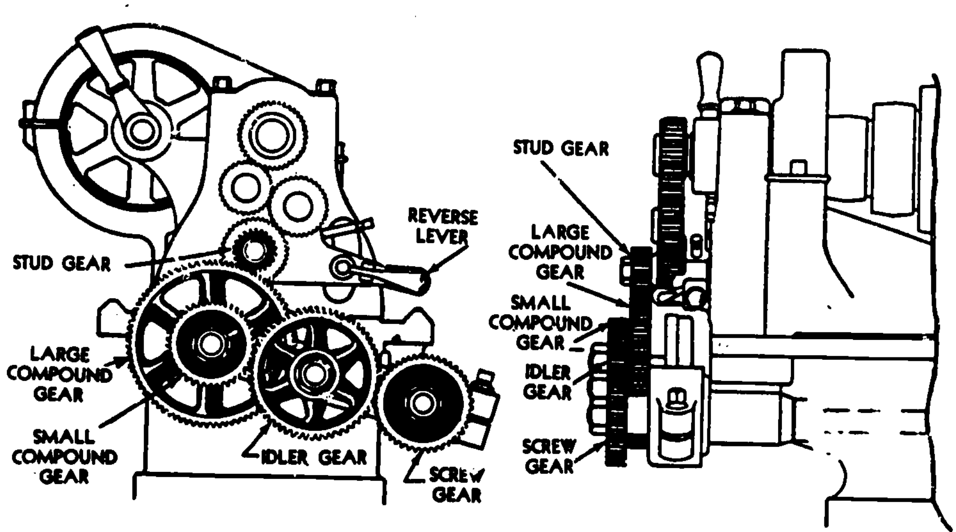
28.83X

Figure 7-12.—Simple gearing on a lathe.

controlled by the reverse lever. When this lever is in the down position, both X and Y are connected in the gear train as shown, and the stud shaft revolves in a direction opposite to that of the spindle; when the lever is raised, gear X is disengaged from the train, and gear Y is meshed directly between the spindle and the stud, thereby reversing the previous direction of the stud gear and all the gears that follow it. NOTE: The reverse lever has a neutral position that disconnects the spindle from the gear train.

The lathe shown in figure 7-12 has permanently mounted spindle and idler gears (X and Y). To vary the thread cutting gear ratios, change the stud gear and the screw gear. You can determine which gears on your machine must be changed by reading the lathe's operating instructions.

Note in figure 7-13 that the stud shaft A has an extension on which is mounted a removable gear called the stud gear. (In comparing the gear



28.84X

Figure 7-13.—Compound gearing on a lathe.

train in fig. 7-13 with the simple gear train described previously, we can consider the stud gear as being mounted directly on the spindle, since it always revolves at a fixed ratio with it.) The screw gear shown is also a removable gear mounted on the lead screw at B. Now all that you need to do to change the gear ratio to cut different threads is to place gears with the correct number of teeth on the stud shaft A and connect. Change gear lathes are equipped with an assortment of gears which provide enough changes for all practical purposes.

A simple rule to follow in determining what gears to use on stud and screw is: Multiply the desired number of threads per inch and the number of threads per inch in the lead screw by the same number; if the products correspond to the number of teeth in any two of the change gears at hand, use those gears; if not, use some other multiplier that will give products to match the gears available. For example, if you want to cut a screw containing 16 threads per inch on the lathe with a lead screw that has 6 threads per inch, use 5 for a multiplier:

$$5 \times 16 = 80$$

$$5 \times 6 = 30$$

If gears with 80 teeth and 30 teeth are on hand, use the 30-tooth gear as the stud gear and the 80-tooth gear as the screw gear. If you do not have those gears, try other multipliers until you arrive at a combination corresponding to gears available.

If you cannot get the proper ratio of gears with the change gears you have at hand or the gears would be too small or too large to connect properly or conveniently (as would be the case if you were cutting very fine threads), you should use COMPOUNDING. Compounding means substituting two gears for an intermediate gear. Compounding changes the ratio of the gear train by the same ratio that the compounding gears bear to each other.

Figure 7-13 shows a compound gear train on a change gear lathe. The only way it differs from the simple gear train (fig. 7-12) is that the two extra gears rotating as one on a common axis are installed in the train following the stud gear. Compounding gears for a lathe usually have a ratio of 2 to 1; they double the ratio that would exist if simple gearing were used.

If a 2:1 compound gear is installed in the manner shown in figure 7-13, the speed transmitted by the stud gear to the large compound gear is reduced by half when it is retransmitted by the small compound gear to the gears that follow. It amounts to the same thing as using a stud gear with half as many teeth.

The advantage of compounding is best demonstrated by the following example:

Suppose a gear ratio of 10 to 1 is required to cut a certain fine thread, and the smallest gear you have that will fit the stud has 20 teeth. You would need a screw gear with 200 teeth, but such a gear is far too large. However, by using a 2:1 compound gear in the manner illustrated, you would need a gear only half as large (100 teeth).

Quick-Change Gear Mechanism

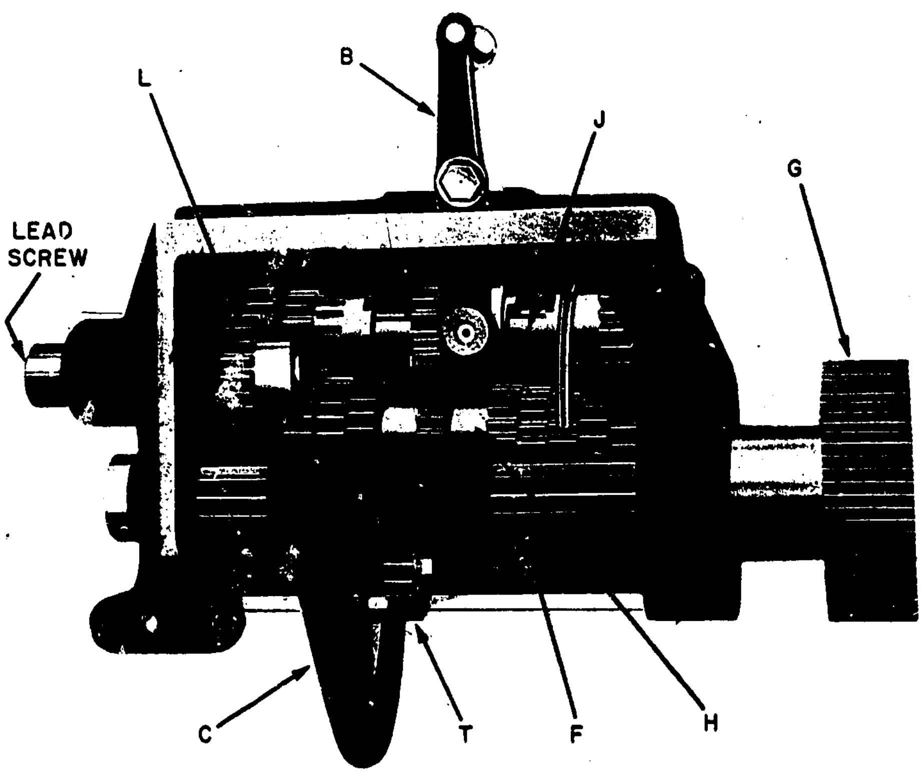
To do away with the inconvenience and loss of time involved in removing and replacing change gears, most modern lathes have a self-contained change gear mechanism, commonly called the QUICK-CHANGE GEAR BOX. There are a number of types used on different lathes but they are all similar in principle.

The mechanism consists of a cone-shaped group of change gears. You can instantly connect any single gear in the gear train by a sliding tumbler gear controlled by a lever. The cone of gears is keyed to a shaft which drives the lead screw (or feed rod) directly or through an intermediate shaft. Each gear in the cluster has a different number of teeth and hence produces a different gear ratio when connected in the train. The same thing happens as when the screw gear in the gear train is changed, described previously. Sliding gears also produce other changes in the gear train to increase the number of different ratios you can get with the cone of change gears described above. All changes are made by shifting appropriate levers or knobs. An index plate or chart mounted on the gear box indicates the position for placing the levers to get the necessary gear ratio to cut the thread or produce the feed desired.

Figure 7-14 is the rear view of one type of gear box, showing the arrangement of gears. The splined shaft F turns with gear G, which is driven by the spindle through the main gear train on the end of the lathe. Shaft F in turn drives shaft H through the tumbler gear T which can be engaged with any one of the cluster of eight different size gears on shaft H by means of the lever C. Shaft H drives shaft J through a double clutch gear, which takes the drive through one of three gears, depending on the position of lever B (right, center, or left). Shaft J drives the lead screw through gear L.

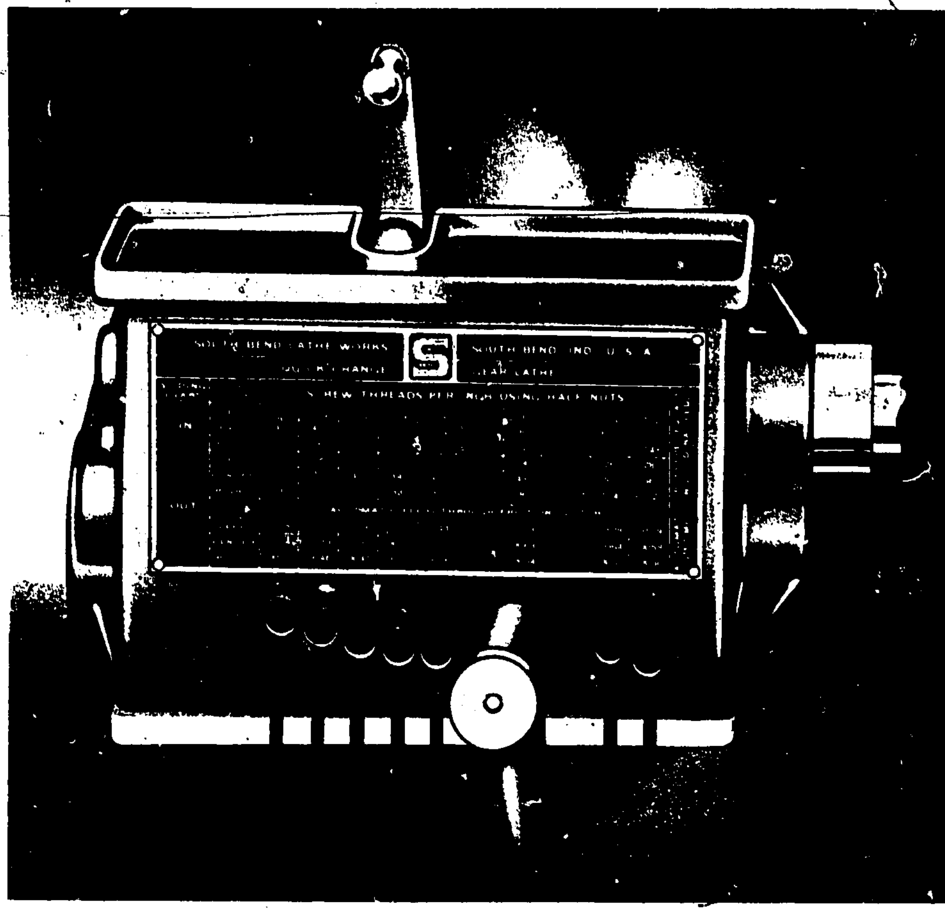
Either the lead screw or the feed rod can be connected to the final driveshaft of the gear box by engaging appropriate gears. The lathe gear box illustrated in figure 7-14 has no feed rod.

Twenty-four different gear ratios are provided by the quick-change gear box. The lower lever has eight positions (fig. 7-15), each of which places a different gear in the gear train and hence produces eight different gear ratios. The three positions of the upper level produce three different gear ratios for each of the 8 changes obtained with the lower lever, thus making 24 combinations in the box alone. You can double this range by using a sliding compound gear which provides a high- and low-gear ratio in the main gear train. This gives two ratios for every combination obtainable in the box, or 48 combinations in all.



28.85X

Figure 7-14.—Quick-change gear box (rear view).



28.87X

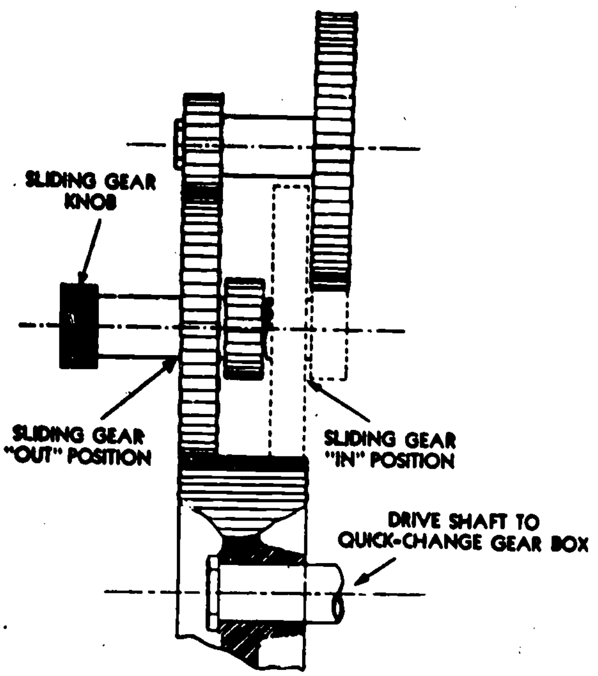
Figure 7-15.—Quick-change gear box.

Figure 7-16 shows how the sliding compound gear produces two different gear ratios when it is moved in or out. The wide gear at the bottom corresponds to gear C in figure 7-14.

INSTRUCTIONS FOR OPERATION.—If you are to cut 16 threads per inch, locate the number 16 on the index plate in the first column and fourth line under SCREW THREADS PER INCH (fig. 7-15). Adust the sliding gear knob (fig. 7-16) to the OUT position as indicated opposite 16 in the first column at the left (fig. 7-15). (You must stop the lathe to adjust the sliding gear.) Start the lathe and set top lever B (fig. 7-14) to the LEFT position as indicated in the second column, opposite 16 (fig. 7-15).

With the lathe running, shift the tumble lever C to position directly under the column in which 16 is located; rock it until the gears mesh and the handle plunger latches in the hole provided. The lathe is now set to cut the desired thread if the half-nuts are clamped on the lead screw.

ADJUSTING THE GEAR BOX FOR POWER FEEDS.—The index chart on the gear box also shows the various rates of power longitudinal feed per spindle revolution that you can get by using the feed mechanism of the apron. For example, in figure 7-15, note that the



28.86X

Figure 7-16.—Showing how the gear ratio is changed by sliding gear.

finest longitudinal feed is 0.0030 inch per revolution of spindle, the next finest is 0.0032 inch, etc. To arrange the gear box for power longitudinal feed, select the feed you wish to use and follow the same procedure explained for cutting screw threads, except that you engage the power feed lever instead of the half-nuts. Crossfeeds are not listed on the chart but you can determine them by multiplying the longitudinal feeds by 0.375, as noted on the index plate.

On a lathe with a separate feed rod, a feed-thread shifting lever located at the gear box (part 9 in fig. 7-1) connects the drive to the feed rod or lead screw as desired. When the feed rod is engaged, the lead screw is disengaged and vice versa.

You can cut metric threads on some lathes that have an English lead screw (threads per inch machined on it) by transposing a set of gears in the lead screw to the spindle gear train that provides the correct conversion ratio. You can find information on this from handbooks for machinists, the equipment technical manual, and direct correspondence with the equipment manufacturer.

COMPOUND REST

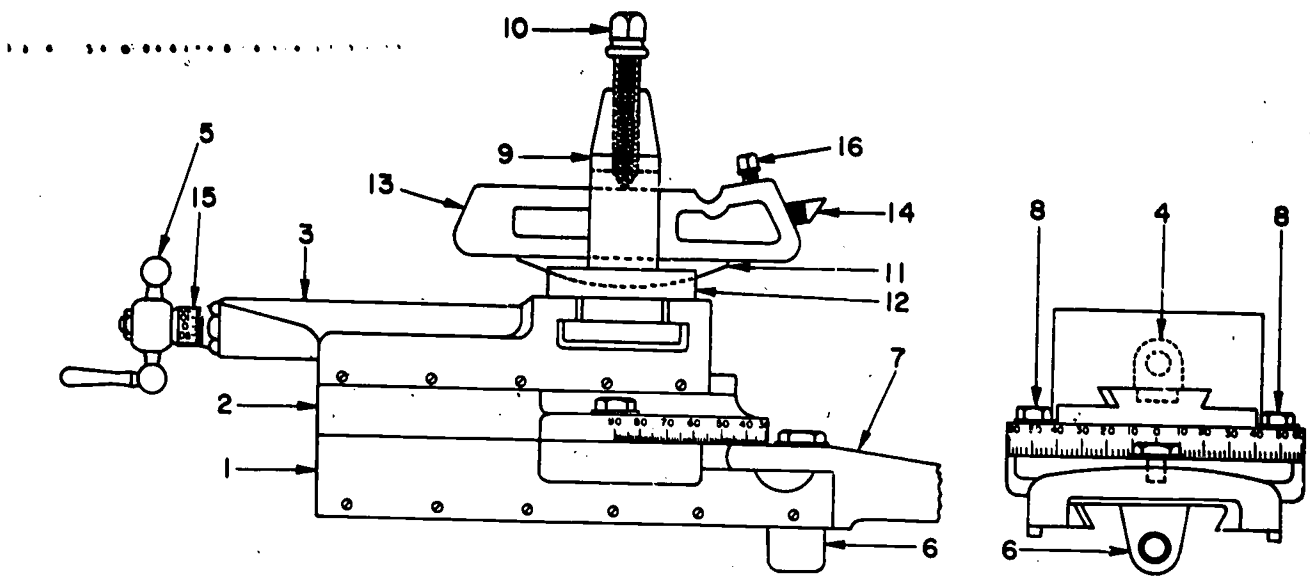
The compound rest provides a rigid adjustable mounting for the cutting tool. The compound rest assembly has the following principal parts (fig. 7-17):

1. The compound rest SWIVEL (2) which can be swung around to any desired angle and clamped in position. It is graduated over an arc of 90° on each side of its center position for ease in setting to the angle you select. This feature is used in machining short, steep tapers such as the angle on bevel gears, valve disks, and lathe centers.

2. The compound rest TOP, or TOPSLIDE (3), is mounted as shown on the swivel section (2) on a dovetailed side. It is moved along the slide by the compound rest feed screw turning in nut (4), operated by handle (5), in a manner similar to the crossfeed described previously. This provides for feeding at any angle (determined by the angular setting of the swivel section), while the cross-slide feed provides only for feeding at right angles to the axis of the lathe. The graduated collar on the compound rest feed screw reads in thousandths of an inch for fine adjustment in regulating the depth of cut.

ATTACHMENTS AND ACCESSORIES

Accessories are the tools and equipment used in routine lathe machining operations. Attachments are special fixtures which may be secured to the lathe to extend the versatility of the lathe to include taper-cutting, milling, and grinding. Some of the common accessories and attachments used on lathes are described in the following paragraphs.



1. Cross-slide.
2. Compound rest swivel.
3. Compound rest top.
4. Compound rest nut.
5. Compound rest feed screw handle.
6. Crossfeed nut.
7. Chip guard.
8. Swivel securing bolts.
9. Toolpost.
10. Toolpost setscrew.
11. Toolpost wedge.
12. Toolpost ring.
13. Toolholder.
14. Cutting tool.
15. Micrometer collar.

28.88X

Figure 7-17.—Compound rest.

TOOLPOSTS

Three popular types—standard, castle, and quick change toolposts are discussed in the following paragraphs. The sole purpose of the toolpost is to provide a rigid support for the toolholder.

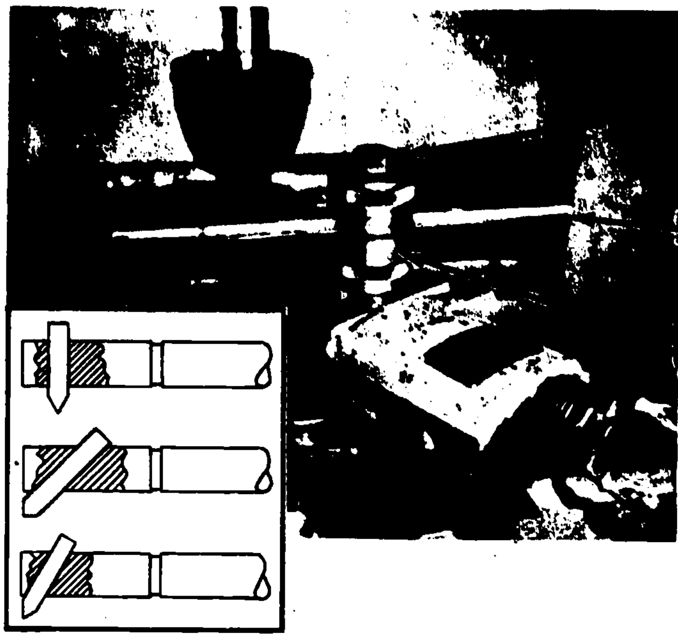
The standard toolpost is mounted in the T-slot of the compound rest top as shown in figure 7-17. A forged tool or a toolholder (13) is inserted in the slot in the toolpost and rests on the toolpost wedge (11) and the toolpost ring (12). By tightening setscrew (10), you clamp the whole unit firmly in place with the tool in the desired position.

The castle type toolpost (fig. 7-18) is used with boring bar type toolholder. It mounts in the T-slot and the toolholder (boring bar) passes through the toolpost and the holddown bolt. By tightening the locking nut, you clamp the entire unit firmly in place. Various size holes through the toolpost allow the use of assorted diameter boring bars.

The quick change type toolpost (fig. 7-19) is available in many Navy machine ships. It mounts in the T-slots and is tightened in place by the locknut, which clamps the toolpost firmly in place. Special type toolholders are used in conjunction with this type toolpost and are held in place by a locking plunger which is operated by the toolholder locking handle. Some toolposts have a sliding gib to lock the toolholder. With this type toolpost, only the toolholders are changed, allowing the toolpost to remain firmly in place.

TOOLHOLDERS

Lathe toolholders are designed to be used with the various types of toolposts. Only the

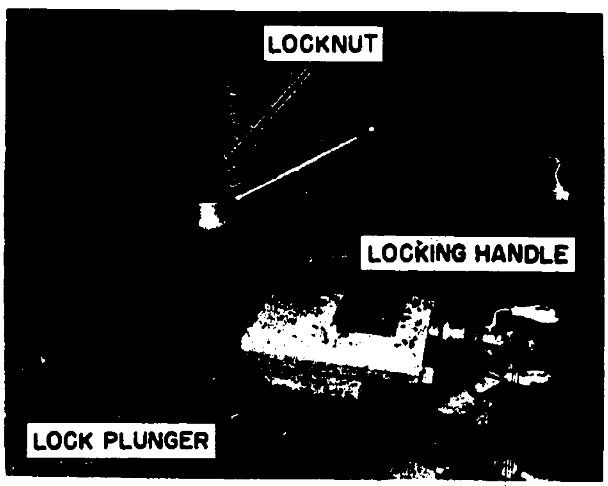


28.299

Figure 7-18.—Castle type toolpost and toolholder.

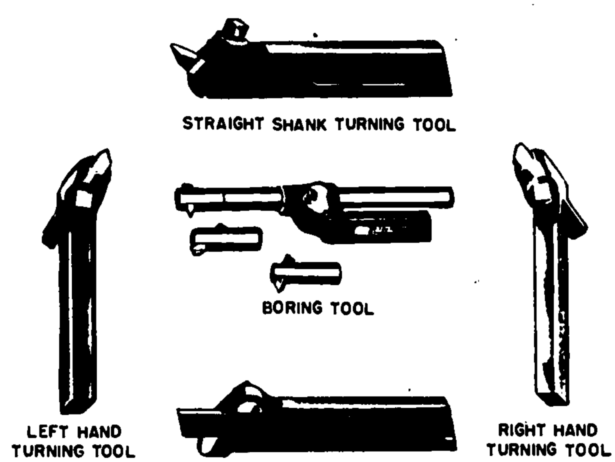
three most commonly used types—standard, boring bar, and quick change—are discussed in this chapter. The toolholder holds the cutting tool (toolbit) in a rigid and stable position. Toolholders are generally made of a softer material than the cutting tool. They are large in size and help to carry the heat generated by the cutting action away from the point of the cutting tool.

Standard type toolholders were discussed briefly in chapter 6 of this manual. However, there are more types (fig. 7-20) than those



28.300

Figure 7-19.—Quick change toolpost.



28.67

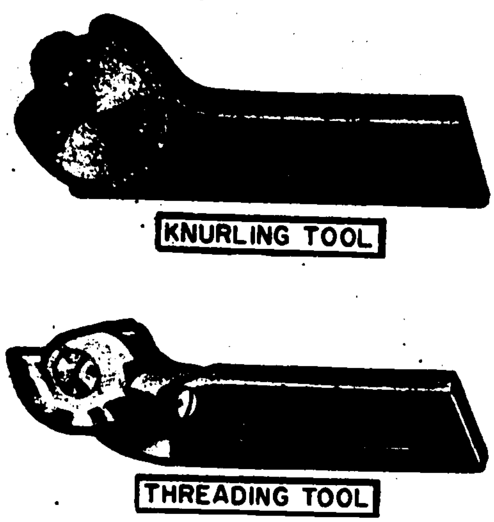
Figure 7-20.—Standard-type lathe toolholders.

discussed in chapter 6. Two that differ slightly from others are the threading and knurling toolholders. (See fig. 7-21.)

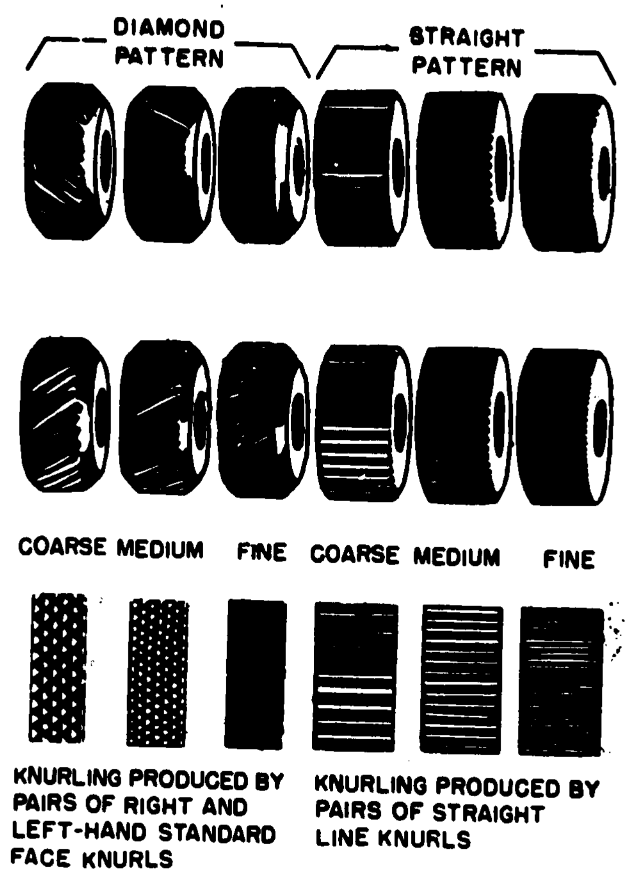
The THREADING TOOL shown in figure 7-21 has a formed cutter which needs to be ground on the top surface only for sharpening, the thread form being accurately shaped over a large arc of the tool. As the surface is worn away by grinding, you can rotate the cutter to the correct cutting position and secure it there by the setscrew. NOTE: The threading tool is not commonly used. It is customary to use a regular toolholder with an ordinary tool bit ground to the form of the thread desired.

A KNURLING TOOL (fig. 7-21) carries pattern on the work by being fed into the work as it revolves. The purpose of knurling is to give a roughened surface on round metal parts like knobs, to give a better grip for handling. The knurled roll comes in a wide variety of patterns. (See fig. 7-22.)

The BORING BAR type toolholder is nothing more than a piece of round stock with a screw-on cap. (See fig. 7-18.) The caps are available with square holes broached through them at various angles (fig. 7-18) and sizes.



28.67X
Figure 7-21.—Knurling and threading tools.



28.301
Figure 7-22.—Types of knurling rolls.

When the proper size toolbit is inserted into the cap and the cap is screwed on to the threaded end of the piece of round stock, the entire unit becomes a very rigid boring tool which is used in conjunction with the castle type toolpost.

The QUICK CHANGE type toolholder (fig. 7-23) is mounted on the toolpost by sliding it from above and downward over the dovetails. This toolholder has a height adjusting ring to allow you to set the proper height prior to locking it in place. The quick change toolholder comes in a wide range of styles. A few of these styles are shown in figure 7-24.

LATHE CHUCKS

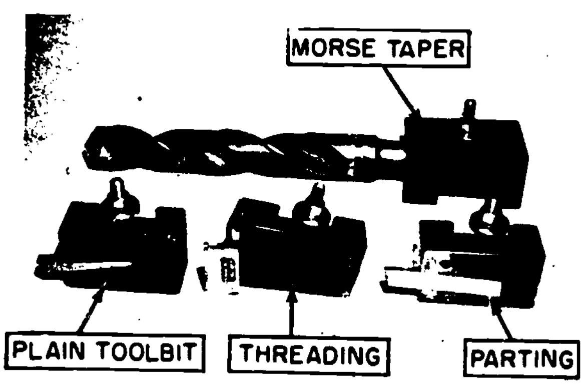
The lathe chuck is a device for holding lathe work. It is mounted on the nose of the spindle.

28.302
Figure 7-23.–Quick change toolpost and toolholder.

28.304
Figure 7-25.–Four-jaw independent chuck.

The work is held by jaws which can be moved in radial slots toward the center to clamp down on the sides of the work. These jaws are moved in and out by screws turned by a chuck wrench applied to the sockets located at the outer ends of the slots.

The 4-JAW INDEPENDENT lathe chuck, figure 7-25, is the most practical for general work. The four jaws are adjusted one at a time, making it possible to hold work of various shapes and to adjust the center of the work to coincide with the axial center of the spindle.



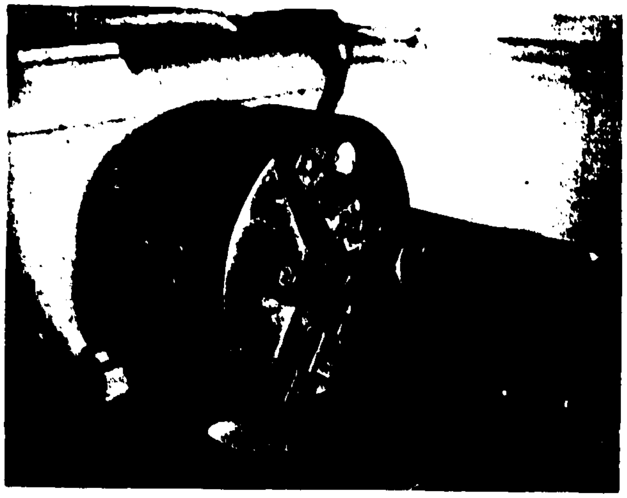
28.303
Figure 7-24.–Quick change toolholder.

There are several different styles of jaws for 4-jaw chucks. You can remove some of the chuck jaws by turning the adjusting screw and then re-inserting them in the opposite direction. Some chucks have two sets of jaws, one set being the reverse of the other. Another style has jaws that are bolted onto a slide by two socket-head bolts. On this style you can reverse the jaws by removing the bolts, reversing the jaws and re-inserting the bolts. You can make special jaws for this style chuck in the shop and machine them to fit a particular size OD or ID.

The 3-JAW UNIVERSAL or scroll chuck (fig. 7-26) can be used only for holding round or hexagonal work. All three jaws move in and out together in one operation. They move simultaneously to bring the work on center automatically. This chuck is easier to operate than the four-jaw type, but when its parts become worn you cannot rely on its accuracy in centering. Proper lubrication and constant care in use are necessary to ensure reliability. The same styles of jaws available for the 4-jaw chuck are also available for the 3-jaw.

COMBINATION CHUCKS are universal chucks having independent movement of each jaw in addition to the universal movement.

Figures 7-3 and 7-5 illustrate the usual means provided for attaching chucks and



28.305
Figure 7-26.—Three-jaw universal chuck.

28.91X
Figure 7-27.—Draw-in collet chuck assembled.

faceplates to lathes. The tapered nose spindle (fig. 7-3) is usually found on lathes that have a swing greater than 12 inches. Matching internal tapers and keyways in chucks for these lathes ensure accurate alignment and radial locking. A free turning, internally threaded collar on the spindle screws on a boss on the back of the chuck to secure the chuck to the spindle nose. On small lathes, chucks are screwed directly on the threaded spindle nose. (See fig. 7-5.)

The DRAW-IN COLLET chuck is used to hold small work for machining in the lathe. It is the most accurate type of chuck made and is intended for precision work.

Figure 7-27 shows the parts assembled in place in the lathe spindle. The collet chuck which holds the work is a split-cylinder with an outside taper that fits into the tapered closing sleeve and screws into the threaded end of the hollow drawbar. When the handwheel is turned clockwise, the collet is pulled into the tapered sleeve, thereby decreasing the diameter of the hole in the collet. As the collet is closed around the work, the work is centered accurately and is held firmly by the chuck.

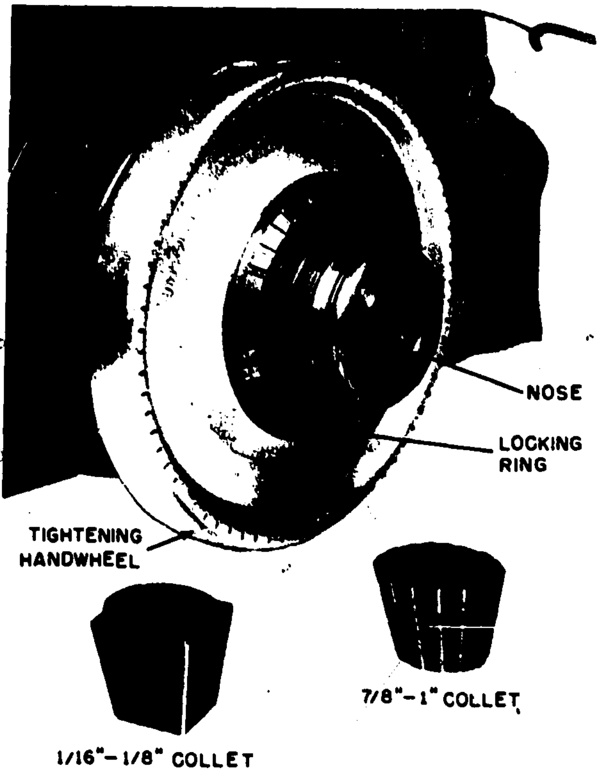
Collets are made with hole sizes ranging from 1/64 inch up, in 1/64-inch steps. The best results are obtained when the diameter of the work is exactly the same size as the dimension stamped on the collet.

To ensure accuracy of the work when using the draw-in collet chuck, be sure that the contact surfaces of the collet and closing sleeve are free of chips and dirt. NOTE: The standard collet has a round hole, but special collets for square and hexagonal shapes are obtainable.

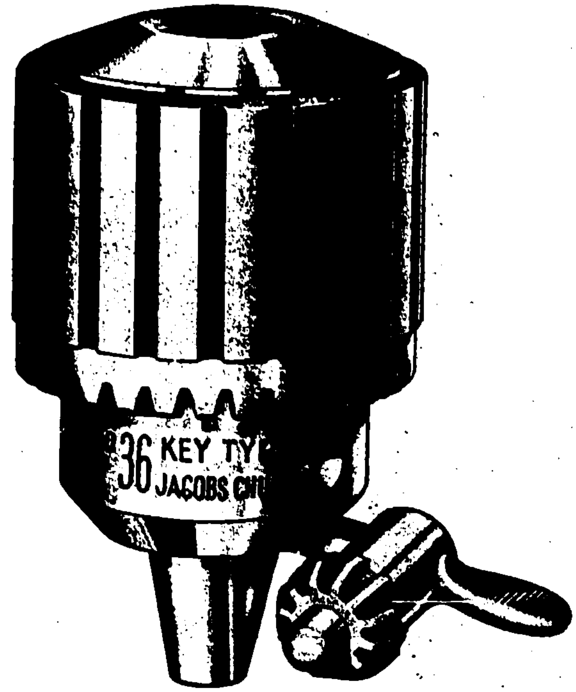
The RUBBER COLLET CHUCK (fig. 7-28) is designed to hold any bar stock from 1/16 inch up to 1 3/8 inch. It is different from the draw-in type collet previously mentioned in that the bar stock does not have to be exact in size.

The rubber flex collet consists of rubber and hardened steel plates. The nose of the chuck has external threads, and, by rotating the handwheel (fig. 7-28), you compress the collet around the bar. This exerts equal pressure from all sides and permits you to make very accurate alignment of the stock. The locking ring, when pressed in, gives a safe lock that prevents the collet from coming loose when the machine is in operation.

DRILL CHUCKS are used to hold center drills, straight shank drills, reamers, taps, or small rods. The drill chuck is mounted on a tapered shank or arbor which fits the Morse taper hole in the headstock or tailstock spindle. Figure 7-29 shows the three-jaw type. A revolving sleeve operated by a key opens or



28.306
Figure 7-28.—Rubber flex collet chuck.



28.92X
Figure 7-29.—Drill chuck.

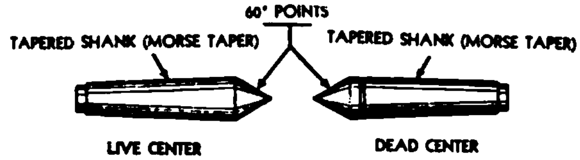
closes the three jaws simultaneously to clamp and center the drill in the chuck.

FACEPLATES are used for holding work of such shape and dimensions that it cannot be swung on centers or in a chuck. The T-slots and other openings on the surface of the faceplate provide convenient anchor points for bolts and clamps used in securing the work to it. The faceplate is mounted on the nose of the spindle.

The DRIVING PLATE is similar to a small faceplate and is used primarily for driving work that is held between centers. A radial slot receives the bent tail of a lathe dog clamped to the work and thereby transmits rotary motion to the work.

LATHE CENTERS

The lathe centers shown in figure 7-30 provide a means for holding the work between points so it can be turned accurately on its axis. The headstock spindle center is called the LIVE center because it revolves with the work. The tailstock center is called the DEAD center because it does not turn. Both live and dead centers have shanks turned to a Morse taper to fit the tapered holes in the spindles; both have points finished to an angle of 60°. They differ only in that the dead center is hardened and tempered to resist the wearing effect of the work revolving on it. The live center revolves



28.93
Figure 7-30.—Lathe centers.

with the work, and it is usually left soft. The dead center and live center must NEVER be interchanged. A dead center requires a lubricant between it and the center hole to prevent seizing and burning of the center. NOTE: There is a groove around the hardened tail center to distinguish it from the live center.

The centers fit snugly in the tapered holes of the headstock and tailstock spindles. If chips, dirt, or burrs prevent a perfect fit in the spindles, the centers will not run true.

To remove the headstock center, insert a brass rod through the spindle hole and tap the center to jar it loose; you can then pick it out with your hand. To remove the tailstock center, run the spindle back as far as it will go by turning the handwheel to the left. When the end of the tailstock screw bumps the back of the center, it will force it out of the tapered hole. (See fig. 7-6.)

For machining hollow cylinders, such as pipe, use a bull-nosed center called a PIPE CENTER. Figure 7-31 shows its construction. The taper shank A fits into the head and tail spindles in the same manner as the lathe centers. The conical disk B revolves freely on the collared end. Different size disks are supplied to accommodate various ranges of pipe sizes.

Ballbearing or nonfriction centers contain bearings that allow the point of the center to rotate with the workpiece while the shank remains stationary in the tailstock spindle. The center hole does not need a lubricant when this type of center is used.

LATHE DOGS

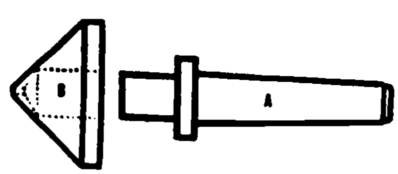
Lathe dogs are used in conjunction with a driving plate or faceplate to drive work being machined on centers because the frictional contact alone between the live center and the work is not sufficient to drive the work.

The common lathe dog shown at left in figure 7-32 is used for round work or work that has a regular section (square, hexagon, octagon). The piece to be turned is held firmly in hole A by setscrew B. The bent tail C projects through a slot or hole in the driving plate or faceplate, so that when the faceplate revolves with the spindle, it also turns the work. The clamp dog illustrated at the right in figure 7-32 may be used for rectangular or irregular shaped work. Such work is clamped between the jaws.

CENTER REST

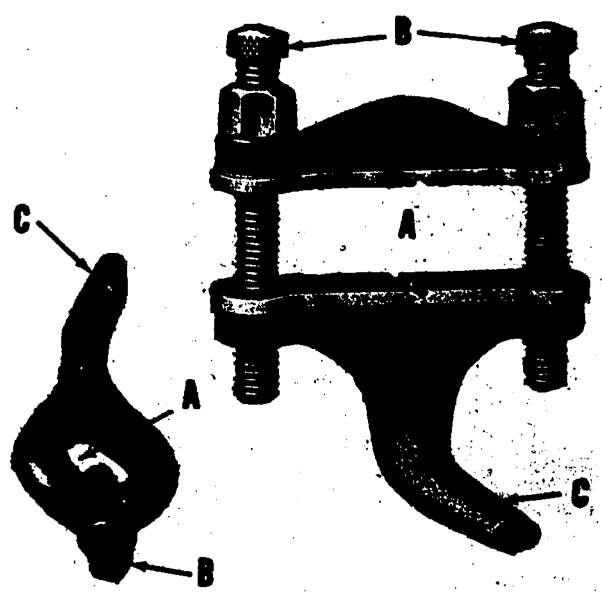
The center rest, also called the steady rest, is used for the following purposes:

1. To provide an intermediate support or rest for long slender bars or shafts being machined between centers. It prevents them



28.94

Figure 7-31.—Pipe center.



28.95X

Figure 7-32.—Lathe dogs.

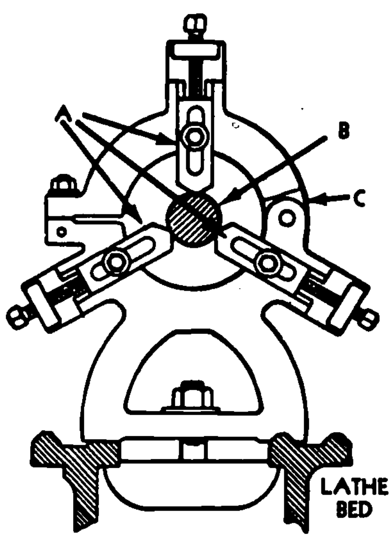
200

from springing under cut or sagging as a result of their otherwise unsupported weight.

2. To support and provide a center bearing for one end of work, such as a spindle, being bored or drilled from the end when it is too long to be supported by the chuck alone. The center rest, kept aligned by the ways, can be clamped at any desired position along the bed, as illustrated in figure 7-33. It is important that the jaws A be carefully adjusted to allow the work B to turn freely and at the same time keep it accurately centered on the axis of the lathe. The top half of the frame is hinged at C to make it easier to place in position without removing the work from the centers or changing the position of the jaws.

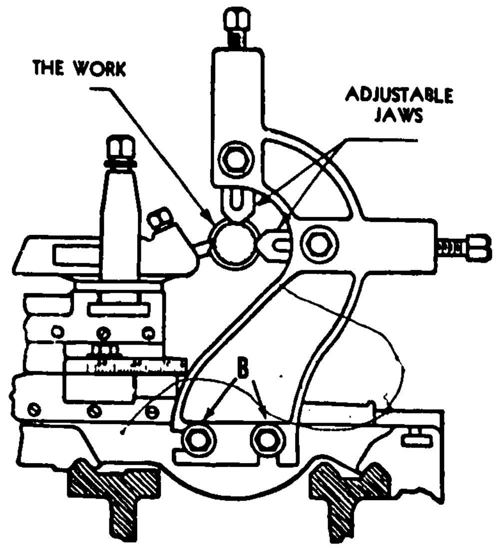
FOLLOWER REST

The follower rest is used to back up work of small diameter to keep it from springing under the stress of cutting. This rest gets its name because it follows the cutting tool along the work. As shown in figure 7-34, it is attached directly to the saddle by bolts B. The adjustable jaws bear directly on the finished diameter of the work opposite and above the cutting tool.



28.96X

Figure 7-33.—Center rest.

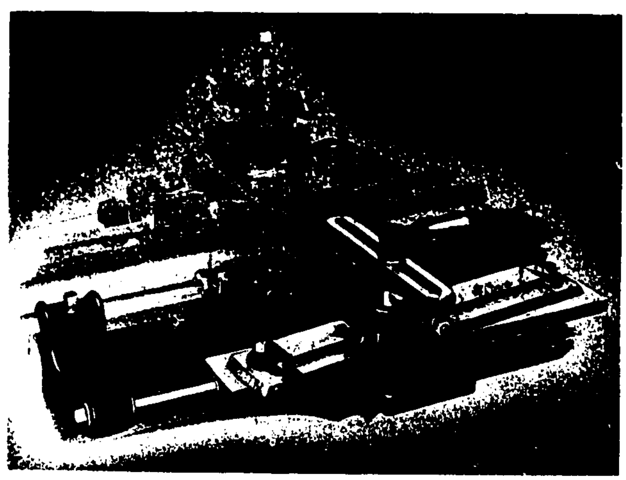


28.97X

Figure 7-34.—Follower rest.

TAPER ATTACHMENT

The taper attachment, illustrated in figure 7-35, is used for turning and boring tapers. It is bolted to the back of the carriage saddle. In operation, it is connected to the cross-slide so that it moves the cross-slide laterally as the



28.98X

Figure 7-35.—A taper attachment.

carriage moves longitudinally, thereby causing the cutting tool to move at an angle to the axis of the work to produce a taper.

The angle of the desired taper is set on the guide bar of the attachment, and the guide bar support is clamped to the lathe bed.

Since the cross-slide is connected to a shoe that slides on this guide bar, the tool follows along a line that is parallel to the guide bar and hence at an angle to the work axis corresponding to the desired taper.

The operation and application of the taper attachment will be further explained under the subject of taper turning in chapter 10.

THREAD DIAL INDICATOR

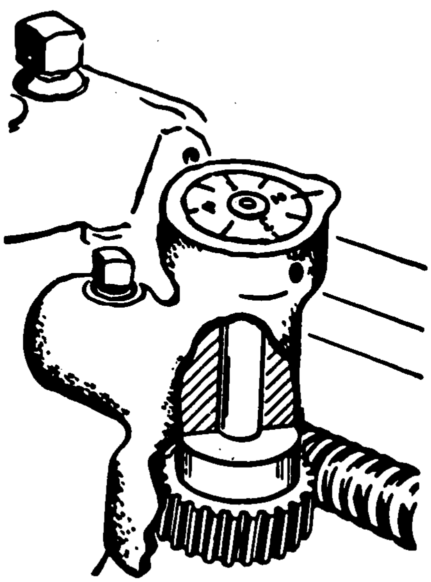
The thread dial indicator, shown in figure 7-36, lets you quickly return the carriage to the beginning of the thread to set up successive cuts. This eliminates the necessity of reversing the lathe and waiting for the carriage to follow the thread back to its beginning. The dial, which is geared to the lead screw, indicates when to clamp the half-nuts on the lead screw for the next cut.

The threading dial consists of a worm wheel which is attached to the lower end of a shaft and meshed with the lead screw. The dial is located on the upper end of the shaft. As the lead screw revolves, the dial turns. The graduations on the dial indicate points at which the half-nuts may be engaged. When the threading dial is not being used, disengage it from the lead screw to prevent unnecessary wear to the worm wheel.

CARRIAGE STOP

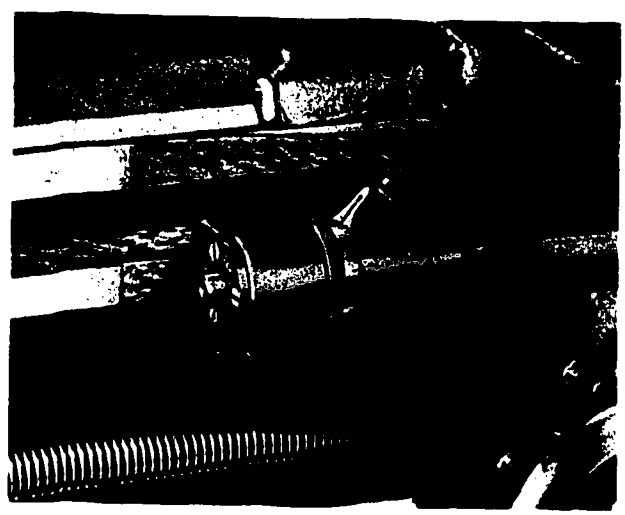
You can attach the carriage stop to the bed at any point where you want to stop the carriage. The carriage stop is used principally in turning, facing, or boring duplicate parts; it eliminates the need for repeated measurements of the same dimension. In operation, set the stop at the point where you want to stop the feed. Just before the carriage reaches this point, shut off the automatic feed and carefully run the carriage up against the stop.

Carriage stops are provided with or without micrometer adjustment. Figure 7-37 shows a micrometer carriage stop. Clamp it on the ways in the approximate position required and then adjust it to the exact setting by means of a micrometer adjustment. NOTE: Do not confuse



28.99X

Figure 7-36.—Thread dial indicator.



28.100X

Figure 7-37.—Micrometer carriage stop.

this stop with the automatic carriage stop that automatically stops the carriage by disengaging the feed or stopping the lathe.

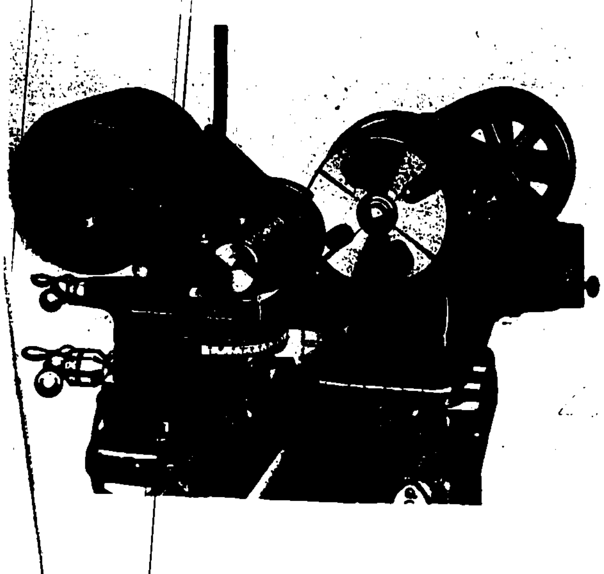
GRINDING ATTACHMENT

The grinding attachment, illustrated in figure 7-38, is a portable grinder with a base that fits on the compound rest in the same manner as the toolpost. Like the cutting tool, the grinding attachment can be fed to the work at any angle. It is used for grinding hard-faced valve disks and seats, for grinding lathe centers, and for all kinds of cylindrical grinding. For internal grinding, small wheels are used on special quills (extensions) screwed on the grinder shaft.

MILLING ATTACHMENT

The milling attachment adapts the lathe to perform milling operations on small work, such as cutting keyways, slotting screwheads, machining flats, and milling dovetails. Figure 7-39 illustrates the setup for milling a dovetail.

The milling cutter is held in an arbor driven by the lathe spindle. The work is held in a vise on the milling attachment. The milling attachment is mounted on the cross-slide and



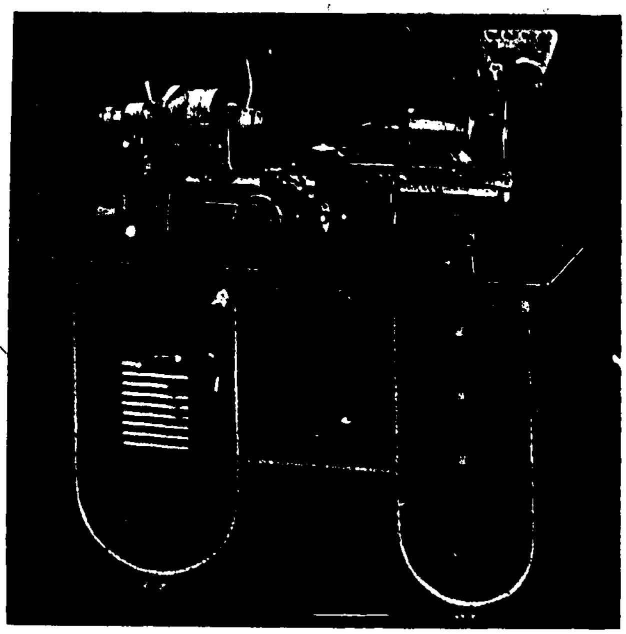
28.101X
Figure 7-38.—Grinder mounted on compound rest.

28.102X
Figure 7-39.—Milling attachment.

therefore its movement can be controlled by the longitudinal feed and crossfeed of the lathe. The depth of the cut is regulated by the longitudinal feed while the length of the cut is regulated by the crossfeed. Vertical motion is controlled by the adjusting screw at the top of the attachment. The vise can be set at any angle in a horizontal or vertical plane. A milling attachment is unnecessary in shops equipped with milling machines.

TRACING ATTACHMENTS

A tracing attachment for a lathe is useful whenever you have to make several parts that are identical in design and size. A tracer is a hydraulically actuated attachment that carries the cutting tool on a path identical to the shape and dimensions of a pattern or template of the part to be made. The major parts of the attachment are a hydraulic power unit, a tracer valve to which the stylus that follows the template is attached, a cylinder and slide assembly that holds the cutting tool and moves



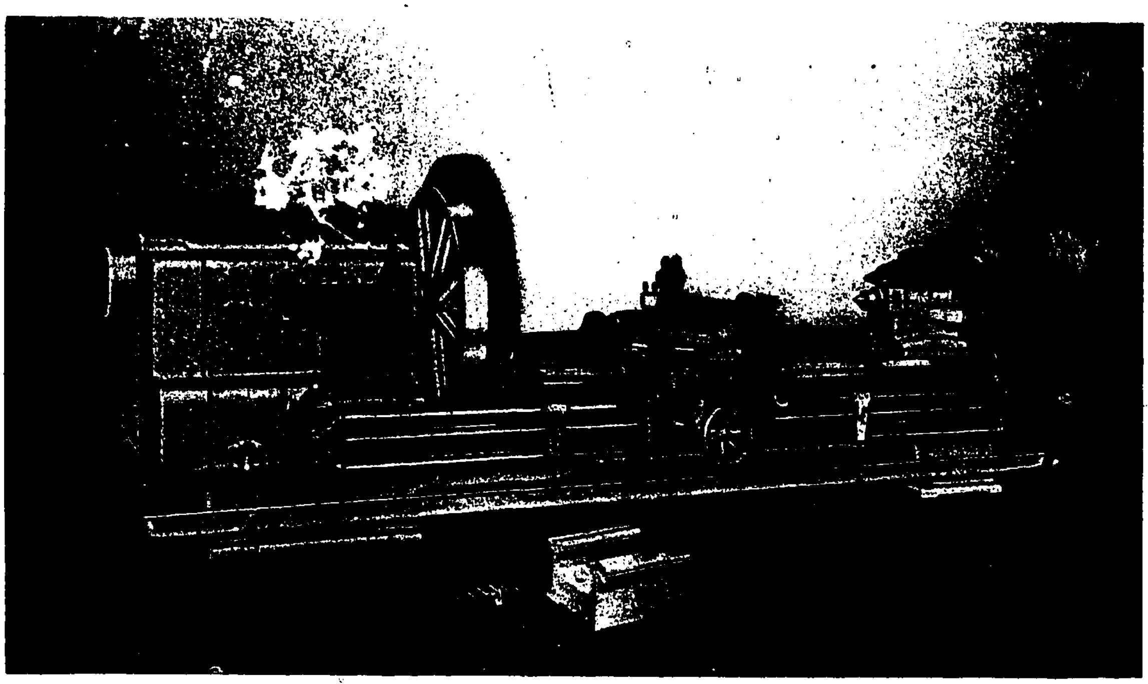
28.103X

Figure 7-40.—A bench lathe.

in or out on the command of the tracer valve hydraulic pressure output, and a template rail assembly that holds the template of the part to be made. There are several different manufacturers of tracers, and each one has a slightly different design and varying operating features. Tracers can be used for turning, facing, and boring and are capable of maintaining a dimensional tolerance equal to that of the lathe being used. Templates for the tracer can be made from either flat steel or aluminum plate or from round bar stock. It is important that the template be exactly like the finished part you require. Any scratch, dent, or mismachined dimension will be reproduced on the parts to be made.

OTHER TYPES OF LATHES

The type of engine lathe that has been described in this chapter is the general-purpose screw cutting precision lathe that is universally used in the machine shops of ships in the Navy.



28.104X

Figure 7-41.—A gap lathe.

Repair ships also carry other types. A short description of some other types follows.

TOOLROOM LATHE is the name commonly applied to an engine lathe intended for very accurate work, such as the manufacture of small precision instruments and tools.

A BENCH LATHE (fig. 7-40) is a small engine lathe mounted on a bench. Such lathes are sometimes used in the toolroom of repair ships.

The GAP (EXTENSION) LATHE shown in figure 7-41 has a removable bed piece shown on the deck in front of the lathe. This piece can be removed from the lathe bed to create a gap into which work of larger diameter may be swung. Some gap lathes are designed so that the ways can be moved longitudir. lly to create the gap.

CHAPTER 8

BASIC ENGINE LATHE OPERATIONS

In chapter 7 you became familiar with the basic design and functions of the engine lathe and the basic attachments used with the engine lathe. In this chapter, we will discuss the fundamentals of engine lathe operations.

PREOPERATIONAL PROCEDURES

As a Machinery Repairman you will be required to know and use specific procedures that must be followed both prior to and during operation of the engine lathe. First, you must be fully aware of and comply with all machine operator safety precautions. Secondly, you must be familiar with the specific type engine lathe you are going to operate.

LATHE SAFETY PRECAUTIONS

In machine operations, there is one sequence of events that you must always follow. SAFETY FIRST, ACCURACY SECOND, AND SPEED LAST. With this in mind, we will discuss the safety of lathe operations first.

1. Prepare yourself by rolling up shirt sleeves and removing watches, rings, and other jewelry that might become caught while you are operating a machine.
2. Wear safety glasses or a face shield of the approved type at all times when operating a lathe or when in the area of lathes that are in operation.
3. Be sure the work area is clear of obstructions that you might fall over or trip on.
4. Keep the deck area around your machine clear of oil or grease to prevent the possibility of anyone slipping and falling into the machine.
5. Use assistance when handling heavy or awkward parts, stock, or machine accessories.
6. Never remove chips with your bare hands; use a stick or brush. (Stop the machine while removing the chips.)
7. Prevent long chips from being caught in the chuck by using good chip control procedures on your setup.
8. Disengage the machine feed before talking to anyone.
9. Know how to stop the machine quickly if an emergency arises.
10. Be attentive, not only to the operation of your machine, but the events going on around it.
11. If it is necessary to operate a lathe while underway, be especially safety conscious. (Machines should then be operated only in relatively calm seas.)
12. Be alert to the location of the cutting tool while taking measurements or making adjustments to the machine.
13. Always observe the specific safety precautions posted for the machine you are operating.

MACHINE CHECKOUT

Before attempting the operation of any lathe, make sure you know how to run it. Read all operating instructions supplied with the machine. Ascertain the location of the various controls and how to operate them. When you are satisfied that you know how they work, start the motor, but first check to see that the spindle clutch and the power feeds are disengaged. Then become familiar with all phases of operation, as follows:

1. Shift the speed change levers into the various combinations; start and stop the spindle after each change. Get the feed of this operation.

2. With the spindle running at its slowest speed, try out the operation of the power feeds and observe their action. Take care not to run the carriage too near the limits of its travel. Learn how to reverse the direction of feeds and how to disengage them quickly. Before engaging either of the power feeds, operate the hand controls to be sure parts involved are free for running.

3. Try out the operation of engaging the lead screw for thread cutting. Remember that you must disengage the feed mechanism before the half-nuts can be closed on the lead screw.

4. Practice making changes with the QUICK CHANGE GEAR MECHANISM by referring to the thread and feed index plate on the lathe you intend to operate. Remember that changes may be made in the gear box with the lathe running slowly, but you must stop the lathe for speed changes made by shifting gears in the main gear train.

Maintenance is an important part of operational procedure for lathes and must be performed in accordance with the Navy's Planned Maintenance System (PMS). This subject is covered in detail in the Military Requirements for Petty Officers training manual. In addition to the regular planned maintenance, make it a point to oil your lathe daily where oil holes are provided. Oil the ways often, not only for lubrication but to protect their scraped surfaces. Oil the lead screw often while it is in use to preserve its accuracy. A worn lead screw lacks precision in thread cutting. Be sure the headstock is filled up to the oil level; drain out and replace the oil when it becomes dirty or gummy. If your lathe is equipped with an automatic oiling system for some parts, be sure all those parts are getting oil. Make it a habit to CHECK frequently for lubrication of all moving parts.

Do not treat your machine roughly. When you shift gears to change speed or feed, remember that you are meshing solid gear teeth; feed the gears into engagement. Disengage the clutch and stop the lathe before shifting gears.

Before engaging the longitudinal feed, be certain that the carriage clamp screw is loose and that the carriage can be moved by hand. Avoid running the carriage against the headstock or tailstock while under power feed; carriage pressure against the headstock or the tailstock puts an unnecessary strain on the lathe and may jam the gears.

Do not neglect the motor just because it may be out of sight; check its lubrication. If it does not run properly, notify the Electrician's Mate whose duty it is to care for motors. He will cooperate with you to keep it in good condition. In a machine that has a belt drive from the motor to the lathe, avoid getting oil or grease on the belt when oiling the lathe or motor.

Keep your lathe CLEAN. A clean and orderly machine is an indication of a good mechanic. Dirt and chips on the ways, the lead screw, or the crossfeed screws will cause serious wear and impair the accuracy of the machine.

Never put wrenches, files, or other tools on the ways. If you must keep tools on the bed, use a board to protect the finished surfaces of the ways.

Never use the bed or carriage as an anvil; remember that the lathe is a precision machine and nothing must be allowed to destroy its accuracy.

SETTING UP THE LATHE

Before starting a lathe machining operation, always ensure that the machine is set up for the job you are doing. If the work is mounted between centers, check the alignment of the dead center with the live center and make any changes required. Ensure that the toolholder and cutting tool are set at the proper height and angle. Check the workholding accessory to ensure that the workpiece is held securely. Use the center rest or follower rest to support long workpieces.

PREPARING THE CENTERS

The first step in preparing the centers is to see that they are accurately mounted in the headstock and tailstock spindles. The centers and the tapered holes in which they are fitted must be perfectly clean. Chips and dirt left on the contact surfaces will impair accuracy by preventing a perfect fit of the bearing surfaces. Be sure that there are no burrs in the spindle hole. If you find burrs, remove them by carefully scraping or reaming the surface with a Morse taper reamer. Burrs will produce the same inaccuracies as chips or dirt.

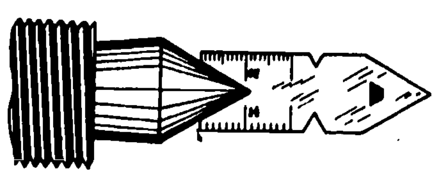
Center points must be accurately finished to an included angle of 60°. Figure 8-1 shows the method of checking the angle with a center gage. The large notch of the center gage is intended for this particular purpose. If the test shows that the point is not perfect, true the point in the lathe by taking a cut over the point with the compound rest set at 30°. The hardened tail center must be annealed before it can be machined in this manner; or it can be ground if a grinding attachment is available.

Aligning and Testing

To turn a shaft straight and true between centers, be sure the centers are in the same plane parallel to the ways of the lathe. You can align the centers by releasing the tailstock from the ways and then moving the tailstock laterally with two adjusting screws. At the rear of the tailstock are two zero lines, and the centers are approximately aligned when these lines coincide. To check the approximate alignment, move the tailstock up until the centers almost touch and observe their relative positions as shown in figure 8-2. To produce very accurate work, especially if it is long, use the following procedure to determine and correct errors in alignment not otherwise detected.

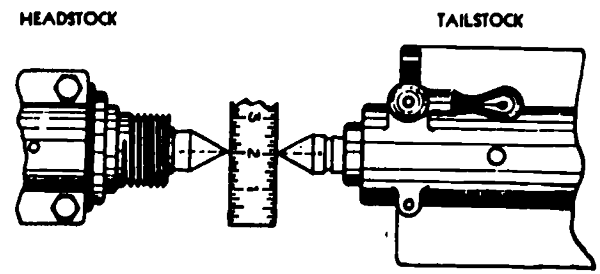
Mount the work to be turned, or a piece of stock of similar length, on the centers. With a turning tool in the toolpost, take a small cut to a depth of a few thousandths of an inch at the headstock end of the work. Then remove the work from the centers to allow the carriage to be run back to the tailstock without withdrawing the tool. Do not touch the tool setting. Replace the work in the centers, and with the tool set at the previous depth take another cut coming in from the tailstock end. Compare the diameters of these cuts with a micrometer. If the diameters are exactly the same, the centers are in perfect alignment. If they are different, the tailstock must be adjusted in the direction required by means of the set-over adjusting screws. Repeat the above test and adjustment until a cut at each end produces equal diameter.

You can also check positive alignment of centers by placing a test bar between centers and bringing both ends of the bar to a zero reading as indicated by a dial indicator clamped in the



28.105X

Figure 8-1.—Checking center point with a center gage.



28.106X

Figure 8-2.—Aligning lathe centers.

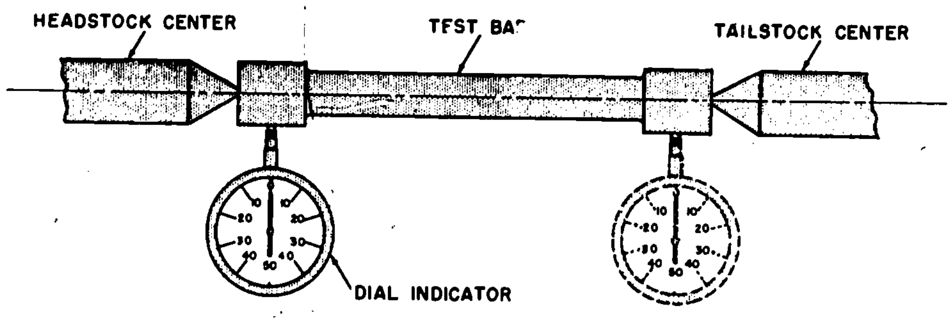


Figure 8-3.—Alignment of lathe centers with a dial indicator.

28.107X

toolpost (fig. 8-3). The tailstock must be clamped to the ways and the test bar must be properly adjusted between centers so there is no end play when you take the indicator readings.

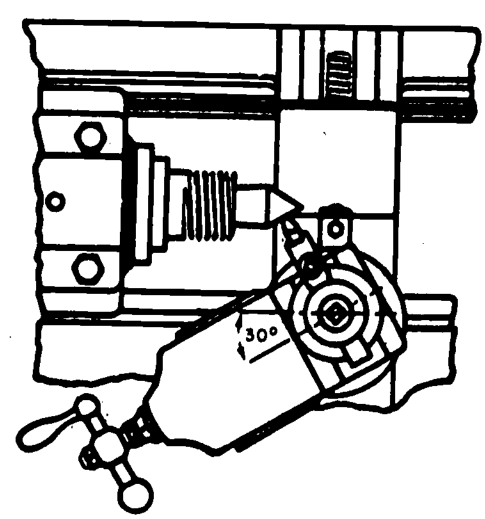
Another method you can use for positive alignment of lathe centers is to take a light cut over the work held between centers. Then measure the work at each end with a micrometer. If the readings differ, adjust the tailstock accordingly. Repeat the procedure until the centers are aligned.

Truing and Grinding

To machine or true a lathe center, remove the faceplate from the spindle. Then insert the live center into the spindle and set the compound rest at an angle of 30° with the axis of the spindle, as shown in figure 8-4. Place a round-nose tool in the toolpost and set the cutting edge of the tool at the exact center point of the lathe center. Machine a light cut on the center point and test the point with a center gage. All lathe centers, regardless of size, are finished to an included angle of 60°.

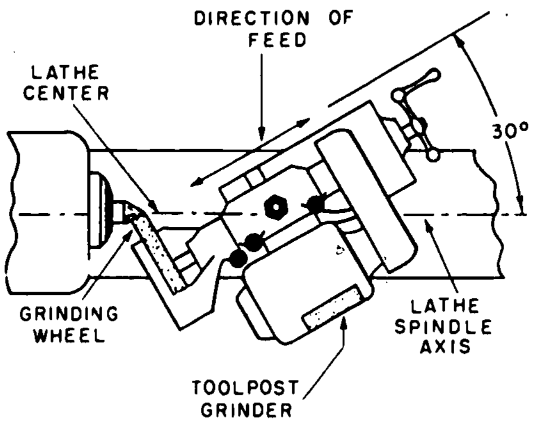
If the tailstock spindle lathe center is to be trued, anneal it and machine it in the headstock spindle, following the same operations described for truing a live center; then remove, harden, and temper the spindle. It is now ready for use in the tailstock.

If a toolpost grinder is available, the hardened center may be trued by grinding without annealing. As in machining, the first step after placing the center in the headstock spindle is to set the compound rest over to 30° with the axis of the lathe. Second, mount a toolpost grinder or grinding attachment on the lathe as shown in figure 8-5. Third, cover the exposed ways of the lathe with cloth or paper to keep the grinding grit out of the bearing surfaces of the bed and cross-slides. Fourth, put the headstock in gear to give approximately 200 rpm to the spindle and take a light cut over the center point, feeding the wheel across the point with the compound rest feed handle. Continue to feed the wheel back and forth until it is cutting evenly all around and the entire length

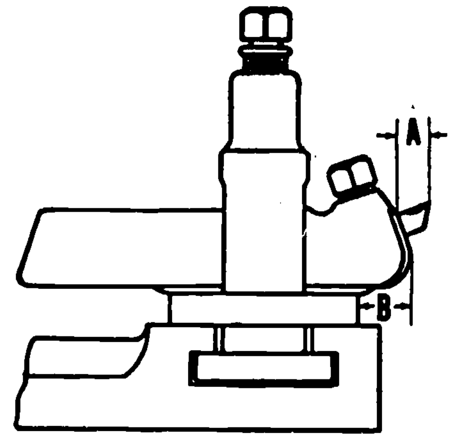


28.108X

Figure 8-4.—Machining a lathe center.



28.341
Figure 8-5.–Grinding a lathe center.



28.110X
Figure 8-6.–Tool overhang.

of the center point. Then check the angle with a center gage. Reset the compound rest if necessary and continue grinding until the center fits the center gage exactly. Observe the accuracy of the fit by placing a light beneath the center and looking for light between the center point surface and the edge of the center point gage.

Additional information on the operation of the toolpost grinder is provided later in this chapter.

SETTING THE TOOLHOLDER AND CUTTING TOOL

The first requirement for setting the tool is to have it rigid. Be sure the tool sets squarely in the toolpost and that the setscrew is tight. Reduce overhang as much as possible to prevent tool springing during cutting. If the tool has too much spring, the point of the tool will catch in the work, causing chatter and damaging both the tool and the work. The relative distances of A and B in figure 8-6 show the correct overhang for the tool bit and the holder. When a quick change type toolholder is used, tool overhang should not exceed twice the width of the cutting tool or of the shank when you use a carbide insert type cutting tool.

The point of the tool must be correctly positioned on the work. When you are using a high-speed steel cutting tool to straight turn steel, cast iron and other relatively hard metals, set the point about 5° above center. A rule measurement of approximately 3/64 inch times the diameter of the workpiece equals 5°. Aluminum being cut with a high-speed steel cutting tool should be set slightly more than 5° above center, while copper, brass and other soft metals require the cutting tool point to be set exactly on center. The point of cast alloy (stellite, etc.), carbide, and ceramic cutting tools should be placed exactly on center regardless of the material being cut. The tool should be placed on center for threading, turning tapers, parting (cutting-off) or boring.

The height of the tool in the toolholder illustrated in figure 8-6 can be adjusted by moving the half-moon wedge beneath the toolholder in or out as required. The quick-change type toolholder usually has an adjusting screw to stop the tool at the correct height. Some square turret type toolholders require a shim beneath the tool to adjust the height.

There are several methods you can use to set a tool on center. A dead center placed in the tailstock can be used to align the point of the tool with the point of the center. The tailstock spindle on many lathes has a line on the side that represents the center. Another method is to place a 6-inch rule against the workpiece in a vertical position and move the cross-slide in until the tool lightly touches the rule and holds it in place. Look at the rule from the side to determine if the height of the tool is correct. The rule will be straight up and down when the tool is exactly on center and will be at an angle when the tool is either high or low.

METHODS OF HOLDING THE WORK

Accurate work cannot be performed if work is improperly mounted. Requirements for proper mounting are:

1. The work centerline must be accurately centered along the axis of the lathe spindle.

2. The work must be rigidly held while being turned.

3. The work must not be sprung out of shape by the holding device.

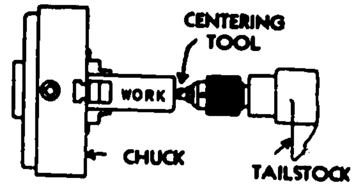
4. The work must be adequately supported against any sagging caused by its own weight and against springing caused by the action of the cutting tool.

There are four general methods of holding work in the lathe: (1) between centers, (2) on a mandrel, (3) in a chuck, and (4) on a faceplate. Work may also be clamped to the carriage for boring and milling; the boring bar or milling cutter is held and driven by the headstock spindle.

Other methods of holding work to suit special conditions are: (1) one end on the live center or in a chuck with the other end supported in a center rest, and (2) one end in a chuck with the other end on the dead center.

HOLDING WORK BETWEEN CENTERS

To machine a workpiece between centers, drill center holes in each end to receive the lathe centers. Secure a lathe dog to the workpiece and then mount the work between the live and dead centers of the lathe.



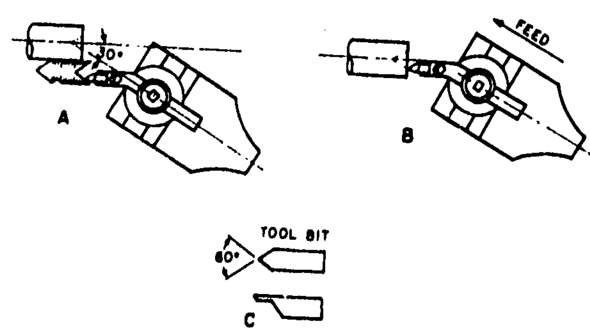
28.111

Figure 8-7.—Drilling center hole.

Centering the Work

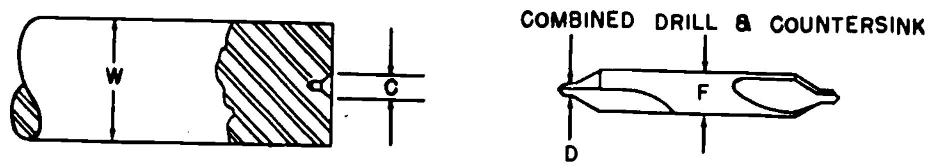
To center drill round stock such as drill rod or cold-rolled steel, secure the work to the head spindle in a universal chuck or a draw-in collet chuck. If the work is too long and too large to be passed through the spindle, use a center rest to support one end. It is good shop practice to first take a light finishing cut across the face of the end of the stock to be center drilled. This will ensure a smooth and even surface and will reduce the danger of "wandering" or breaking of the center drill. The centering tool is held in a drill chuck in the tailstock spindle and fed to the work by the tailstock handwheel (fig. 8-7).

If a piece must be centered very accurately, the tapered center hole should be bored after center drilling to correct any run-out of the drill. You can do this by grinding a tool bit to a center gage at a 60° angle. Then, with the toolholder held in the toolpost, set the compound rest at 30° with the line of center as shown in figure 8-8. Set the tool exactly on the



28.112

Figure 8-8.—Boring center hole.



NO. OF COMB. DRILL AND COUNTERSINK	DIA. OF WORK (W)	LARGE DIAMETER OF COUNTERSUNK HOLE (C)	DIA. OF DRILL (D)	DIA. OF BODY (F)
1	3/16" TO 5/16"	1/8"	1/16"	13/64"
2	3/8" TO 1"	1/16"	3/32"	3/16"
3	1 1/4" TO 2"	1/4"	1/8"	5/16"
4	2 1/4" TO 4"	5/16"	5/32"	7/16"

28.113X

Figure 8-9.—Correct size of center holes.

center for height and adjust to the proper angle with the center gage as shown at A. Feed the tool as shown at B to correct any run-out of the center. The tool bit should be relieved under the cutting edge as shown at C to prevent the tool from dragging or rubbing in the hole.

For center drilling a workpiece, the combined drill and countersink is the most practical tool. The combined drills and countersinks vary in size and the drill points also vary. Sometimes a drill point on one end will be 1/8 inch in diameter and the drill point on the opposite end will be 3/16 inch in diameter. The angle of the center drill is always 60° so that the countersunk hole will fit the angle of the lathe center point.

If a center drill is not available, you may center the work with a small twist drill. Let the drill enter the work a sufficient depth on each end; then follow with a countersink which has a 60° point.

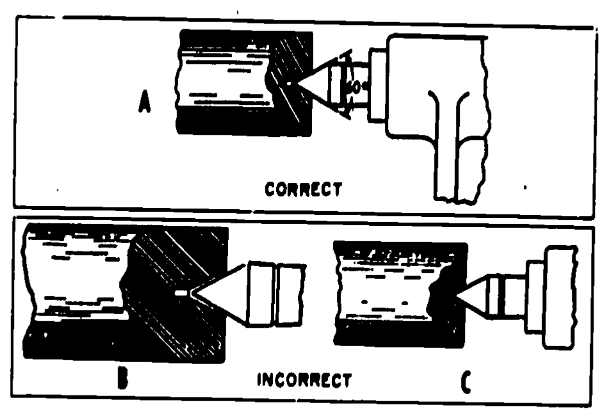
The drawing and tabulation in figure 8-9 shows the correct size of the countersunk center hole for the diameter of the work.

In center drilling, use a drop or two of oil on the drill. Feed the drill slowly and carefully so as not to break the tip. Use extreme care when the work is heavy, because it is then more difficult to "feel" the proper feed of the work on the center drill.

If the center drill breaks in countersinking and part of the broken drill remains in the work, you must remove the broken part. Sometimes it can be driven out by a chisel or jarred loose, but it may stick so hard you cannot easily remove it. If so, anneal the broken part of the drill and drill it out.

The importance of having proper center holes in the work and a correct angle on the point of the lathe centers cannot be overemphasized. To do an accurate job between centers on the lathe, you must countersink holes of the proper size and depth, and be sure the points of the lathe centers are true and accurate.

Figure 8-10 shows correct and incorrect countersinking for work to be machined on



28.114X

Figure 8-10.—Examples of center holes.

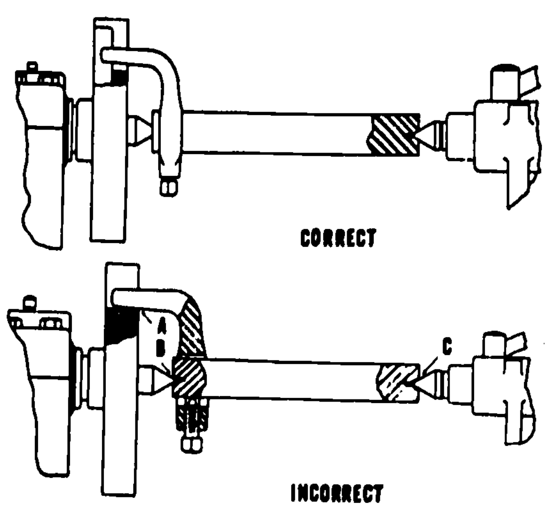
centers. In example A, the correctly countersunk hole is deep enough so that the point of the lathe centers does not come in contact with the bottom of the hole.

In example B of figure 8-10, the countersunk hole is too deep, causing only the outer edge of the work to rest on the lathe center. Work cannot be machined on centers countersunk in this manner.

Example C shows a piece of work that has been countersunk with a tool of an improper angle. This work rests on the point of the lathe center only. It is evident that this work will soon destroy the end of the lathe center, thus making it impossible to do an accurate job.

Mounting the Work

Figure 8-11 shows correct and incorrect methods of mounting work between centers. In the correct example, the driving dog is attached to the work and rigidly held by the setscrew. The tail of the dog rests in the slot of the drive plate and extends beyond the base of the slot so that the work rests firmly on both the headstock center and tailstock center.



28.115X
Figure 8-11.—Examples of work mounted between centers.

In the incorrect example, the tail of the dog rests on the bottom of the slot on the faceplate at A, thereby pulling the work away from the center points, as shown at B and C, and causing the work to revolve eccentrically.

When you mount work between centers for machining, th^re should be no end play between the work and the dead center. However, if the work is held too tightly by the tail center, when the work begins revolving it will heat the center point and destroy both the center and the work. To prevent overheating, lubricate the tail center with a heavy oil or a lubricant specially made for this purpose.

HOLDING WORK ON A MANDREL

Many parts, such, as bushings, gears, collars, and pulleys, require all the finished external surfaces to run true with the hole which extends through them. That is, the outside diameter must be true with the inside diameter or bore.

General practice is to finish the hole to a standard size, within the limit of the accuracy desired. Thus, a 3/4-inch standard hole will have a finished dimension of from 0.7505 to 0.7495 inch, or a tolerance of one-half thousandth of an inch above or below the true standard size of exactly 0.750 inch. First, drill or bore the hole to within a few thousandths of an inch of the finished size; then remove the remainder of the material with a machine reamer.

Press the piece on a mandrel tightly enough so the work will not slip while it is machined and clamp a dog on the mandrel, which is mounted between centers. Since the mandrel surface runs true with respect to the lathe axis, the turned surfaces of the work on the mandrel will be true with respect to the hole in the piece.

A mandrel is simply a round piece of steel of convenient length which has been centered and turned true with the centers. Commercial mandrels are made of tool steel, hardened and ground with a slight taper (usually 0.0005 inch per inch). On sizes up to 1 inch the small end is usually one-half thousandth of an inch under the standard size of the mandrel, while on larger sizes this dimension is usually one thousandth of an inch under standard. This taper allows the standard hole in the work to vary according to the usual shop practice, and still provides the necessary fit to drive the work when the mandrel is pressed into the hole. However, the taper is not great enough to distort the hole in the work. The countersunk centers of the mandrel are lapped for accuracy, while the ends are turned smaller than the body of the mandrel and are provided with flats, which give a driving surface for the lathe dog.

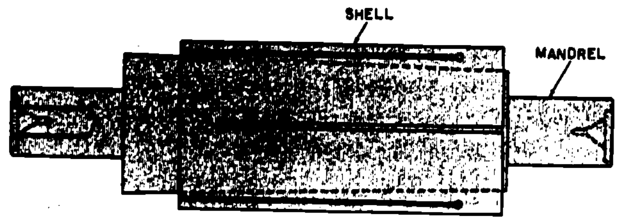
The size of the mandrel is always marked on the large end to avoid error and for convenience in placing work on it. The work is driven or pressed on from the small end and removed the same way.

When the hole in the work is not standard size, or if no standard mandrel is available, make a soft mandrel to fit the particular piece to be machined.

Use a few drops of oil to lubricate the surface of the mandrel before pressing it into the work, because clean, metallic surfaces gall or stick when pressed together. If you do not use lubricant, the mandrel cannot be driven out without ruining the work.

Whenever you machine work on a mandrel, be sure that the lathe centers are true and accurately aligned; otherwise, the finished turned surface will not be true. Before turning accurate work, test the mandrel on centers before placing any work on it. The best test for run-out is made with a dial indicator. Mount the indicator on the toolpost so that the point of the indicator just touches the mandrel. As the mandrel is turned slowly between centers, any run-out will be registered on the indicator dial.

If run-out is indicated and you cannot correct it by adjusting the tailstock, the mandrel itself is at fault (assuming that the lathe centers are true) and cannot be used. The countersunk holes may have been damaged, or the mandrel may have been bent by careless handling. Be sure you always protect the ends of the mandrel when pressing or driving it into the work. A piece of work mounted on a mandrel must have a tighter press fit to the mandrel for roughing cuts than for finishing cuts. Thick-walled work can be left on the mandrel for the finishing cut but thin-walled work should be removed from the mandrel after the roughing cut and lightly reloaded on the mandrel before the finish cut is taken.



28.116

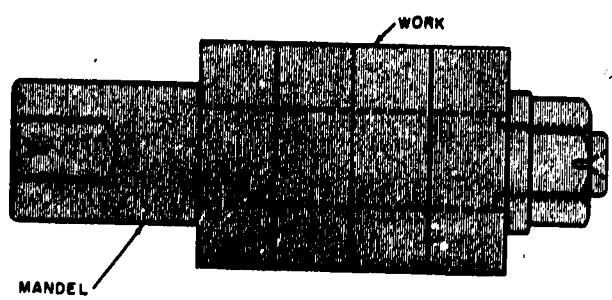
Figure 8-12.—A split-shell expansion mandrel.

In addition to the standard lathe mandrel just described, there are expansion mandrels, gang mandrels, and eccentric mandrels.

An EXPANSION mandrel is used to hold work that is reamed or bored to nonstandard size. Figure 8-12 shows an expansion mandrel composed of two parts: a tapered pin that has a taper of approximately 1/16 inch for each inch of length and an outer split shell that is tapered to fit the pin. The split shell is placed in the work and the tapered pin is forced into the shell, causing it to expand until it properly holds the work.

A GANG mandrel (fig. 8-13) is used for holding several duplicate pieces such as gear blanks. The pieces are held tightly against a shoulder by a nut at the tailstock end.

An ECCENTRIC mandrel has two sets of countersunk holes, one pair of which is



28.117

Figure 8-13.—Gang mandrel.

off-center an amount equal to the eccentricity of the work to be machined. Figure 8-14 illustrates its application: A is to be machined concentric with the hole in the work, while B is to be machined eccentric to it.

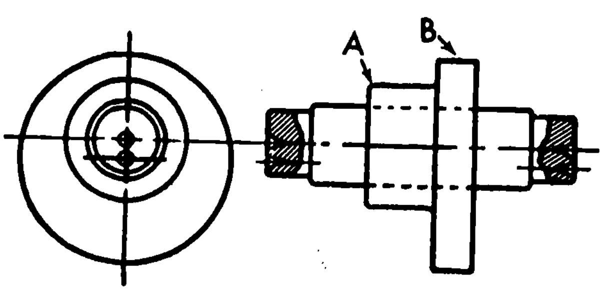
HOLDING WORK IN CHUCKS

The independent chuck and universal chuck are used more often than other workholding devices in lathe operations. A universal chuck is used for holding relatively true cylindrical work when accurate concentricity of the machined surface and holding power of the chuck are secondary to the time required to do the job. An independent chuck is used when the work is irregular in shape, must be accurately centered, or must be held securely for heavy feeds and depth of cut.

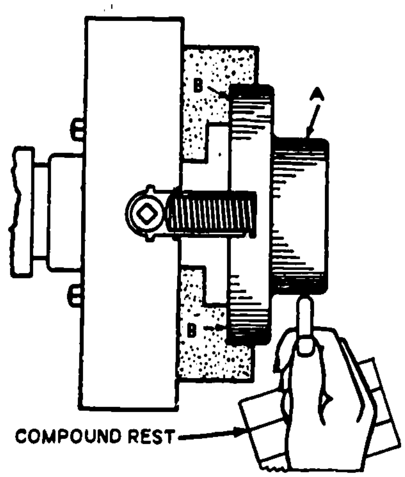
Four-Jaw Independent Chuck

Figure 8-15 shows a rough casting mounted in a four-jaw independent lathe chuck on the spindle of the lathe. Before truing the work, determine which part you wish to have turn true. To mount a rough casting in the chuck, proceed as follows:

1. Adjust the chuck jaws to receive the casting. Each jaw should be concentric with the ring marks indicated on the face of the chuck. If there are no ring marks, set the jaws equally distant from the circumference of the chuck body.



28.118
Figure 8-14. –Work on an eccentric mandrel.



28.119
Figure 8-15.—Work mounted in a 4-jaw independent chuck.

2. Fasten the work in the chuck by turning the adjusting screw on jaw No. 1 and jaw No. 3, a pair of jaws which are opposite each other. Next tighten jaws No. 2 and No. 4 (opposite each other).

3. At this stage the work should be held in the jaws just tightly enough so it will not fall out of the chuck while being trued.

4. Revolve the spindle slowly, and with a piece of chalk mark the high spot (A in fig. 8-15) on the work while it is revolving. Steady your hand on the toolpost while holding the chalk.

5. Stop the spindle. Locate the high spot on the work and adjust the jaws in the proper direction to true the work by releasing the jaw opposite the chalk mark and tightening the one nearest the mark.

6. Sometimes the high spot on the work will be located between adjacent jaws. When it is, loosen the two opposite jaws and tighten the jaws adjacent to the high spot.

7. When the work is running true in the chuck, tighten the jaws gradually, working the jaws in pairs as described previously, until all four jaws are clamping the work tightly. Be sure that the back of the work rests flat against the inside face of the chuck, or against the faces of the jaw stops (B in figure 8-15).

Use the same procedure to clamp semi-finished or finished pieces in the chuck, except center these pieces more accurately in the chuck. If the run-out tolerance is very small, use a dial indicator to determine the run-out.

Figure 8-16 illustrates the use of a dial test indicator in centering work that has a hole bored in its center. As the work is revolved, the high spot is indicated on the dial of the instrument to a thousandth of an inch. The jaws of the chuck are adjusted on the work until the indicator hand registers no deviation as the work is revolved.

When the work consists of a number of duplicate parts that are to be tightened in the chuck, release two adjacent jaws and remove the work. Place another piece in the chuck and retighten the two jaws just released.

Each jaw of a lathe chuck, whether an independent or a universal chuck, has a number stamped on it to correspond with a similar number on the chuck. When you remove a chuck jaw for any reason, you should always put it back into the proper slot.

When the work to be chucked is frail or light, tighten the jaw carefully so that the work will not bend, break, or spring.

To mount rings or cylindrical disks on a chuck, expand the chuck jaws against the inside of the workpiece. (See fig. 8-17.)

Regardless of how you mount the workpiece, NEVER leave the chuck wrench in the chuck while the chuck is on the lathe spindle. If the lathe should be started, the wrench could fly off the chuck and injure you or a bystander.

Three-Jaw Universal Chuck

A three-jaw universal, or scroll, chuck allows all jaws to move together or apart in unison. A universal chuck will center almost exactly at the first clamping, but after a period of use it is not uncommon to find inaccuracies of from .002 to .010 inch in centering the work, and consequently the run-out of the work must be corrected. Sometimes you can make the correction by inserting a piece of paper or thin shim stock between the jaw and the work on the HIGH SIDE.

When chucking thin sections, be careful not to clamp the work too tightly, since the diameter of the piece will be machined while the

28.120X
Figure 8-16.—Centering work with a dial indicator.

28.121
Figure 8-17.—Work held from inside by a 4-jaw independent chuck.

piece is distorted. When the pressure of the jaws is released after the cut, there will be as many high spots as there are jaws, and the turned surface will not be true.

Draw-In Collet Chuck

A draw-in collet chuck is used for very fine accurate work of small diameter. Long work can be passed through the follow drawbar, and short work can be placed directly into the collet from the front. Tighten the collet on the work by rotating the drawbar handwheel to the right. This draws the collet into the tapered closing sleeve; the opposite operation releases the collet.

You will get the most accurate results when the diameter of the work is the same as the dimension stamped on the collet. The actual diameter of the work may vary from the collet dimension by ±0.001 inch. However, if the work diameter varies more than this, the accuracy of the finished work will be affected. Most draw-in collet chuck sets are sized in 1/64-inch increments to allow you to select a collet within the required tolerances.

Rubber Flex Collet Chuck

A rubber flex collet chuck is basically the same as the draw-in type collet, except that the size of the stock held is not as critical. The rubber collets are graduated in 1/16-inch steps and will tighten down with accuracy on any size within the 1/16-inch range.

CARE OF CHUCKS

To preserve a chuck's accuracy, handle it carefully and keep it clean. Never force a chuck jaw by using a pipe as an extension on the chuck wrench.

Before mounting a chuck, remove the live center and fill the hole with a rag to prevent chips and dirt from getting into the taper hole of the spindle.

Clean and oil the threads of the chuck and the spindle nose. Dirt or chips on the threads will not allow the chuck to properly seat against the spindle shoulder and will prevent the chuck from running true. Screw the chuck carefully onto the spindle and the threads and make it difficult to remove the chuck. Never use mechanical power to install a chuck, but rotate the spindle with your left hand while you support the chuck in the hollow of your right arm.

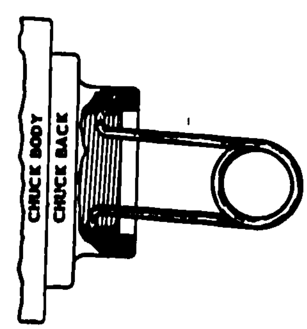
To remove a small chuck, place a chuck wrench in the square hole in one of the jaws and strike a smart blow on the wrench handle with your hand. To remove a heavy chuck, rotate it against a block of wood held between a jaw and the lathe bed. When mounting or removing a heavy chuck, lay a board across the bed ways to protect them and to help support the chuck as you put it on or take it off. Most larger chucks are drilled and taped to accept a padeye for lifting with a chainfall.

The procedures for mounting and removing faceplates are the same as for mounting and removing chucks.

Figure 8-18 shows a simple device made of brass wire for cleaning the threads of a chuck or faceplate.

HOLDING WORK ON A FACEPLATE

A faceplate is used for mounting work that cannot be chucked or turned between centers because of its peculiar shape. A faceplate is also used when holes are to be accurately machined



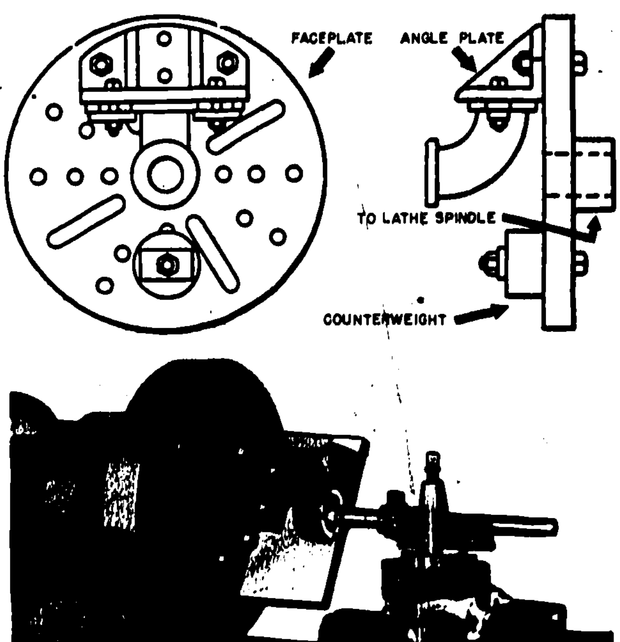
28.122X

Figure 8-18.—Tool for cleaning thread of a chuck or faceplate.

28.123X
Figure 8-19.–Eccentric machining of work mounted on a faceplate.

in flat work, as in figure 8-19, or when large and irregularly shaped work is to be faced on the lathe.

Work is secured to the faceplate by bolts, clamps, or any suitable clamping means. The holes and slots in the faceplate are used to anchor the holding bolts. Angle plates may be used to locate the work at the desired angle, as shown in figure 8-20. (Note the counterweight added for balance.)



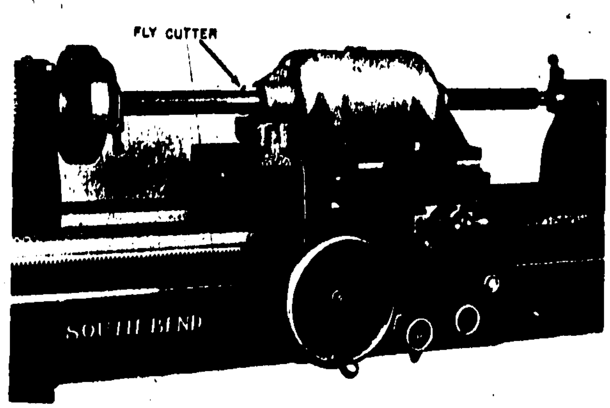
28.124X
Figure 8-20.–Work clamped to an angle plate.

To mount work accurately on a faceplate, the surface of the work in contact with the faceplate must be accurate. Check the accuracy with a dial indicator. If you find run-out, reface the surface of the work that is in contact with the faceplate. It is good practice to place a piece of paper between the work and the faceplate to keep the work from slipping.

Before securely clamping the work, move it about on the surface of the faceplate until the point to be machined is centered accurately over the axis of the lathe. Suppose you wish to bore a hole, the center of which has been laid out and marked with a prick punch. First, clamp the work to the approximate position on the faceplate. Then slide the tailstock up to where the dead center just touches the work. Note, the dead center should have a sharp, true point. Now revolve the work slowly and, if the work is off center, the point of the dead center will scribe a circle on the work. If the work is on center, the point of the dead center will coincide with the prick punch mark.

HOLDING WORK ON THE CARRIAGE

If a piece of work is too large or bulky to swing conveniently in a chuck or on a faceplate, you can bolt it to the carriage or the cross-slide and machine it with a cutter mounted on the spindle. Figure 8-21 shows a piece of work being machined by a fly cutter mounted in a boring bar which is held between centers and driven by a lathe dog.



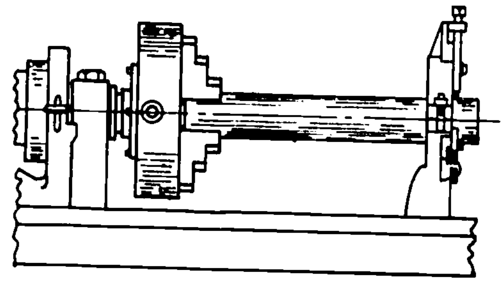
28.128X
Figure 8-21.–Work mounted on a carriage for boring.

USING THE CENTER REST AND FOLLOWER REST

Long slender work often requires support between its ends while it is turned; otherwise the work would spring away from the tool and chatter. The center rest is used to support such work so it can be accurately turned with a faster feed and cutting speed than would be possible without the center rest. (See fig. 8-22.)

The center rest should be placed where it will give the greatest support to the piece to be turned. This is usually at about the middle of its length.

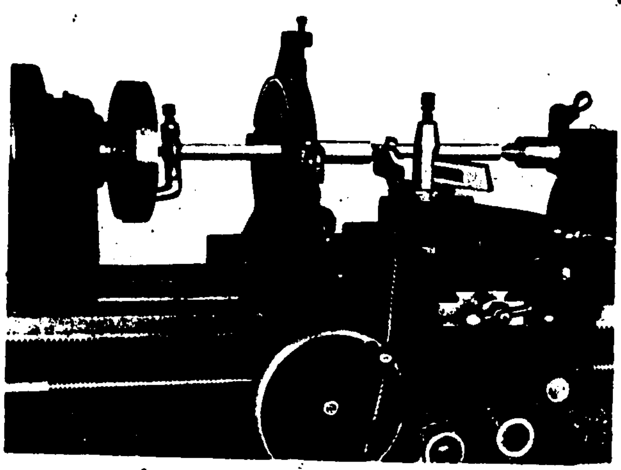
Ensure that the center point between the jaws of the center rest coincides exactly with the axis of the lathe spindle. To do this, place a short piece of stock in a chuck and machine it to the diameter of the workpiece to be supported. Without removing the stock from the chuck, clamp the center rest on the ways of the lathe and adjust the jaws to the machined surface. Without changing the jaw settings, slide the center rest into position to support the workpiece. Remove the stock used for setting the center rest and set the workpiece in place. Use a dial indicator to true the workpiece at the chuck. Figure 8-23 shows how a chuck and center rest are used to machine the end of a workpiece.



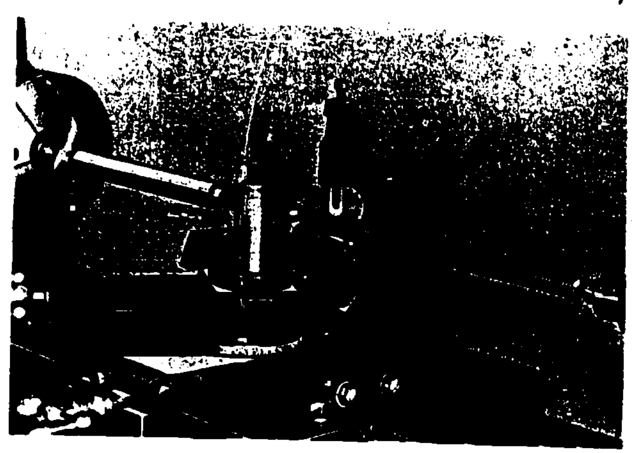
28.126X
Figure 8-23.—Work mounted in a chuck and center rest.

The follower rest differs from the center rest in that it moves with the carriage and provides support against the forces of the cut. To use the tool, turn a "spot" of the desired finish diameter and about 5/8 to 3/4 inch wide on the workpiece. Then, adjust the jaws of the follower rest against the area you just machined. The follower rest will move with the cutting tool and support the point of the work that is being machined.

The follower rest (fig. 8-24) is indispensable for chasing threads on long screws, as it allows the cutting of a screw with a uniform pitch



28.125X
Figure 8-22.—Use of a center rest to support work between centers.



28.127X
Figure 8-24.—Follower rest supporting screw while thread is being cut.

diameter. Without the follower rest, the screw would be inaccurate because it would spring away from the tool.

Use a sufficient amount of grease, oil or other available lubricant on the jaws of the center rest and follower rest to prevent "seizing" and scoring the workpiece. Check the jaws frequently to see that they do not become hot. The jaws may expand slightly if they get hot thus pushing the work out of alignment (when the follower rest is used) or binding (when the center rest is used).

MACHINING OPERATIONS

Up to this point, you have studied the preliminary steps leading up to the performance of machine work in the lathe. You have learned how to mount the work and the tool, and which tools are used for various purposes. The next step is to learn how to use the lathe to turn, bore, and face the work to the desired form or shape.

TURNING is the machining of the outside surface of a cylinder.

BORING is the machining of the inside surface of a cylinder.

FACING is the machining of flat surfaces.

Remember that accuracy is the prime requisite of a good machine job; so before you start, be sure that the centers are true and properly aligned, that the work is mounted properly, and that the cutting tools are correctly ground and sharpened.

PLANNING THE JOB

It is important that you study the blueprint of the part to be manufactured before you begin machining. Check over the dimensions and note the points or surfaces from which they are laid out. Plan the steps of your work in advance to determine the best way to proceed. Check the overall dimensions and be sure that the stock you intend to use is large enough for the job. For example, small design features, such as collars on pump shafts or valve stems, will require that you use stock of much larger diameter than that required for the main features of the workpiece.

CUTTING SPEEDS AND FEEDS

Cutting speed is the rate at which the surface of the work passes the point of the cutting tool. It is expressed in feet per minute (fpm).

To find the cutting speed, multiply the diameter of the work (DIA) in inches times 3.1416 times the number of revolutions per minute (rpm) and divide by 12.

$$CS = \frac{DIA \times 3.1416 \times rpm}{12}$$

The result is the peripheral or cutting speed in feet per minute. For example, a 2-inch diameter part turning at 100 rpm will produce a cutting speed of

$$\frac{2 \times 3.1416 \times 100}{12} = 52.36 \text{ fpm}$$

If you have selected a recommended cutting speed from a chart for a specific type of metal, you will need to figure what rpm is required to obtain the recommended cutting speed, or feet per minute. This may be done by the following formula:

$$rpm = \frac{CS \times 12}{DIA \times 3.1416}$$

Table 8-1 gives the approximate recommended cutting speeds for various metals, using a high-speed steel tool bit. To obtain an approximate cutting speed for the other types of cutting tool materials multiply the cutting speeds recommended in table 8-1 and other charts, which you will find in different handbooks, by the following factors:

Carbon steel tools – 50% of HSS, multiply by 0.5

Cast alloy tools – 160% of HSS, multiply by 1.6

Carbide tools – 200% to 400% of HSS, multiply by 2.0 to 4.0

Ceramic tools – 400% to 1600% of HSS, multiply by 4.0 to 16.0

Table 8-1.—Cutting Speeds Using High-speed Steel Tool Bits

TYPE OF MATERIAL	Cutting Speed (fpm)
Low carbon steel	40-140
Medium carbon steel	70-120
High carbon steel	65-100
Stainless steel, Cl 302, 304	60
Stainless steel, Cl 310, 316	70
Stainless steel, Cl 410	100
Stainless steel, Cl 416	140
Stainless steel, Cl 17-4, pH	50
Alloy steel, SAE 4130, 4140	70
Alloy steel, SAE 4030	90
Gray cast iron	20-90
Aluminum alloys	600-750
Brass	200-350
Bronze	100-110
Nickel alloy, Monel 400	40-60
Nickel alloy, Monel K500	30-60
Nickel alloy, Inconel	5-10
Titanium alloy	20-60

FEED is the amount the tool advances in each revolution of the work. It is usually expressed in thousandths of an inch per revolution of the spindle. The index plate on the quick-change gear box indicates the setup for obtaining the feed desired. The amount of feed to use is best determined from experience.

Cutting speeds and tool feeds are determined by various considerations: the hardness and toughness of the metal being cut; the quality, shape, and sharpness of the cutting tool; the depth of the cut; the tendency of the work to spring away from the tool; and the rigidity and power of the lathe. Since conditions vary, it is good practice to find out what the tool and work will stand, and then select the most practicable and efficient speed and feed consistent with the finish desired.

If the cutting speed is too slow, the job takes longer than necessary and the work produced is often unsatisfactory because of a poor finish. On the other hand, if the speed is too great the tool edge will dull quickly and will require frequent regrinding. The cutting speeds possible are greatly affected by the use of a suitable cutting lubricant. For example, steel that can be rough turned dry at 60 rpm can be turned at about 80 rpm when flooded with a good cutting lubricant.

When ROUGHING parts down to size, use the greatest depth of cut and feed per revolution that the work, the machine, and the tool will stand at the highest practicable speed. On many pieces, when tool failure is the limiting factor in the size of roughing cut, it is usually possible to reduce the speed slightly and increase the feed to a point that the metal removed is much greater. This will prolong tool life. Consider an example when the depth of cut is 1/4 inch, the feed 20 thousandths of an inch per revolution, and the speed 80 fpm. If the tool will not permit additional feed at this speed, you can usually drop the speed to 60 fpm and increase the feed to about 40 thousandths of an inch per revolution without having tool trouble. The speed is therefore reduced 25% but the feed is increased 100%. The actual time required to complete the work is less with the second setup.

On the FINISH TURNING OPERATION, a very light cut is taken since most of the stock has been removed on the roughing cut. A fine feed can usually be used, making it possible to run a high surface speed. A 50% increase in speed over the roughing speed is commonly used. In particular cases, the finishing speed may be twice the roughing speed. In any event, the work should be run as fast as the tool will withstand to obtain the maximum speed in this operation. A sharp tool should be used to finish turning.

Coolants

A cutting lubricant serves two main purposes: (1) It cools the tool by absorbing a portion of the heat and reduces the friction between the tool and the metal being cut. (2) It keeps the cutting edge of the tool flushed clean. A cutting lubricant generally allows you to use a higher cutting speed, heavier feeds, and depths of cut than if you performed the machinery operation dry. The life of the cutting tool is also prolonged by lubricants. Some common materials and their cutting lubricants are as follows:

Cast iron–usually worked dry or with a soluble oil mixture of 1 part of oil to 30 parts of water or mineral lard oil.

Alloy steel–soluble oil mixture of 1 part of oil to 10 parts of water or mineral lard oil.

Low/medium carbon steel–soluble oil mixture of 1 part of oil to 20 parts of water or mineral lard oil.

Brasses and bronzes–soluble oil mixture of 1 part of oil to 20 parts of water or mineral lard oil.

Stainless steel–soluble oil mixture of 1 part of oil to 5 parts of water or mineral lard oil.

Aluminum–soluble oil mixture of 1 part of oil to 25 parts of water or dry.

Nickel alloys/Monel–soluble oil mixture of 1 part of oil to 20 parts of water or a sulfur-based oil.

Babbitt–dry or with a mixture of mineral lard oil and kerosene.

While the use of a lubricant for straight turning is desirable, it is very important for threading. The various operations used and materials machined on a lathe may cause problems in the selection of the proper lubricant. A possible solution is to select a lubricant that is suitable for the majority of the materials you plan to work with.

Chatter

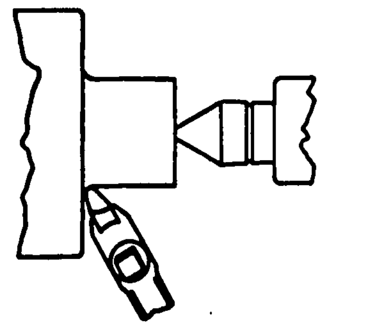
A symptom of improper lathe operation is known as "chatter." Chatter is vibration in either the tool or the work. The finished work surface will appear to have a grooved or lined finish instead of the smooth surface that is expected. The vibration is set up by a weakness in the work, work support, tool, or tool support and is perhaps the most elusive thing you will find in the entire field of machine work. As a general rule, strengthening the various parts of the tool support train will help. It is also advisable to support the work by a center rest or follower rest.

Begin your search for the cause of the chatter by making sure that the surface speed is not excessive. Since excessive speed is probably the most frequent cause of chatter, reduce the speed and see if the chatter stops. You may also increase the feed, particularly if you are taking a rough cut and the finish is not important. Another adjustment you can try is to reduce the lead angle of the tool (the angle formed between the surface of the work and the side cutting edge of the tool). You may do this by positioning the tool closer to a perpendicular to the work.

If none of the above actions work, examine the lathe and its adjustments. Gibs may be too loose or bearings may be worn after a long period of heavy service. If the machine is in perfect condition, the fault may be in the tool or the tool setup. Check to be sure the tool has been properly sharpened to a point or as near to a point as the specified finish will permit. Reduce the overhang of the tool as much as possible and recheck the gib and bearing adjustments. Finally, be sure that the work is properly supported and that the cutting speed is not too high.

Direction of Feed

Regardless of how the work is held in the lathe, the tool should feed toward the headstock. This causes most of the pressure of the cut to be exerted on the workholding device and the spindle thrust bearings. When you must feed the cutting tool toward the tailstock, take lighter cuts at reduced feeds. In facing, the general practice is to feed the tool from the center of the workpiece out toward the periphery.



28.130X
Figure 8-26.—Facing a shoulder.

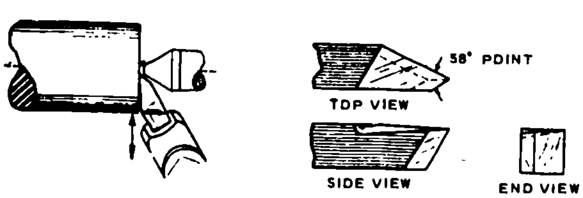
FACING

Facing is the machining of the end surfaces and shoulders of a workpiece. In addition to squaring the ends of the work, facing will let you accurately cut the work to length. Generally, in facing the workpiece you will need to take only light cuts since the work has already been cut to approximate length or rough machined to the shoulder.

Figure 8-25 shows how to face a cylindrical piece. Place the work on centers and install a dog. Using a right-hand side tool, take one or two light cuts from the center outward to true the work.

If both ends of the work must be faced, reverse the piece so that the dog drives the end just faced. Use a steel ruler to layout the required length, measuring from the faced end to the end to be faced. After you ensure that there is no burr on the finished end to cause an inaccurate measurement, mark off the desired dimension with a scribe and face the second end.

Figure 8-26 shows the facing of a shoulder having a fillet corner. First, take a finish cut on the outside of the smaller diameter section. Next, machine the fillet with a light cut by manipulating the apron handwheel and the crossfeed handle in unison to produce a smooth rounded surface. Finally, use the tool to face from the fillet to the outside diameter of the work.



28.129X
Figure 8-25.—Right-hand side tool.

In facing large surfaces, back the carriage in position since only cross feed is required to traverse the tool across the work. With the compound rest set at 90° (parallel to the axis of the lathe), use the micrometer collar to feed the tool to the proper depth of cut in the face. For greater accuracy in getting a given size when finishing a face, set the compound rest at 30°. In this position, .001-inch movement of the compound rest will move the tool exactly .0005-inch in a direction parallel to the axis of the lathe. (In a 30° - 60° right triangle, the length of the side opposite the 30° angle is equal to one-half of the length of the hypotenuse.)

TURNING

Turning is the machining of excess stock from the periphery of the workpiece to reduce the diameter. Bear in mind that the diameter of the work being turned is reduced by the amount equal to twice the depth of the cut; thus, to reduce the diameter of a piece by 1/4 inch, you must remove 1/8 inch of metal from the surface.

To remove large amount of stock in most lathe machining, you will take a series of roughing cuts to remove most of the excess stock and then take a finishing cut to accurately "size" the workpiece.

Rough Turning

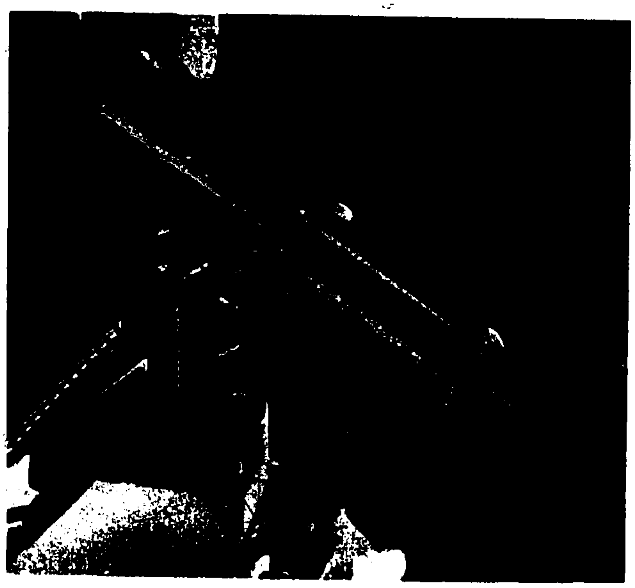
Figure 8-27 illustrates a lathe taking a heavy cut. This is called rough turning. When a great deal of stock is to be removed, you should take heavy cuts in order to complete the job in the least possible time.

Be sure to select the proper tool for taking a heavy chip. The speed of the work and the amount of feed of the tool should be as great as the tool will stand.

When taking a roughing cut on steel, cast iron, or any other metal that has a scale on its surface, be sure to set the tool deep enough to get under the scale in the first cut. Unless you do, the scale on the metal will dull the point of the tool.

Rough machine the work to almost the finished size; then be very careful in measuring.

Often the heat produced during rough turning will expand the workpiece, and the lubricant will flow out of the live center hole. This will result in both center and center hole becoming worn. Always check the center carefully and adjust as needed during rough turning operations.



28.131X

Figure 8-27.—Rough turning.

Figure 8-28 shows the position of the tool for taking a heavy chip on large work. Set the tool so that if anything causes it to change position during the machining operation, the tool will move away from the work, thus preventing damage to the work. Also, setting the tool in this position may prevent chatter.

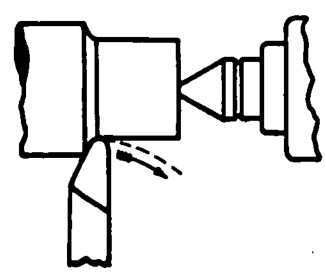
Finish Turning

When the work has been rough turned to within about 1/32 inch of the finished size, take a finishing cut. A fine feed, the proper lubricant, and above all a keen-edged tool are necessary to produce a smooth finish. Measure carefully to be sure that you are machining the work to the proper dimension. Stop the lathe whenever you take any measurements.

If you must finish the work to extremely close tolerances, wait until the piece is cool before taking the finish cut. If you take the finish cut while the piece is hot and slightly expanded, the piece will be undersize after it has cooled and contracted.

If you plan to finish the work on a cylindrical grinder, leave the stock slightly oversize to allow for the metal the grinder will remove.

Perhaps the most difficult operation for a beginner in machine work is to make accurate measurements. So much depends on the accuracy of the work that you should make every effort to become proficient in using



28.132X

Figure 8-28.—Position of tool for heavy cut.

measuring instruments. You will develop a certain "feel" through experience. Do not be discouraged if your first efforts do not produce perfect results. Practice taking measurements on pieces of know dimensions. You will acquire the skill if you are persistent.

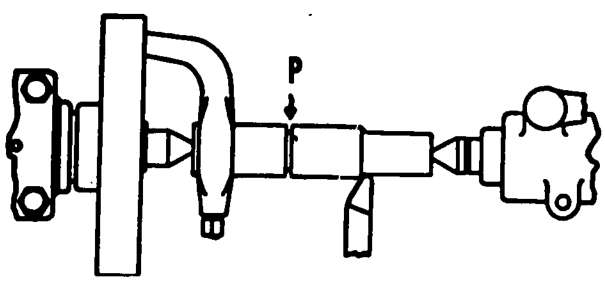
Turning to a Shoulder

A time saving procedure for machining a shoulder is illustrated in figure 8-29. First, locate and scribe the exact location of the shoulder on the work. Next, use a parting tool to machine a groove 1/32 inch from the scribe line toward the smaller finish diameter end and 1/32 larger than the smaller finish diameter. Then take heavy cuts up to the shoulder made by the parting tool. Finally, take a finish cut from the small end to the shoulder scribe line. This procedure eliminates detailed measuring and speeds up production.

PARTING AND GROOVING

One of the methods of cutting off a piece of stock while it is held in a lathe is a process called parting. This process uses a specially shaped tool with a cutting edge similar to that of a square nose tool. The parting tool is fed into the rotating work, perpendicular to its axis, cutting a progressively deeper groove as the work rotates. When the cutting edge of the tool gets to the center of the work being parted, the work drops off as if it were sawed off. Parting is used to cut off parts that have already been machined in the lathe or to cut tubing and bar stock to required lengths.

Parting tools can be the inserted blade type or can be ground from a standard tool blank. Figure 8-30 shows the two types of parting tools. For the tool to have maximum strength, the length of the cutting portion of the blade that extends from the holder should be only slightly greater than half the diameter of the work to parted. Place the end cutting edge of the parting tool exactly on the centerline of the lathe. To do this, place a center in the tailstock and align the parting tool with the tip of the



28.133X

Figure 8-29.—Machining to a shoulder.

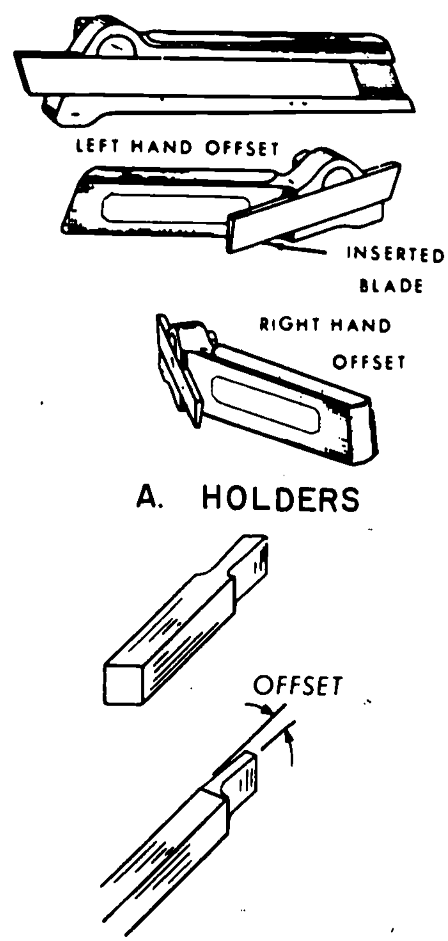


Figure 8-30.—Parting tools.

center. The chuck should hold the work to be parted with the point at which the parting is to occur as close as possible to the chuck jaws. Always make the parting cut at a right angle to the centerline of the work. Feed the tool into the revolving work with the cross-slide until the tool completely severs the work.

Cutting speeds for parting are usually slower than turning speeds. You should use a feed that will keep a thin chip coming from the work. If chatter occurs, decrease the speed and increase the feed slightly. If the tool tends to gouge or dig in, decrease the feed.

Grooves are machined in shafts to provide for tool runout in threading to a shoulder, to allow clearance for assembly of parts, to provide lubricating channels, or to provide a seating surface for seals and O-rings. Square, round, and "V" grooves and the tools which are used to produce them are shown in figure 8-31.

The grooving tool is a type of forming tool. It is ground without side rake or back rake and is set to the work at center height with a minimum of overhang. The side and end relief angles are generally somewhat less than for turning tools. When you machine a groove, reduce the spindle speed to prevent chatter which often develops at high speeds because of the greater amount of tool contact with the work.

DRILLING AND REAMING

Drilling operations performed in a lathe differ very little from drilling operations performed in a drilling machine. For best results,

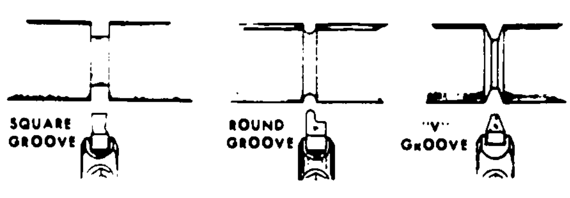


Figure 8-31.—Three common types of grooves.

start the drilling operation by drilling a center hole in the work, using a combination center drill and countersink. The combination countersink-center drill is held in a drill chuck which is mounted in the tailstock spindle. After you have center drilled the work, replace the drill chuck with a taper shank drill. (Note: BEFORE you insert any tool into the tailstock spindle, inspect the shank of the tool for burrs. If the shank is burred, remove the burrs with a handstone.) Feed the drill into the work by using the tailstock handwheel. Use a coolant/lubricant whenever possible and maintain sufficient pressure on the drill to prevent chatter, but not enough to overheat the drill.

If the hole is quite long, back the drill out occasionally to clear the flutes of metal chips. Large diameter holes may require you to drill a pilot hole first. This is done with a drill that is smaller than the finished diameter of the hole. After you have drilled the pilot hole to the proper depth, enlarge the hole with the finish drill. If you plan to drill the hole completely through the work, slow down the feed as the drill nears the hole exit. This will produce a smoother exit hole by causing the drill to take a finer cut as it exits the hole.

If the twist drill is not ground correctly, the drilled hole will be either excessively oversized or out of round. Check the drill for the correct angle, clearance, cutting edge lengths and straightness before setting it up for drilling. It is almost impossible to drill a hole exactly the same size as the drill regardless of the care taken in ensuring an accurately ground drill and the proper selection of speeds and feeds. For this reason, any job which requires close tolerances or a good finish on the hole should be reamed or bored to the correct size.

If the job requires that the hole be reamed, it is good practice to first take a cleanup cut through the hole with a boring tool. This will true up the hole for the reaming operation. Be sure to leave about 1/64 inch for reaming. The machine reamer has a taper shank and is held in and fed by the tailstock. To avoid overheating the reamer, the work speed should be about half that used for the drilling operation. During the reaming operation, keep the reamer well lubricated. This will keep the reamer cool and also flush the chips from the flutes. Do not feed the reamer too fast; it may tear the surface of the hole and ruin the work.

BORING

Boring is the machining of holes or any interior cylindrical surface. The piece to be bored must have a drilled or core hole, and the hole must be large enough to insert the tool. The boring process merely enlarges the hole to the desired size or shape. The advantage of boring is that you get a perfectly true round hole. Also, you can bore two or more holes of the same or different diameters at one setting, thus ensuring absolute alignment of the axis of the holes.

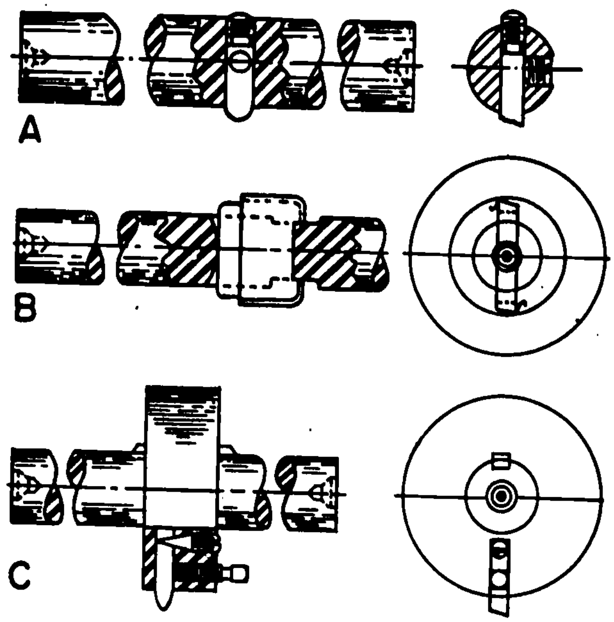
It is usual practice to bore a hole to within a few thousandths of an inch of the desired size and then finish it to the exact size with a reamer.

Work to be bored may be held in a chuck, bolted to the faceplate, or bolted to the carriage. Long pieces must be supported at the free end of a center rest.

When the boring tool is fed into the hole in work being rotated on a chuck or faceplate, the process is called single point boring. It is the same as turning except that the cutting chip is taken from the inside. The cutting edge of the boring tool resembles that of a turning tool. Boring tools may be the solid forged type or the inserted cutter bit type.

When the work to be bored is clamped to the top of the carriage, a boring bar is held between centers and driven by a dog. The work is fed to the tool by the automatic longitudinal feed of the carriage. Three types of boring bars are shown in figure 8-32. Note the countersunk center holes at the ends to fit the lathe centers.

Part A of figure 8-32 shows a boring bar fitted with a fly cutter held by a headless setscrew. The other setscrew, bearing on the end of the cutter, is for adjusting the cutter to the work.

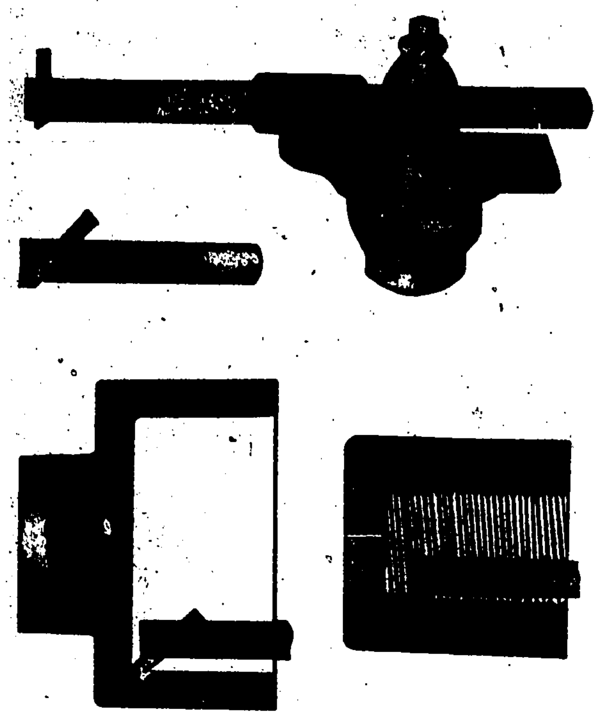


28.134
Figure 8-32.—Various boring bars.

Part B of figure 8-32 shows a boring bar fitted with a two-edge cutter held by a taper key. This is more of a finishing or sizing cutter, as it cuts on both sides and is used for production work.

The boring bar shown in part C of figure 8-32 is fitted with a cast iron head to adapt it for boring work of large diameter. The head is fitted with a fly cutter similar to the one shown in part A. The setscrew with the tapered point adjusts the cutter to the work.

Figure 8-33 shows a common type of boring bar holder and applications of the boring bar for boring and internal threading. When threading is to be done in a blind hole, it sometimes becomes necessary to undercut or relieve the bottom of the hole. This will enable mating parts to be screwed all the way to the shoulder and make



28.135
Figure 8-33.–Application of boring bar holder.

the threading operation much easier to accomplish.

KNURLING

Knurling is the process of rolling or squeezing impressions into the work with hardened steel rollers that have teeth milled in their faces. Examples of the various knurling patterns are shown in chapter 7, figure 7-22. Knurling provides a gripping surface on the work; it is used also for decoration. Knurling increases the diameter of the workpiece slightly when the metal is raised by the forming action of the knurl rolls.

The knurling tool (fig. 7-23) is set up so that the faces of the rolls are parallel to the surface of the work and with the upper and lower rolls equally spaced above and below the work axis or centerline. The spindle speed should be about half the roughing speed for the type of metal being machined. The feed should be between 0.015 inch and 0.025 inch per revolution. The work should be rigidly mounted in the tailstock to help offset the pressure exerted by the knurling operation.

The actual knurling operation is simple if you follow a few basic rules. The first step is to make sure that the rolls in the knurling tool turn freely and are free of chips and imbedded metal between the cutting edges. During the knurling process, apply an ample supply of oil at the point of contact to flush away chips and provide lubrication. Position the carriage so that 1/3 to 1/2 of the face of the rolls extends beyond the end of the work. This eliminates part of the pressure required to start the knurl impression. Force the knurling rolls into contact with the work. Engage the spindle clutch. Check the knurl to see if the rolls have tracked properly, as shown in figure 8-34, by disengaging the clutch after the work has revolved 3 or 4 times and by backing the knurling tool away from the work.

If the knurls have double tracked, as shown in figure 8-34, move the knurling tool to a new location and repeat the operation. If the knurl is correctly formed, engage the spindle clutch and the carriage feed. Move the knurling rolls into contact with the correctly formed knurled impressions. The rolls will align themselves with the impressions. Allow the knurling tool to feed to within about 1 /32 inch of the end of the surface to be knurled. Disengage the carriage

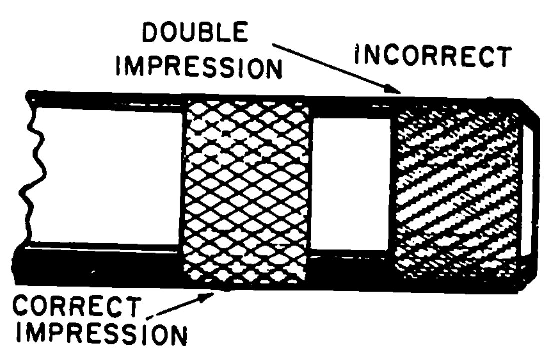


Figure 8-34.–Knurled impressions.

feed and with the work revolving, feed the carriage by hand to extend the knurl to the end of the surface. Force the knurling tool slightly deeper into the work, reverse the direction of feed and engage the carriage feed. Allow the knurling tool to feed until the opposite end of the knurled surface is reached. Never allow the knurls to feed off the surface.

Repeat the knurling operation until the diamond impressions converge to a point. Passes made after the correct shape is obtained will result in stripping away the points of the knurl. Clean the knurl with a brush and remove any burrs or sharp edges with a file. When knurling, do not let the work rotate while the tool is in contact with it if the feed is disengaged. This will cause rings to be formed on the surface, as shown in figure 8-35.

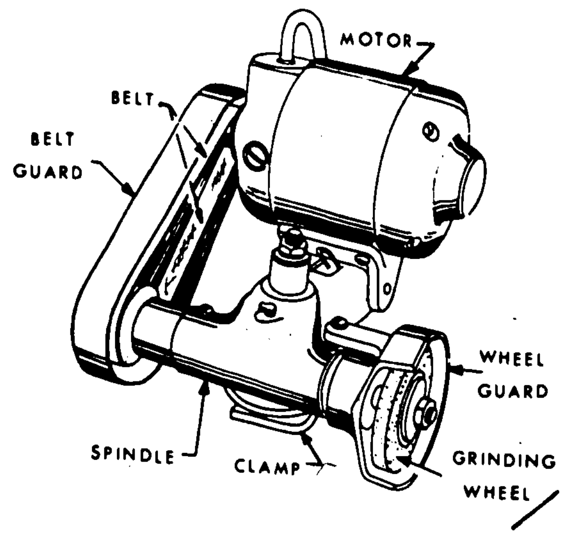


Figure 8-36.—Toolpost grinder.

SETTING UP THE TOOLPOST GRINDER

The toolpost grinder is a portable grinding machine that can be mounted on the compound rest of a lathe in place of the toolpost. It can be used to machine work that is too hard to cut by ordinary means or to machine work that requires a very fine finish. Figure 8-36 shows a typical toolpost grinder.

The grinder must be set on center, as shown in figure 8-37. The centering holes located on the spindle shaft are used for this purpose. The grinding wheel takes the place of a lathe cutting tool; it can perform most of the same operations as a cutting tool. Cylindrical, tapered, and internal surfaces can be ground with the toolpost grinder. Very small grinding wheels are mounted on tapered shafts, known as quills, to grind internal surfaces.

The grinding wheel speed is changed by using various sizes of pulleys on the motor and spindle shafts. An instruction plate on the grinder gives both the diameter of the pulleys required to obtain a given speed and the maximum safe speed for grinding wheels of various diameters. Grinding wheels are safe for operation at a speed just below the highest

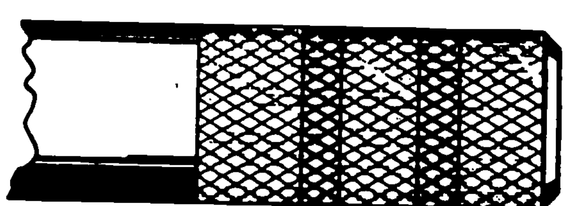


Figure 8-35.—Rings on a knurled surface.

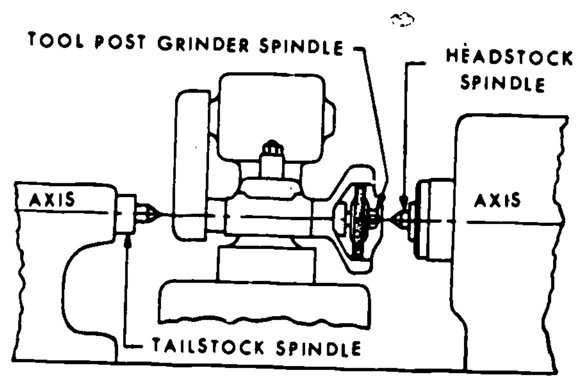


Figure 8-37.—Mounting the grinder at center height.

recommended speed. A higher than recommended speed may cause the wheel to disintegrate. For this reason, wheel guards are furnished with the toolpost grinder to protect against injury.

Always check the pulley combinations given on the instruction plate of the grinder when you mount a wheel. Be sure that the combination is not reversed, because this may cause the wheel to run at a speed far in excess of that recommended. During all grinding operations, wear goggles to protect your eyes from flying abrasive material.

The grinding wheel must be dressed and trued. Use a diamond wheel dresser to dress and true the wheel. The dresser is held in a holder that is clamped to the chuck or faceplate of the lathe. Set the point of the diamond at center height and at a 10° to 15° angle in the direction of the grinding wheel rotation, as shown in figure 8-38. The 10° to 15° angle prevents the diamond from gouging the wheel. Lock the lathe spindle by placing the spindle speed control lever in the low rpm position. (Note: The lathe spindle does not revolve when you are dressing the grinding wheel.)

Bring the grinding wheel in contact with the diamond by carefully feeding the cross-slide in by hand. Move the wheel clear of the diamond and make a cut by means of the cross-slide. The maximum depth of cut is 0.002 inch. Move the wheel slowly by hand back and forth over the point of the diamond. Move the carriage if the face of the wheel is parallel to the ways of the lathe. Move the compound rest if the face of the wheel is at an angle. Make the final depth of cut of 0.0001 inch with a slow, even feed to obtain a good wheel finish. Remove the diamond dresser holder as soon as you finish dressing the wheel and adjust the grinder to begin the grinding operation.

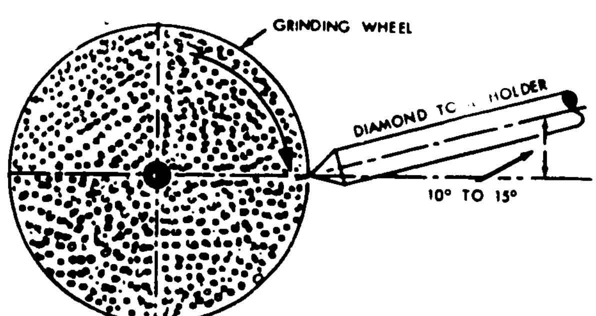


Figure 8-38.—Position of the diamond dresser.

Rotate the work at a fairly low speed during the grinding operation The recommended surface foot speed is 60 to 100 feet per minute (fpm). The depth of cut depends upon the hardness of the work, the type of grinding wheel, and the desired finish. Avoid taking grinding cuts deeper than 0.002 inch until you gain experience. Use a fairly low rate of feed. You will soon be able to judge whether the feed should be increased or decreased. Never stop the work or the grinding wheel while they are in contact with each other.

To refinish a damaged lathe center, you should first install headstock and tailstock centers after ensuring that the spindle holes, drill sleeves, and centers are clean and free of burrs. Next, position the compound rest parallel to the ways; then, mount the toolpost grinder on the compound rest. Make sure that the grinding wheel spindle is at center height and aligned with the lathe centers. Move the compound rest 30° to the right of the lathe spindle axis, as shown in figure 8-5. Mount the wheel dresser, covering the ways and carriage with rags to protect them from abrasive particles. Wear goggles to protect your eyes.

Start the grinding motor, by alternately turning it on and off (let it run a bit longer each time) until the abrasive wheel is brought up to top speed. Dress the wheel, feeding the grinder with the compound rest. Then move the grinder clear of the headstock center and remove the wheel dresser. Set the lathe for the desired spindle speed and engage the spindle. Pick up the surface of the center. Take a light depth of cut and feed the grinder back and forth with the compound rest. Do not allow the abrasive wheel to feed entirely off the center. Continue taking additional cuts until the center cleans up. To produce a good finish, reduce the feed rate and the depth of cut to .0005. Grind off the center's sharp point, leaving a flat with a diameter about 1/32 inch. Move the grinder clear of the headstock and turn it off.

Figure 8-39 illustrates refacing the seat of a high-pressure steam valve which has a hard, Stellite-faced surface. The refacing must be done with a toolpost grinder. Be sure that all inside diameters run true before starting the machine work. Spindle speed of the lathe should be about 40 rpm or less. Too high a speed will cause the grinding wheel to vibrate. Set the compound rest to correspond with the valve seat angle. Use the cross-slide hand feed or the micrometer stop on the carriage for controlling depth of cut; use the compound rest for traversing the grinding wheel across the work surface. When grinding on a lathe, always place a cloth across the ways of the bed and over any other machined surfaces that could become contaminated from grinding dust.

28.136
Figure 8-39.—Refacing seat of high-pressure steam valve.

CHAPTER 9

ADVANCED ENGINE LATHE OPERATIONS

In chapter 8 you studied a number of lathe operations, the various methods of holding and centering work on the engine lathe, and how to set lathe tools. This chapter is a continuation of engine lathe operations and deals primarily with cutting tapers, boring, and cutting screw threads.

TAPERS

Taper is the gradual decrease in the diameter of thickness of a piece of work toward one end. To find the amount of taper in any given length of work, subtract the size of the small end from the size of the large end. Taper is usually expressed as the amount of taper per foot of length, or as an angle. The following examples explain how to determine taper per foot of length.

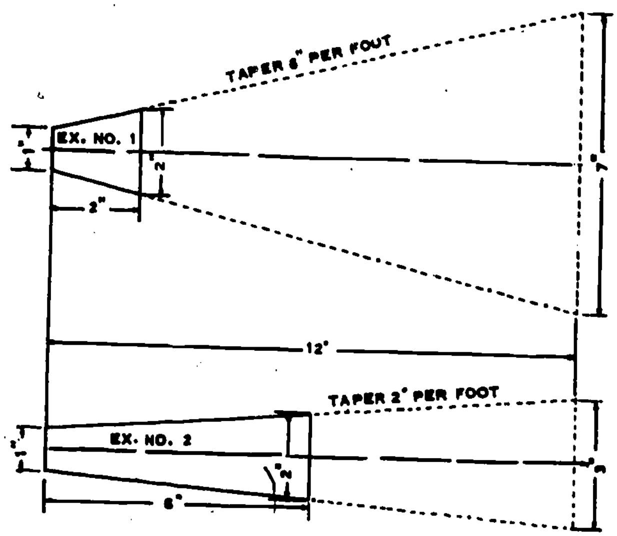
EXAMPLE 1: Find the taper per foot of a piece of work 2 inches long: Diameter of the small end is 1 inch; diameter of the large end is 2 inches.

The amount of the taper is 2 inches minus 1 inch, which equals 1 inch. The length of the taper is given as 2 inches. Therefore, the taper is 1 inch in 2 inches of length. In 12 inches of length it would be 6 inches. (See fig. 9-1.)

EXAMPLE 2: Find the taper per foot of a piece 6 inches long. Diameter of the small end is 1 inch; diameter of the large end is 2 inches.

The amount of taper is the same as in example 1; that is, 1 inch. (See fig. 9-1.) However, the length of this taper is 6 inches; hence the taper per foot is 1 inch X 12/6 = 2 inches per foot.

From the foregoing, you can see that the length of a tapered piece is very important in computing the taper. If you bear this in mind when machining tapers, you will not go wrong. Use the formula:



28.137

Figure 9-1.—Tapers.

TPF = FPI X 12

where: TPF = TAPER PER FOOT

T = TAPER (Difference between large and small diameters), expressed in inches

L = LENGTH of taper, expressed in inches

Other formulas used in figuring tapers are as follows:

$$TPI = \frac{T}{L}$$

where: TPI = TAPER PER INCH

T = TAPER (Difference between large and small diameters), expressed in inches

L = LENGTH of taper, expressed in inches

$$T = \frac{L \times TPF}{12} \text{ and } T = TPI \times L$$

$$TPI = \frac{TPF}{12}$$

Tapers are frequently cut by setting the angle of the taper on the appropriate lathe attachment. There are two angles associated with a taper: the included angle and the angle with the centerline. The included angle is the angle between the two angled sides of the taper. The angle with the centerline is the angle between the centerline and either of the angled sides. Since the taper is turned about a centerline, the angle between one side and the centerline is always equal to the angle between the other side and the centerline. Therefore, the included angle is always twice the angle with the centerline. The importance of this relationship will be shown later in this chapter. Table 9-1 is a machinist's chart showing the relationship between taper per foot, included angle, and angle with the centerline.

There are several well-known tapers that are used as standards for machines on which they are used. These standards make it possible to make or get parts to fit the machine in question without detailed measuring and fitting. By designating the name and number of the standard taper being used, you can immediately find the length, the diameter of the small and

Table 9-1.—Tapers Per Foot/Angles

Taper per foot	Angle included		Angle with centerline		Taper per inch from centerline
	Degrees	Minutes	Degrees	Minutes	Inches
1/8	0	36	0	18	0.01042
3/16.	0	54	0	27	.01563
1/4	1	12	0	36	.02083
5/16.	1	30	0	45	.02604
3/8	1	47	0	54	.03125
7/16.	2	5	1	3	.03646
1/2	2	23	1	12	.04167
9/16.	2	41	1	21	.04688
5/8	3	0	1	30	.05208
11/16	3	17	1	38	.05729
3/4	3	35	1	47	.06250
13/16	3	53	1	56	.06771
7/8	4	11	2	5	.07292
15/16	4	28	2	14	.07813
1	4	46	2	23	.08333
2	9	32	4	46	.16667

28.183.0

large ends, the taper per foot, and all other pertinent measurements in appropriate tables found in most machinist's handbooks.

There are three standard tapers with which you should be familiar: (1) the MORSE TAPER (approximately 5/8 inch per foot) used for the taper holes in lathe and drill press spindles and the attachments that fit them, such as lathe centers, drill shanks, etc; (2) the BROWN & SHARPE TAPER (1/2 inch per foot except No. 10 which is 0.5161 inch per foot) used for milling machine spindle shanks; and (3) the MILLING MACHINE TAPER of 3.5 inches per foot used by some manufacturers because of the ease with which its dimensions can be determined:

$$\text{Diameter of large end} = \frac{\text{taper number}}{8}$$

$$\text{Diameter of small end} = \frac{\text{taper number}}{10}$$

$$\text{Length of taper} = \frac{\text{taper number}}{2}$$

Two additional tapers which are considered standard are the tapered pin and pipe thread tapers. Tapered pins have a taper of 1/4 inch per foot while tapered pipe threads have a taper of 3/4 inch per foot.

A copy of a Morse taper table is shown in figure 9-2. You will no doubt have more use for this taper than any other standard taper.

METHODS OF TURNING TAPERS

In ordinary straight turning, the cutting tool moves along a line parallel to the axis of the work, causing the finished job to be the same diameter throughout. If, however, in cutting, the tool moves at an angle to the axis of the work, a taper will be produced. Therefore, to turn a taper, you must either mount the work in the lathe so the axis upon which it turns is at an angle to the axis of the lathe, or cause the cutting tool to move at an angle to the axis of the lathe.

There are three methods in common use for turning tapers:

1. SET OVER THE TAILSTOCK, which moves the dead center away from the axis of the lathe and causes work supported between centers to be at an angle with the axis of the lathe.

2. USE THE COMPOUND REST set at an angle, which causes the cutting tool to be fed at the desired angle to the axis of the lathe.

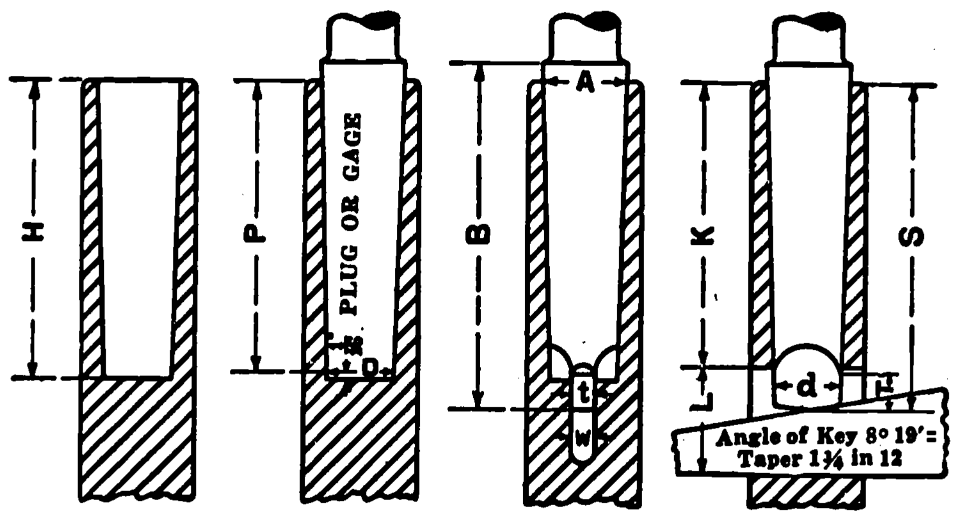
3. USE THE TAPER ATTACHMENT, which also causes the cutting tool to move at an angle to the axis of the lathe.

In the first method, the cutting tool is fed by the longitudinal feed parallel to the lathe axis, but a taper is produced because the work axis is at an angle. In the second and third methods, the work axis coincides with the lathe axis, but a taper is produced because the cutting tool moves at an angle.

Setting Over the Tailstock

As stated in chapter 7, you can move the tailstock top sideways on its base by using the adjusting screws. In straight turning, you will recall that you used these adjusting screws to align the dead center with the tail center by moving the tailstock to bring it on the centerline of the spindle axis. For taper turning, you deliberately move the tailstock off center, and the amount you move it determines the taper produced. You can approximate the amount of setover by the zero lines inscribed on the base and top of the tailstock as shown in figure 9-3. Then for final adjustment, measure the setover with a scale between center points as illustrated in figure 9-4.

In turning a taper by this method, the distance between centers is of utmost



DETAIL DIMENSIONS

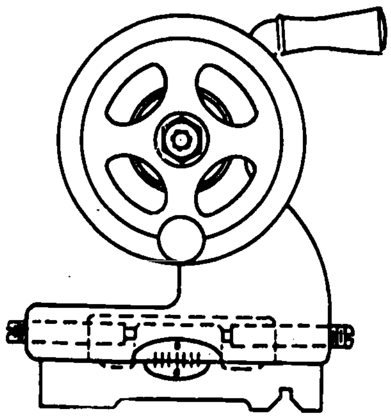
Number of Taper	0	1	2	3	4	5	6	7
Diameter of plug at small end. . D	0.252	0.369	0.572	0.778	1.020	1.475	2.116	2.750
Diameter at end of socket . . . . A	.3561	.475	.700	.938	1.231	1.748	2.494	3.270
Shank:								
Whole length of shank . . . . . B	2-11/32	2-9/16	3-1/8	3-7/8	4-7/8	6-1/8	8-9/16	11-5/8
Shank depth. . . . . S	2-7/32	2-7/16	2-15/16	3-11/16	4-5/8	5-7/8	8-1/4	11-1/4
Depth of hole. . . . . H	2-1/32	2-3/16	2-5/8	3-1/4	4-1/8	5-1/4	7-3/8	10-1/8
Standard plug depth . . . . P	2	2-1/8	2-9/16	3-3/16	4-1/16	5-3/16	7-1/4	10
Tongue:								
Thickness of tongue . . . . . . t	5/32	13/64	1/4	5/16	15/32	5/8	3/4	1-1/8
Length of tongue . . . . T	1/4	3/8	7/16	9/16	5/8	3/4	1-1/8	1-3/8
Diameter of tongue. . . . . . . d	.235	.343	17/32	23/32	31/32	1-13/32	2	2-5/8
Keyway:								
Width of keyway. . . . . W	.160	.213	.260	.322	.478	.635	.760	1.135
Length of keyway L	9/16	3/4	7/8	1-3/16	1-1/4	1-1/2	1-3/4	2-5/8
End of socket to keyway . . . . . K	1-15/16	2-1/16	2-1/2	3-1/16	3-7/8	4-15/16	7	9-1/2
Taper per foot.	.625	.600	.602	.602	.623	.630	.626	.625
Taper per inch	.05208	.05	.05016	.05016	.05191	.0525	.05216	.05208
Number of key.	0	1	2	3	4	5	6	7

28.138X

Figure 9-2.—Morse tapers.

importance. To illustrate, figure 9-5 shows two very different tapers produced by the same amount of setover of the tailstock, because for one taper the length of the work between centers is greater than for the other. THE CLOSER THE DEAD CENTER IS TO THE LIVE CENTER, THE STEEPER IS THE TAPER PRODUCED.

Suppose you want to turn a taper on the full length of a piece 12 inches long with one end having a diameter of 3 inches, and the other end a diameter of 2 inches. The small end is to be 1 inch smaller than the large end; so you set the tailstock over one-half this amount or 1/2 inch in this example. Thus, at one end the cutting tool will be 1/2 inch closer to the center of the



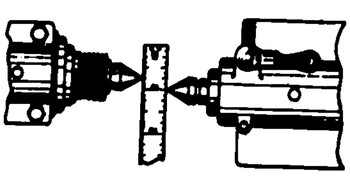
28.139(28D)X
Figure 9-3.—Tailstock setover lines for taper turning.

work than at the other end; so the diameter of the finished job will be 2 X 1/2 or 1 inch less at the small end. Since the piece is 12 inches long, you have produced a taper of 1 inch per foot. Now, if you wish to produce a taper of 1 inch per foot on a piece only 6 inches long, the small end will be only 1/2 inch less in diameter than the larger end, so you should set over the tailstock 1/4 inch or one-half of the distance used for the 12-inch length.

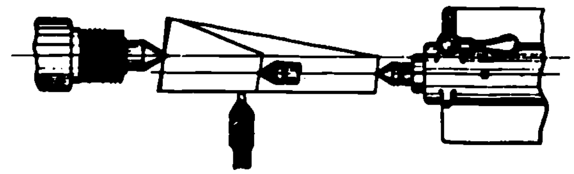
By now you can see that the setover is proportional to the length between centers. Setover is computed by the following formula:

$$S = \frac{T}{2} \times \frac{L}{12}$$

where:
S = setover in inches
T = taper per foot in inches
$\frac{L}{12}$ = length in foot of taper



28.140X
Figure 9-4.—Measuring setover of dead center.



28.141X
Figure 9-5.—Setover of tailstock showing importance of considering length of work.

Remember that L is the length of the work from live center to dead center. If the work is on a mandrel, L is the length of the mandrel between centers. You cannot use the setover tailstock method for steep tapers because the setover would be too great and the work would not be properly supported by the lathe centers. It is obvious that with setover there is not a true bearing between the work centers and the lathe center points and the bearing surface becomes less and less satisfactory as the setover is increased. CAUTION: DO NOT EXCEED .250-inch setover.

After turning a taper by the tailstock setover method, do not forget to realign the centers for straight turning of your next job.

Using the Compound Rest

The compound rest is generally used for short, steep tapers. Set it at the angle the taper is to make with the centerline (that is, half the included angle of the taper). Then feed the tool to the work at this angle by the compound rest feed screw. The length of taper you can machine is necessarily short because of limited travel of the compound rest top.

Truing a lathe center is one example of using the compound rest for taper work. Other examples are refacing an angle type valve disk, machining the face of a bevel gear, and similar work. Such jobs are often referred to as working to an angle rather than as taper work.

The graduations marked on the compound rest are a quick means for setting it to the angle desired. When set at zero, the compound rest is perpendicular to the lathe axis. When set at 90° on either side, the compound rest is parallel to the lathe axis.

On the other hand, when the angle to be cut is measured from the centerline, the setting of the compound rest corresponds to the complement of that angle (the complement of an angle is that angle which added to it makes a right angle; that is, angle plus complement = 90°). For example, to machine a 50° included angle (25° angle with the centerline), set the compound rest at 90° - 25°, or 65°.

When a very accurate setting of the compound rest is to be made to a fraction of a degree for example, run the carriage up to the faceplate and set the compound rest with a vernier bevel protractor set to the required angle. Hold the blade of the protractor on the flat surface of the faceplate and hold the base of the protractor against the finished side of the compound rest.

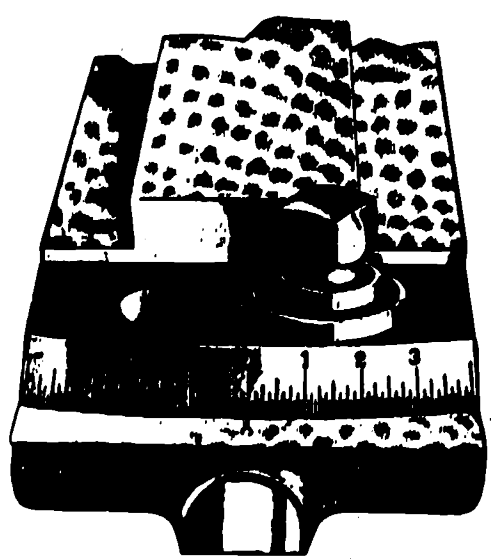
For turning and boring long tapers with accuracy, the taper attachment is indispensable. It is especially useful in duplicating work; you can turn and bore identical tapers with one setting of the taper guide bar. Set the guide bar at an angle to the lathe which corresponds to the desired taper. The tool cross-slide is then moved laterally by a shoe which slides on the guide bar as the carriage moves longitudinally. The resulting movement of the cutting tool is along a line that is parallel to the guide bar, and therefore a taper is produced whose angular measurement is the same as that set on the guide bar. The guide bar is graduated in degrees at one end and in inches per foot of taper at the other end to facilitate rapid setting. Figure 9-6 is a view of the end that is graduated in inches per foot of taper.

When preparing to use the taper attachment, run the carriage up to the approximate position of the work to be turned. Set the tool on line with the centers of the lathe. Then bolt or clamp the holding bracket to the ways of the bed (the attachment itself is bolted to the back of the carriage saddle) and tighten clamp C, figure 9-7. The taper guide bar now controls the lateral movement of the cross-slide. Set the guide bar for the taper desired; the attachment is ready for operation. The final adjustment of the tool for size must be made by means of the compound rest feed screw, since the crossfeed screw is inoperative.

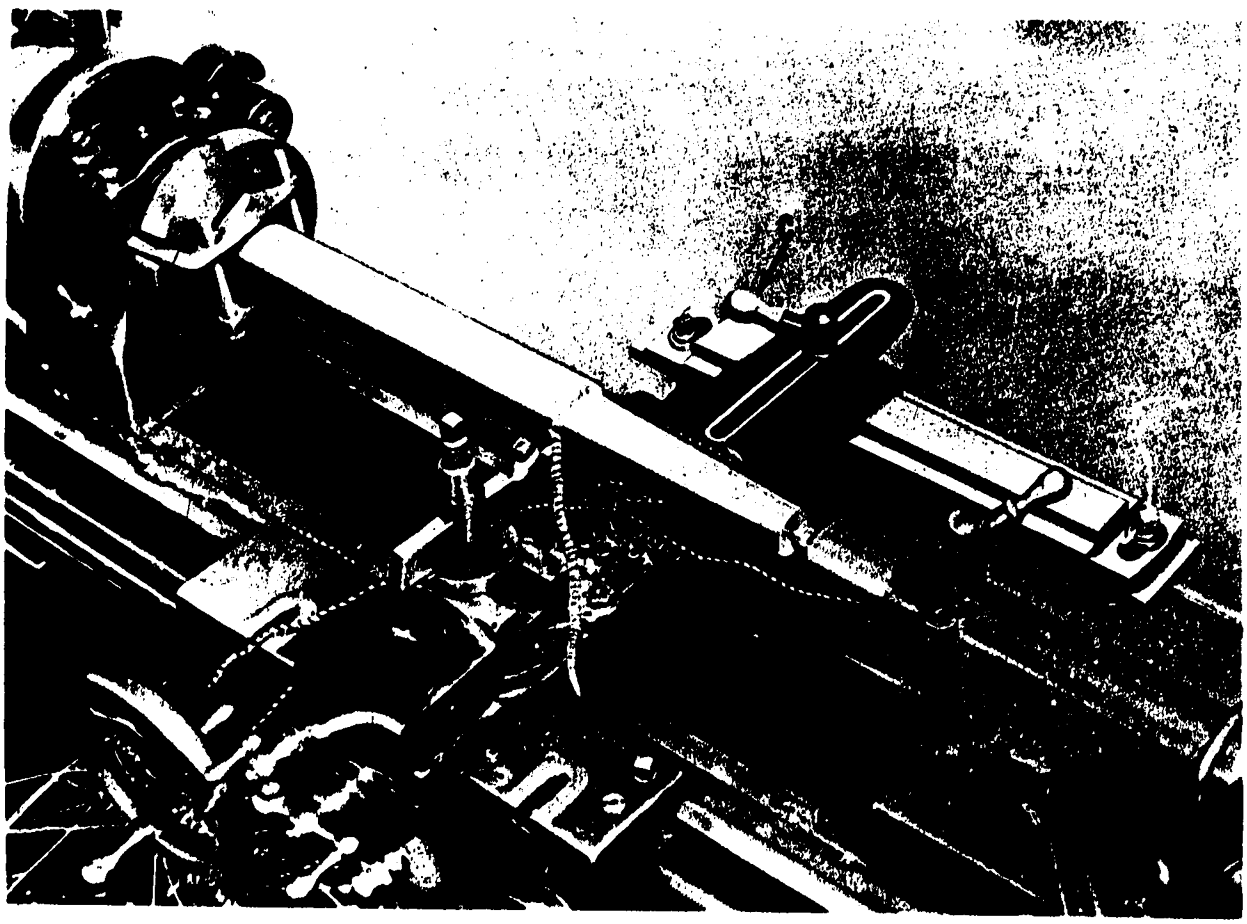
There will be a certain amount of lost motion or backlash when the tool first starts to feed along the work. This is caused by looseness between the crossfeed screw and the cross-slide nut. If the backlash is not eliminated, a straight portion will be turned on the work. You can remove the backlash by moving the carriage and tool slightly past the start of the cut and then returning the carriage and tool to the start of the cut.

TAPER BORING

Taper boring is usually done with either the compound rest or the taper attachment. The rules that apply to outside taper turning also apply to the boring of taper holes. Begin by drilling the hole to the correct depth with a drill of the same size as the specified small diameter of the taper. This gives you the advantage of



28.142X
Figure 9-6.—End view of taper guide bar.



28.143X

Figure 9-7.–Turning a taper using taper attachment.

boring to the right size without having to remove metal at the bottom of the bore, which is rather difficult, particularly in small, deep holes.

For turning and boring tapers, set the tool cutting edge exactly at the center of the work. That is, set the point of the cutting edge even with the height of the lathe centers; otherwise, the taper may be inaccurate.

Cut the hole and measure its size and taper using a taper plug gage and the "cut and try" method:

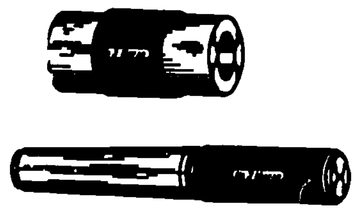
1. After you have taken one or two cuts, clean the bore.

2. Rub the gage lightly with chalk (or prussian blue if the width must be highly accurate).

3. Insert the gage into the hole and turn it SLIGHTLY so that the chalk (or prussian blue) rubs from the gage onto the surface of the hole. If the workpiece is to be mounted on a spindle, use the tapered end of the spindle instead of a gage to test the taper.

4. Areas that do not touch the gage will be shown by a lack of chalk (or prussian blue).

5. Continue making minor corrections until all, or an acceptable portion, of the hole's surface touches the gage. Be sure the taper diameter is correct before you turn the taper to its finish diameter.



28.144X
Figure 9-8.—Morse taper socket gage and plug gage.

Figure 9-8 shows a Morse standard taper plug and a taper socket gage. They not only give the proper taper, but also show the proper distance that the taper should enter the spindle.

SCREW THREADS

Much of the machine work performed by a Machinery Repairman includes the use of screw threads. The thread forms you will be working with most are V-form threads, Acme threads, and square threads. Each of these thread forms is used for specific purposes. V-form threads are commonly used on fastening devices such as bolts and nuts as well as on machine parts. Acme screw threads are generally used for transmitting motion, such as between the lead screw and lathe carriage. Square threads are used to increase mechanical advantage and to provide good clamping ability as in the screw jack or vise screw. Each of these screw forms is discussed more fully later in the chapter.

There are several terms used in describing screw threads and screw thread systems which you must know before you can calculate and machine screw threads. Figure 9-9 shows the application of some of the following terms:

EXTERNAL THREADS: A thread on the external surface of a cylinder.

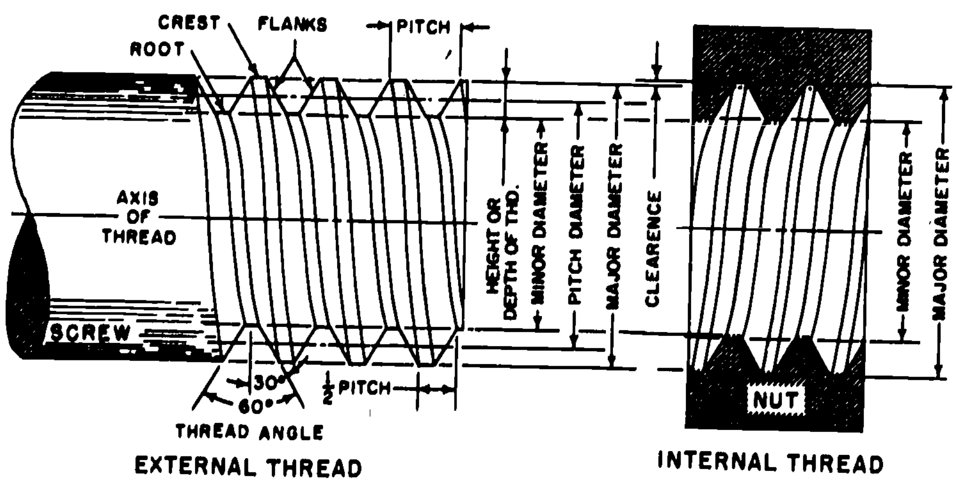
INTERNAL THREAD: A thread on the internal surface of a hollow cylinder.

RIGHT-HAND THREAD: A thread which, when viewed axially, winds in a clockwise and receding direction.

LEFT-HAND THREAD: A thread which, when viewed axially, winds in a counterclockwise and receding direction.

LEAD: The distance a threaded part moves axially in a fixed mating part in one complete revolution.

PITCH: The distance between corresponding points on adjacent threads.



28.145(28D)
Figure 9-9.—Screw thread nomenclature.

SINGLE THREAD: A single (single start) thread whose lead equals the pitch.

MULTIPLE THREAD: A multiple (multiple start) thread whose lead equals the pitch multiplied by the number of starts.

CLASS OF THREADS: Classes of threads are distinguished from each other by the amount of tolerance and allowance specified.

THREAD FORM: The axial plane profile of a thread for a length of one pitch.

FLANK: The side of the thread.

CREST: The top of the thread (bounded by the major diameter on external threads; by the minor diameter on internal threads).

ROOT: The bottom of the thread (bounded by the minor diameter on external threads; by the major diameter on internal threads).

THREAD ANGLE: The angle formed by adjacent flanks of a thread.

PITCH DIAMETER: The diameter of an imaginary cylinder that is concentric with the thread axis and whose periphery passes through the thread profile at the point where the width of the thread and the thread groove are equal. The pitch diameter is the diameter which is measured when the thread is machined to size. A change in pitch diameter changes the fit between the thread being machined and the mating thread.

NOMINAL SIZE: The size which is used for identification. For example, the nominal size of a 1/2-20 thread is 1/2 inch, but its actual size is slightly smaller to provide clearance.

ACTUAL SIZE: The measured size.

BASIC SIZE: The theoretical size. The basic size is changed to provide the desired clearance or fit.

MAJOR DIAMETER: The diameter of a cylinder that bounds the crest of an external thread or the root of an internal thread.

MINOR DIAMETER: The diameter of a cylinder that bounds the root of an external thread or the crest of an internal thread.

HEIGHT OF THREAD: The distance from the crest to the root of a thread measured perpendicular to the axis of the threaded piece (also called straight depth of thread).

SLANT DEPTH: The distance from the crest to the root of a thread measured along the angle forming the side of the thread.

ALLOWANCE: An intentional difference between the maximum material limits of mating parts. It is the minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

TOLERANCE: The total permissible variation of a size. The tolerance is the difference between the limits of size.

THREAD FORM SERIES: Threads are made in many different shapes, sizes, and accuracies. When special threads are required by the product designer, he will specify in detail all the thread characteristics and their tolerances for production information. When a standard thread is selected, however, he needs only to specify size, number of threads per inch, designation of the standard series and the class of fit. With this specification, all other information necessary for production is obtained from the established standard, as published. The abbreviated designations for the different series are as follows:

Abbreviation	Full Title of Standard Series
UNC	Unified coarse thread series
UNF	Unified fine thread series
UNEF	Unified extra fine thread series
NC	American National coarse thread series
NF	American National fine thread series

Abbreviation	Full Title of Standard Series
NEF	American National extra fine thread series
UN	Unified constant pitch series including 4, 6, 8, 12, 16, 20, 28, and 32 threads per inch
NA	American National acme thread series
NPT	American National tapered pipe thread series
NPS	American National straight pipe thread series
NH	American National hose coupling thread series
NS	American National form thread-special pitch
N BUTT	National Buttress Thread

THREAD DESIGNATION: A thread is designated according to the nominal size, the number of threads per inch, the series symbol, and the class symbol, in that order. For example, the designation 1/4-20 UNC-3A is explained as follows:

1/4 = nominal thread diameter

20 = number of threads per inch

UNC = series (Unified coarse)

3 = class

A = external thread

Unless the designation LH (left hand) follows the class designation, the thread is assumed to be a right-hand thread. An example of the designation for a left-hand thread is: 1/4-20 UNC-3A-LH.

V-FORM THREADS

The three forms of V-threads that you must know how to machine are the V-sharp, the American National and The American Standard unified. All of these threads have a 60° included angle between their sides. The V-sharp thread has a greater depth than the others and the crest and root of this thread have little or no flat. The external American Standard unified thread has slightly less depth than the external American National thread but is otherwise similar. The American Standard unified thread is actually a modification of the American National thread. This modification was made so that the unified series of threads, which permits interchangeability of standard threaded fastening devices manufactured in the United States, Canada, and the United Kingdom, could be included in the threading system used in the United States. The Naval Sea Systems Command and naval procurement activities use American Standard unified threading system specifications whenever possible; this system is recommended for use by all naval activities.

To cut a V-form screw thread, you need to know (1) the pitch of the thread, (2) the straight depth of the thread, (3) the slant depth of the thread, and (4) the width of the flat at the root of the thread. The pitch of a thread is the basis for calculating all other dimensions and is equal to 1 divided by the number of threads per inch. The basis for determining the bore diameter is: minor diameter of the nut equals the major diameter of the externally threaded part minus 1.0825 divided by the number of threads per inch:

$$\text{Boresize} = \frac{1.0825}{N}$$

When the thread cutting tool is fed into the workpiece at one-half of the included angle of the thread, the slant-depth is the dimension necessary to determine how far to feed the tool into the work. The point of the threading tool must have a flat equal to the width of the flat at the root of the thread (external or internal thread, as applicable). If the flat at the point of the tool is too wide, the resulting thread will be too thin if the cutting tool is fed in the correct amount. If the flat is too narrow, the thread will be too thick.

The following formulas will provide the information you need for cutting V-form threads:

1. V-SHARP THREAD.

   Pitch $= \frac{1}{n}$ or 1 ÷ number of threads per inch

   Straight Depth of thread = 0.886 X pitch

2. AMERICAN NATIONAL THREAD.

   Pitch = 1 ÷ number of threads per inch or $\frac{1}{n}$

   Straight depth of external thread = 0.64952 X pitch or 0.64952p

   Straight depth of internal thread = 0.541266 X pitch or 0.541266p

   Width of flat at point of tool for external and internal threads = 0.125 X pitch or 0.125p

   Slant depth of external thread = 0.750 X pitch or 0.750p

   Slant depth of internal thread = 0.625 X pitch or 0.625p

3. AMERICAN STANDARD UNIFIED.

   Pitch = 1 ÷ number of threads per inch or $\frac{1}{n}$

   Straight depth of external thread = 0.61343 inch X pitch or 0.61343p

   Straight depth of internal thread 0.54127 inch X pitch = 0.54127p

   Width of flat at root of external thread = 0.125 inch X pitch or 0.125p

   Width of flat at crest of external threads = 0.125 inch X pitch or 0.125p

   Width of flat at root of internal thread = 0.125 inch X pitch or 0.125p

   Width of flat at crest of internal thread = 0.25 inch X pitch or 0.25p

   Double height of external thread = 1.22687 inch X pitch or 1.22687p

   Double height of internal thread = 1.08253 inch X pitch or 1.08253p

   Slant depth of external thread = 0.708 X pitch or 0.708p

   Slant depth of internal thread = 0.525 X pitch or 0.625p

(NOTE: MULTIPLYING the constant by the pitch, as in the preceding formulas, produces the same result as if you divide the constant by the number of threads per inch.)

To produce the correct thread profile, you must use a tool accurately ground to the correct angle and contour. Also, you must set the cutting tool in the correct position. Figure 9-10 shows how a tool must be ground and set.

You must grind the point of the tool to an angle of 60°, as shown in A of figure 9-10. Use a center gage or a thread-tool gage for grinding the tool to the exact angle required. The top of the tool is usually ground flat, with no side rake or back rake. However, for cutting threads in steel, side rake is sometimes used.

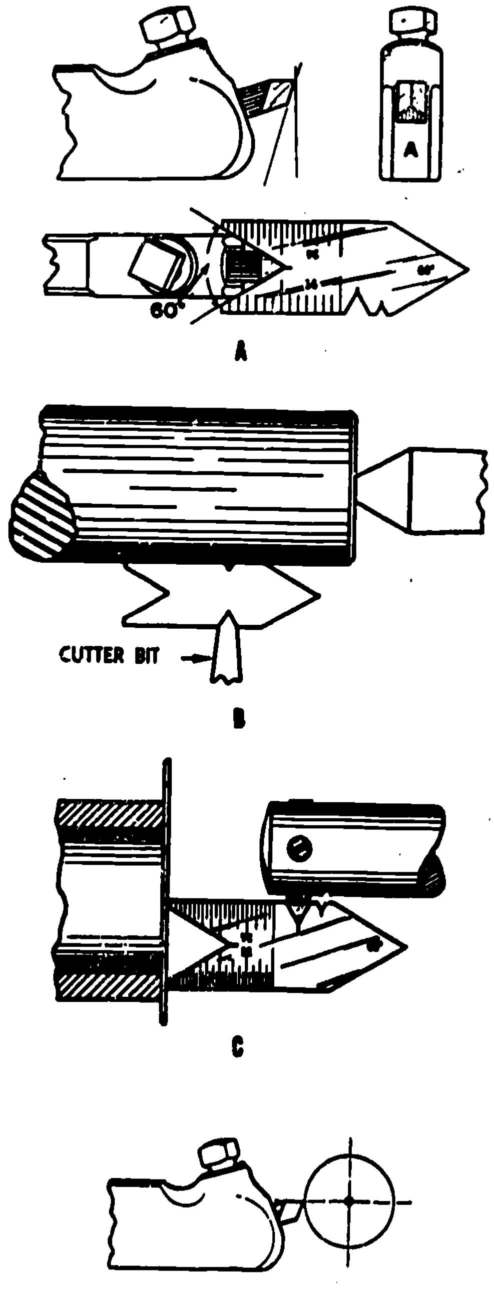
Set the threading tool square with the work, as shown in B and C of figure 9-10. Use the center gage to adjust the point of the threading tool; if you carefully set the tool, a perfect thread will result. If you do not set the threading tool perfectly square with the work, the angle of the thread will be incorrect.

For cutting external threads, place the top of the threading tool exactly on center as shown in D of figure 9-10. Note that the top of the tool is ground flat and is in exact alignment with the lathe center. This is necessary to obtain the correct angle of the thread.

Size of the threading tool for cutting an internal thread is important. The tool head must be small enough to be backed out of the thread and still leave enough clearance to be drawn from the threaded hole without injuring the thread. However, the boring bar which holds the threading tool for internal threading should be both as large as possible in diameter and as short as possible to prevent its springing away from the work during cutting.

OTHER FORMS OF THREADS

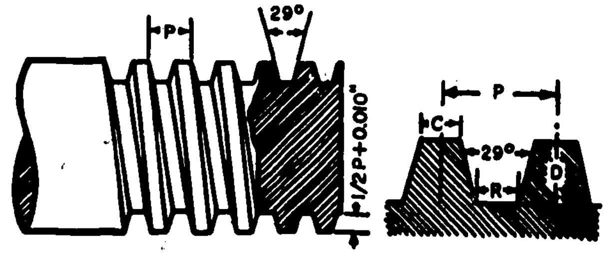
As stated previously, the cutting tool is ground to the form of the thread desired. In the following section Acme and square screw threads are illustrated with pertinent information on cutting these threads.



28.146X
Figure 9-10.—Threading tool setup for V-form threads.

The Acme Screw Thread

The Acme screw thread is used on valve stems, the lead screw of a lathe, and other places that require a strong thread. The top and bottom of the threads have an included angle of 29° (fig. 9-11).



FORMULAS

PITCH (P) = $\frac{1}{\text{No. Threads per inch}}$

DEPTH (D) = 1/2 P+0.010"

SLANT DEPTH = 0.516P+0.010"

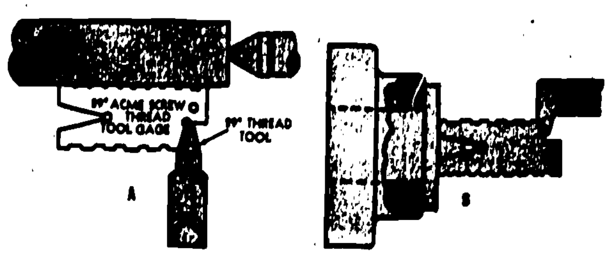
CREST (C) = 0.3707 P

ROOT (R) = 0.3707 P−0.0052"

28.147X
Figure 9-11.—Acme thread and formulas for cutting.

Parts A and B of figure 9-12 show the method of setting an Acme threading tool for cutting an external and internal Acme thread, respectively. Note that a 29° Acme thread gage is used in the same manner as the center gage was used for V-form screw threads. Adjust the cutting edge of the tool to line it up exactly with the beveled edge of the gage. The notches in the Acme thread gage let you grind the squared front edge of the tool bit accurately in accordance with the pitch of the thread to be machined.

In cutting an Acme thread, be sure the clearance is 0.010 inch between the top of the



28.148X
Figure 9-12.—Use of Acme thread tool gage.

thread of the screw and the bottom of the thread of the nut in which it fits.

The Square Thread

The square thread (fig. 9-13) is used when heavy threads are required, such as in jack screws, press screws and feed screws. It is used for much the same purpose as the Acme thread, which is used in many places where the square thread was formerly used. The disadvantage of square threads is that the straight sides do not allow sideplay adjustment.

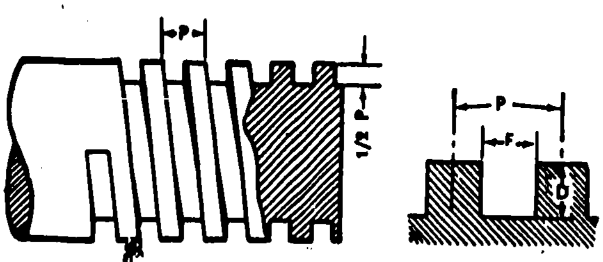
The cutting edge width of the tool for cutting square screw threads is exactly one-half the pitch, but the width of the edge of the tool for threading nuts is from 0.001 to 0.003 inch larger. This permits a sliding fit on the screw.

Set the threading tool for cutting square threads square with the work. Since the edge of the tool is square, you need to adjust only the edge to the surface of the work.

Be sure the clearance between the top of the thread of the screw and that of the bottom of the thread of the nut is about 0.005 to 0.008 inch for each inch of thread diameter.

BUTTRESS THREAD

On a buttress thread (fig. 9-14) the load resisting side is nearly perpendicular to the thread axis and is called the pressure flank. The



FORMULAS

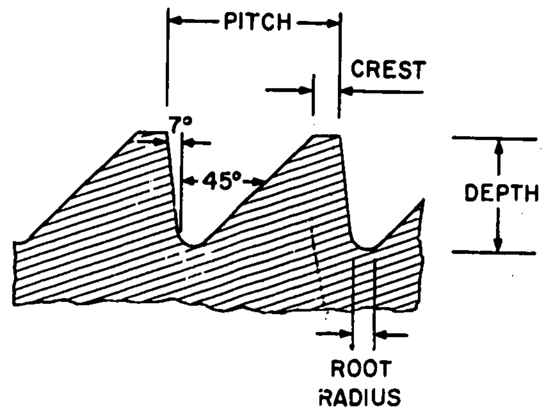
PITCH (P) = $\frac{1}{\text{No. Threads per Inch}}$

DEPTH (D) = P x 0.500" + 0.005" to 0.008"

SPACE (F) = P x 0.500"

28.149X

Figure 9-13.—Square thread and formulas.



28.350

Figure 9-14.—Buttress thread.

American Standard form of the buttress thread has a 7° angle on the pressure flank; other forms have 0°, 3°, or 5°. However, the American Standard form is most often used, and the formulas in this section apply to this form. The buttress thread can be designed to either push or pull against the internal thread of the mating part into which it is served. The direction of this thrust will determine the way you grind your tool for machining the thread. An example of the designation symbols for an American Standard Buttress thread form are as follows:

6 - 10 (←N BUTT-2

where 6 = basic major diameter of 6.000 inches

10 = 10 threads per inch

(← = internal member to push against external member

N BUTT = National Buttress Form

2 = class of fit

NOTE: A symbol such as "←(" indicates that the internal member is to pull against the external member.

The fomulas for the basic dimensions of the American Standard Buttress external thread are as follows:

$$\text{Pitch} = \frac{1}{n}$$

Width of flat at crest = 0.1631 X pitch

Root radius = 0.0714 X pitch

Depth of thread = 0.6627 X pitch

The classes of fit are: 1 = free, 2 = medium, 3 = close. The specific dimensions involved apply to the tolerance to the pitch diameter and the major diameter and vary according to the nominal or basic size. Consult a handbook for specific information on the dimensions for the various classes of fit.

PIPE THREADS

American National Standard Pipe threads are similar to the unified threads in that both have an included angle of 60° and a flat on the crest and the root of the thread. Pipe threads can be either tapered or straight, depending on the intended use of the threaded part. A description of the two types is given in the following paragraphs.

TAPERED PIPE THREADS

Tapered pipe threads are used to provide a pressure tight joint when the internal and external mating parts are assembled correctly. Depending on the closeness of the fit of the mating parts, you may need to use a sealer (pipe compound) to prevent leakage at the joint. The taper of the threads is 3/4 inch per foot. Machine and thread the section of pipe to be threaded at this angle. The hole for the internal threads should be slightly larger than the minor diameter of the small end of the external threaded part.

The designation of pipe threads is expressed as follows:

NPT 1/4-18

where NPT = tapered pipe thread

1/4 = inside diameter of the pipe in inches

18 = threads per inch

Figure 9-15 shows the typical dimensions of the most common tapered pipe threads.

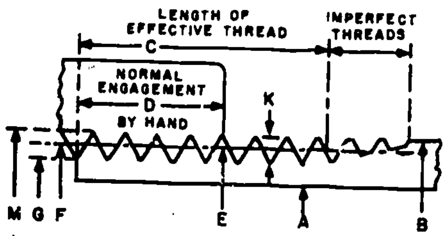
STRAIGHT PIPE THREADS

Straight pipe threads are similar in form to the tapered pipe thread except that they are not tapered. The same nominal outside diameter and thread dimensions apply. Straight pipe threads are used for mechanically joining components and are not satisfactory for high-pressure applications. Sometimes a straight pipe thread is used with a tapered pipe thread to form a low-pressure seal in a vibration free environment.

CLASSES OF THREADS

Classes of fit for threads are determined by the amount of tolerance and allowance allowed for each particular class. The tolerance (amount that a thread may vary from the basic dimension) decreases as the class number increases. As an example, a class 1 thread has more tolerance than a class 3 thread. The pitch diameter of the thread is the most important thread element in controlling the class of fit. The major diameter for an external thread and the minor diameter or bore size for an internal thread are also important, however, since they control the crest and root clearances more than the actual fit of the thread. A brief description of the different classes of fit are as follows:

- Classes 1A and 1B: Class 1A for external threads and 1B for internal threads are used on threaded components where quick and easy assembly is necessary and where a liberal



FOR ALL DIMENSIONS SEE CORRESPONDING REFERENCE LETTERS BELOW.

ANGLE BETWEEN SIDES OF THREAD IS 60°. TAPER OF THREAD, ON DIAMETER, IS 3/4 INCH PER FOOT.

THE BASIC THREAD DEPTH IS 0.8 X PITCH OF THREAD AND THE CREST AND ROOT ARE TRUNCATED AN AMOUNT EQUAL TO 0.039 X PITCH, EXCEPTING 8 THREADS PER INCH WHICH HAVE A BASIC DEPTH OF 0.788 X PITCH AND ARE TRUNCATED 0.045 X PITCH AT THE CREST AND 0.033 X PITCH AT THE ROOT.

PIPE SIZE		NUMBER OF THREADS PER INCH	PITCH DIAMETER		LENGTH OF EFFECTIVE THREAD	LENGTH OF HAND-TIGHT ENGAGEMENT	IMPERFECT THREADS	DEPTH OF THREAD (MAX.)	PITCH OF THREAD	MINOR DIAM AT SMALL END	MAJOR DIAM AT SMALL END
NOMINAL PIPE SIZE	OUTSIDE DIAMETER		AT END OF EXTERNAL THREAD	AT END OF INTERNAL THREAD							
A	B		F	E	C	D		K		G	H
1/8	0.405	27	0.36351	0.37476	0.2638	0.180	0.1285	0.02963	0.03704	0.334	0.39
1/4	0.540	18	0.47739	0.48989	0.4018	0.200	0.1928	0.04444	0.05556	0.433	0.52
3/8	0.675	18	0.61201	0.62701	0.4078	0.240	0.1928	0.04444	0.05556	0.568	0.65
1/2	0.840	14	0.75843	0.77843	0.5337	0.320	0.2478	0.05714	0.07143	0.701	0.81
3/4	1.050	14	0.96768	0.98887	0.5457	0.339	0.2478	0.05714	0.07143	0.911	0.02
1	1.315	11½	1.21363	1.23863	0.6828	0.400	0.3017	0.06957	0.08696	1.144	1.28

28.351

Figure 9-15.—Taper pipe thread dimensions.

allowance is required to permit ready assembly, even with slightly bruised or dirty threads.

- Classes 2A and 2B: Class 2A for external threads and 2B for internal threads are the most commonly used thread standards for general applications including production of bolts, screws, nuts and similiar threaded fasteners.

- Classes 3A and 3B: Class 3A for external threads and 3B for internal threads provide for applications where closeness of fit and accuracy of lead and angle of thread are important. They are obtainable consistently only by high quality production methods, with a very efficient system of gaging and inspection.

Tables of the basic dimensions and the maximum and minimum dimensions for each size and class of fit of threads are found in most publications and handbooks for machinists. An example of the dimensions required to accurately machine a specific class of fit on a thread is shown in Table 9-2.

MEASURING SCREW THREADS

Thread measurement is needed to ensure that the thread and its mating part will fit properly. It is important that you know the various measuring methods and the calculations

Table 9-2.—Classes of Fit and Tolerances for 1/4-20 UNC Thread

1/4-20 UNIFIED SCREW THREAD (EXTERNAL)

Designation	Basic Major Diameter	Maximum Major Diameter	Minimum Major Diameter	Basic Pitch Diameter	Maximum Pitch Diameter	Minimum Pitch Diameter
1/4-20 UNC-1A	0.250	0.2489	0.2367	0.2175	0.2164	0.2108
1/4-20 UNC-2A	0.250	0.2489	0.2408	0.2175	0.2164	0.2127
1/4-20 UNC-3A	0.250	0.2500	0.2419	0.2175	0.2175	0.2147

1/4-20 UNIFIED SCREW THREAD (INTERNAL)

Designation	Basic Minor Diameter (Bore Size)	Maximum Minor Diameter (Bore Size)	Minimum Minor Diameter (Bore Size)	Basic Pitch Diameter	Maximum Pitch Diameter	Minimum Pitch Diameter
1/4-20 UNC-1B	0.1876	0.196	0.207	0.2175	0.2248	0.2175
1/4-20 UNC-2B	0.1876	0.196	0.207	0.2175	0.2223	0.2175
1/4-20 UNC-3B	0.1887	0.196	0.2067	0.2175	0.2211	0.2175

28.352

which are used to determine the dimensions of threads.

The use of a mating part is common practice when average accuracy is required. The thread is simply machined until the mating part will assemble. A snug fit is usually desired with very little, if any, play between the parts.

You will sometimes be required to machine threads that need a specific class of fit, or you may not have the mating part to use as a gage. In these cases, you must use a method of measuring the thread to make sure you get the required fit.

An explanation of the various methods normally available to you is given in the following paragraphs:

THREAD MICROMETER

Thread micrometers are used to measure the pitch diameter of threads. They are graduated and read in the same manner as ordinary micrometers. However, the anvil and spindle are ground to the shape of a thread, as shown in figure 9-16. Thread micrometers come in the same size ranges as ordinary micrometers: 0 to 1 inch, 1 to 2 inches, etc. In addition, they are available in various pitch ranges. The number of threads per inch must be within the pitch range of the thread.

RING AND PLUG GAGES

Go and no-go gages, such as those shown in figure 9-17, are often used to check threaded parts. The thread should fit the "go" portion of the gage, but should not screw into or onto the "no-go" portion. Ring and plug gages are available for the various sizes and classes of fit of thread. They are probably the most accurate method of checking threads because they envelop the total thread form, and in effect, check not only the pitch diameter, major and minor diameter, but also the lead of the thread.

THREE WIRE METHOD

The pitch diameter of a thread can be accurately measured by an ordinary micrometer and three wires, as shown in figure 9-18.

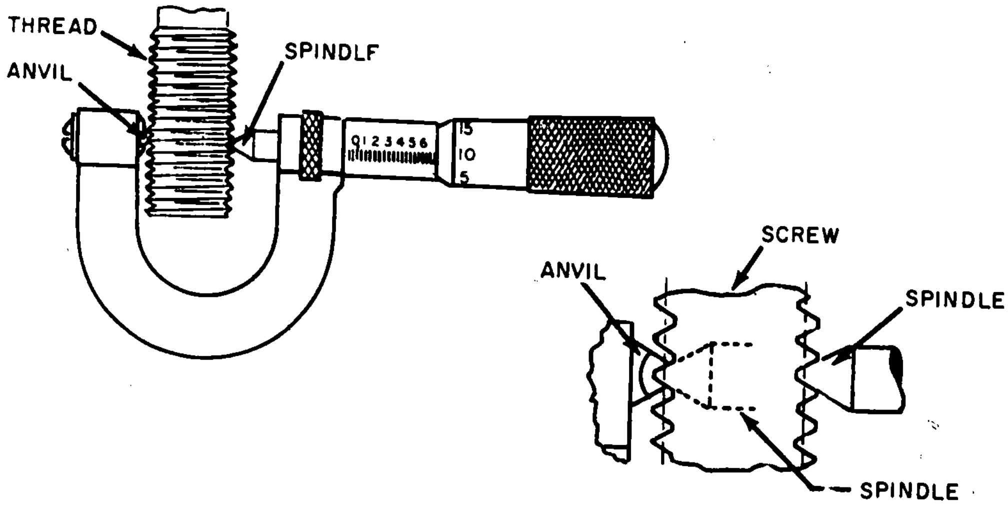


Figure 9-16.–Measuring threads with a thread micrometer. 28.353

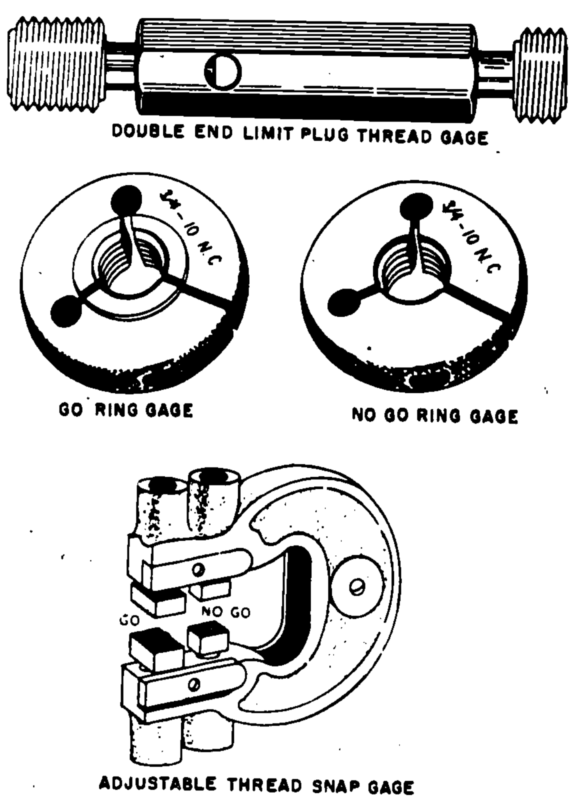


Figure 9-17.–Thread gages. 28.354

The wire size you should use to measure the pitch diameter depends upon the number of threads per inch. The most accurate results are obtained when you use the "best wire size." The best size is not always available, but you will get satisfactory results if you use wire diameters

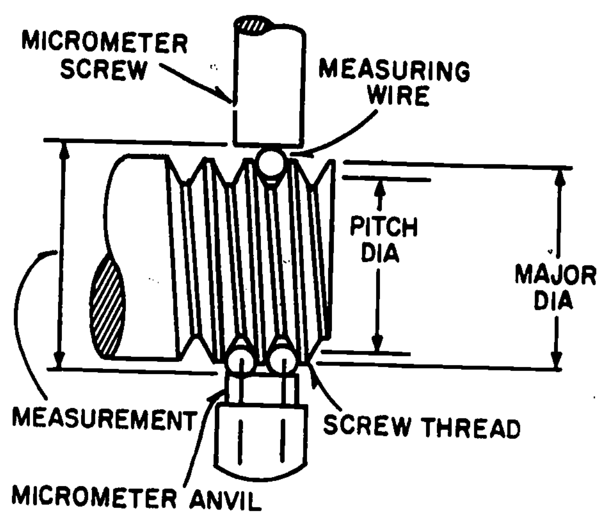


Figure 9-18.–Measuring threads using three wires. 28.355

within a given range. Use a wire size as close as possible to the best wire size. You can use these formulas:

Best wire size	= 0.57735	inch X pitch
Smallest wire size	= 0.56	inch X pitch
Largest wire size	= 0.90	inch X pitch

For example, the diameter of the best wire for measuring a thread that has 10 threads per inch is 0.577 inch, but you could use any size between 0.056 inch and 0.090 inch.

NOTE: The wires should be fairly hard and uniform in diameter. All three wires must be the same size. You can use the shanks of drill bits as substitutes for the wires.

Use the following formulas to determine what the measurement over the wires should be for a given pitch diameter.

Measurement = pitch diameter - (0.86603 X pitch) + (3 X wire diameter)

M = PD - (0.86603 X P) + (3 X W)

Use the actual size of the wires in the formula, not the calculated size.

Example: What should the measurement be over the wires for a 3/4-10 UNC-2A? First, you must determine the required pitch diameter for a class 2A 3/4-10 UNC thread. You can find this information in charts in several handbooks for machinists. The limits of the pitch diameter for this particular thread size and class are between 0.6832 and 0.6773 in. Use the maximum size (0.6832 inch) for this example. Next, you must calculate the pitch for 10 threads per inch. The formula, "one divided by the number of threads per inch" will give you pitch = $\frac{1}{n}$. For 10 TPI, the pitch is 0.100 in. As previously stated, the best wire size for measuring 10 TPI is 0.577 in., so assume that you have this wire size available.

Now make the calculation. The data collected so far are:

Thread = 3/4-10 UNC - 2A

Pitch diameter (PD) = 0.6832 in.

Pitch (P) = 0.100 in.

Wire size (W) = 0.0577 in.

The standard formula for the measurement over the wires was M = PD - (0.86603 X P) + (3 X W). Enter the collected data in the correct position of the formula:

M = 0.6832 in. - (0.86603 in. X 0.100 in.) + (3 X 0.577 in.)

M = 0.6832 in. - 0.086603 in. + 0.1731 in.

M = 0.769697 in.

The measurement over the wires should be 0.769697 in. or when rounded to four decimal places, 0.7697 in.

As mentioned in the beginning of the section on classes of threads, the major diameter is a factor also considered in each different class of fit. The basic or nominal major diameter is seldom the size actually machined on the outside diameter of the part to be threaded. The actual size is smaller than the basic size. In the case of the 3/4 - 10 UNC - 2A thread, the basic size is 0.750 in., however, the size that the outside diameter should be machined is between 0.7482 and 0.7353 in.

HOW TO CUT SCREW THREADS ON A LATHE

Screw threads are cut on the lathe by connecting the headstock spindle of the lathe with the lead screw by a series of gears to get a positive carriage feed, and the lead screw is driven at the required speed in relation to the headstock spindle. You can arrange the gearing between the headstock spindle and lead screw so that you can cut any desired pitch of the thread. For example, if the lead screw has 8 threads per inch and you arrange the gears so that the headstock spindle revolves four times while the lead screw revolves once, the thread you cut will be four times as fine as the thread on the lead screw, or 32 threads per inch. With the quick-change gear box, you can quickly and easily make the proper gearing arrangement by placing the levers as indicated on the index plate for the thread desired.

When you have the lathe set up to control the carriage movement for cutting the desired thread pitch, your next consideration is shaping the thread. Grind the cutting tool to the shape required for the form of the thread to be cut, that is–V, Acme, square, etc. Adjust the cross-slide to get the depth of the thread.

Mounting Work in the Lathe

When mounting work between lathe centers for cutting screw threads, be sure the lathe dog is securely attached before starting to cut the thread. If the dog should slip, the thread will be ruined. Do not remove the lathe dog from the work until you have completed the thread. If you must remove the work from the lathe before the thread is completed, be sure to replace the lathe dog in the same slot of the driving plate.

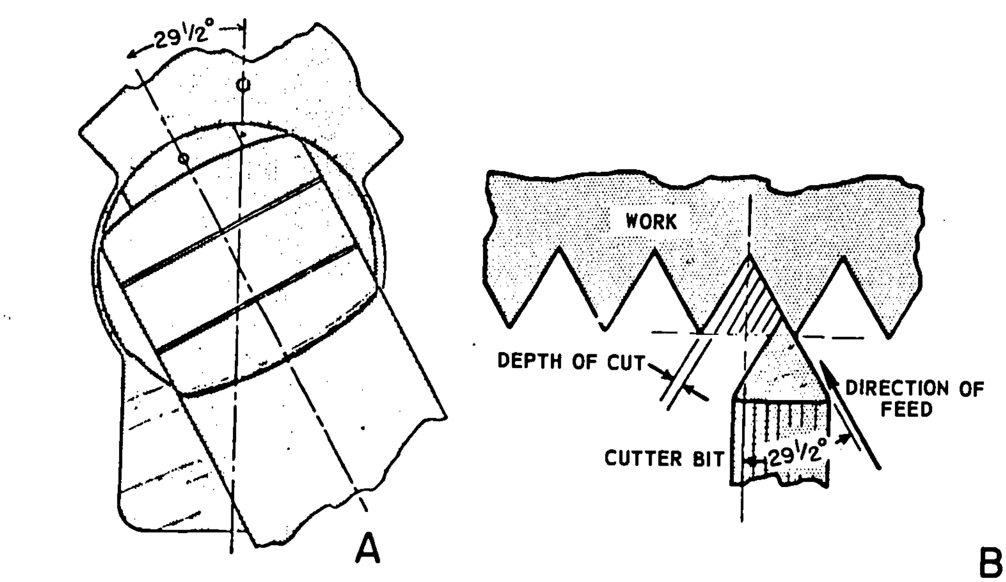
When threading work in the lathe chuck, be sure the chuck jaws are tight and the work is well supported. Never remove the work from the chuck until the thread is finished.

When threading long slender shafts, use a follower rest. You must use the center rest to support one end of long work that is to be threaded on the inside.

Position of Compound Rest for Cutting Screw Threads

Ordinarily on threads of fine lead, you feed the tool straight into the work in successive cuts. For coarse threads, it is better to set the compound rest at one-half of the included angle of the thread and feed in along the side of the thread. For the last few finishing cuts, you should feed the tool straight in with the crossfeed of the lathe to make a smooth, even finish on both sides of the thread.

In cutting V-form threads and when maximum production is desired, it is customary to place the compound rest of the lathe at an angle of 29 1/2°, as shown in Part A of figure 9-19. When you set the compound rest in this position and use the compound rest screw to



28.150X

Figure 9-19.–Compound rest set at 29 1/2°.

250

adjust the depth of cut, you remove most of the metal by the left side of the threading tool (B of fig. 9-19). This permits the chip to curl out of the way better than if you feed the tool straight in, and it prevents tearing of the thread. Since the angle on the side of the threading tool is 30°, the right side of the tool will shave the thread smooth and produce a better finish; although it does not remove enough metal to interfere with the main chip, which is taken by the left side of the tool.

Using the Thread-Cutting Stop

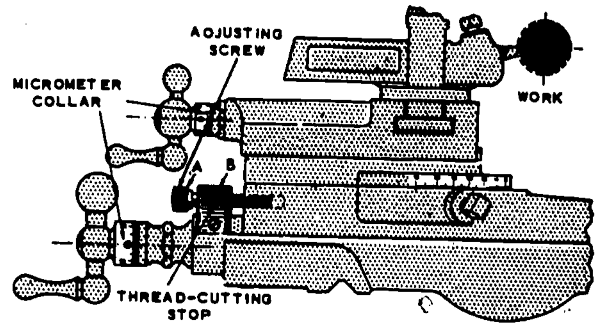
Because of the lost motion caused by the play necessary for smooth operation of the change gears, lead screw, half-nuts, etc, you must withdraw the thread-cutting tool quickly at the end of each cut. If you do not withdraw quickly, the point of the tool will dig into the thread and may break off.

To reset the tool accurately for each successive cut and to regulate the depth of the chip, use the thread-cutting stop.

First, set the point of the tool so that it just touches the work, then lock the thread-cutting stop by turning the thread-cutting stop screw A (fig. 9-20) until the shoulder is tight against stop B (fig. 9-20). When you are ready to take the first chip, run the tool rest back by turning the crossfeed screw to the left several times, and move the tool to the point where the thread is to start. Then, turn the crossfeed screw to the right until the thread-cutting stop screw strikes the thread-cutting stop. The tool is now in the original position. By turning the compound rest feed screw in 0.002 inch or 0.003 inch, you will have the tool in a position to take the first cut.

For each successive cut after returning the carriage to its starting point, you can reset the tool accurately to its previous position. Turn the crossfeed screw to the right until the shoulder of screw A strikes stop B. Then, you can regulate the depth of the next cut by adjusting the compound rest feed screw as it was for the first chip.

For cutting an internal thread, set the adjustable thread-cutting stop with the head of the adjusting screw on the inside of the stop.



28.151X

Figure 9-20.—Adjustable thread-cutting stop mounted on carriage saddle (clamped to dovetail).

Withdraw the tool by moving it toward the center or axis of the lathe.

You can use the micrometer collar on the crossfeed screw in place of the thread-cutting stop, if desired. To do this, first bring the point of the threading tool up so that it just touches the work; then adjust the micrometer collar on the crossfeed screw to zero. All adjustments for obtaining the desired depth of cut should be made with the compound rest screw. Withdraw the tool at the end of each cut by turning the crossfeed screw to the left one complete turn; return the tool to the starting point, and turn the crossfeed screw to the right one turn, stopping at zero. You can then adjust the compound rest feed screw for any desired depth.

Engaging the Thread Feed Mechanism

When cutting threads on a lathe, clamp the half-nuts over the lead screw to engage the threading feed and release at the end of the cut by means of the threading lever. Use the threading dial (discussed in chapter 7 of this training course and illustrated in fig. 7-37) to determine when to engage the half-nuts so that the cutting tool follows the same path during each cut. When an index mark on the threading dial aligns with the witness mark on its housing, engage the half-nuts. For some thread pitches, however, you can engage the half-nuts only when certain index marks are aligned with the witness mark. On most lathes you can engage the half-nuts as follows:

For all even-numbered threads per inch, close the half-nuts at any line on the dial.

For all odd-numbered threads per inch, close the half-nuts at any numbered line on the dial.

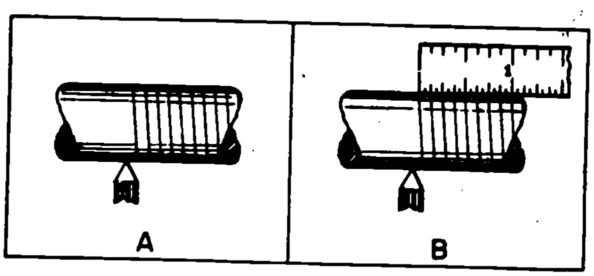
For all threads involving one-half of a thread in each inch, such a 11 1/2, close the half-nuts at any odd-numbered line.

Cutting the Thread

After setting up the lathe, as explained previously, take a very light trial cut just deep enough to scribe a line on the surface of the work, as shown in A of figure 9-21. The purpose of this trial cut is to be sure that the lathe is arranged for cutting the desired pitch of thread.

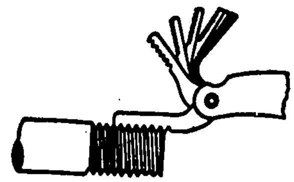
To check the number of threads per inch, place a rule against the work, as shown in B of figure 9-21, so that the end of the rule rests on the point of a thread or on one of the scribed lines. Count the scribed lines between the end of the rule and the first inch mark. This will give the number of threads per inch.

It is quite difficult to accurately count fine pitches of screw threads. A screw pitch gage, used as illustrated in figure 9-22, is very convenient for checking the finer screw threads. The gage consists of a number of sheet metal plates in which are cut the exact form of threads of the various pitches; each plate is stamped with a number indicating the number of threads per inch for which it is to be used.



28.152X

Figure 9-21.—The first cut.



28.153X

Figure 9-22.—Screw pitch gage.

Make the final check for both diameter and pitch of the thread with the nut that is to be used or with a ring thread gage, if one is available. The nut should fit snugly without play or shake but should not bind on the thread at any point.

Lubricants for Cutting Threads

To produce a smooth thread in steel, use lard oil as a lubricant. If oil is not used, the cutting tool will tear the steel, and the finish will be very rough.

If lard oil is unavailable, use any good cutting oil or machine oil. If you experience trouble in producing a smooth thread, add a little powdered sulfur to the oil.

Apply the oil generously before each cut. A small paint brush is ideal for applying the oil when cutting external screw threads. Since lard oil is quite expensive, many machinists place a small tray or cup just below the cutting tool on the lathe cross-slide to catch the surplus oil which drips off the work.

Resetting the Tool or Picking Up the Existing Thread

If the thread-cutting tool needs resharpening or gets out of alignment or if you are chasing the threads on a previously threaded piece, you must reset the tool so that it will follow the original thread groove. To reset the tool, (1) use the compound rest feed screw and crossfeed screw to jockey the tool to the proper position, (2) disengage the change gears and turn the spindle until the tool is positioned properly, or (3) loosen the lathe dog (if used) and turn the work until the tool is in proper position with the thread groove. Regardless of which method you use, you will usually have to reset the micrometer collars on the crossfeed screw and the compound rest screw.

Before adjusting the tool in the groove, use the appropriate thread gage to set the tool square with the workpiece. Then with the tool a few thousandths of an inch away from the workpiece, start the machine and engage the threading mechanism. When the tool has moved to a position such as shown in figure 9-23, stop the lathe without disengaging the thread mechanism.

The most practical and commonly used method for resetting a threading tool for machining angular form threads is the compound rest and crossfeed positioning method. By adjusting the compound rest slide forward or backward, you can move the tool laterally to the axis of the work as well as toward or away from the work. When the point of the tool coincides with the original thread groove (see phantom view of tool in fig. 9-23), use the crossfeed screw to bring the tool point directly into the groove. When you get a good fit between the cutting tool and thread groove, set the micrometer collar on the crossfeed screw on zero and set the micrometer collar on the compound rest feed screw to the depth of cut previously taken or to zero, as required. (Note: Be sure that the thread mechanism is engaged and the tool is set square with the work before adjusting the position of the tool along the axis of the workpiece.)

If it is inconvenient to use the compound rest for readjusting the threading tool, loosen the lathe dog (if used); turn the work so that the threading tool will match the groove, and tighten the lathe dog. If possible, however, avoid the necessity of doing this.

Another method, which is sometimes used, is to disengage the reverse gears or the change gears; turn the headstock spindle until the point of the threading tool enters the groove in the work, and then engage the gears.

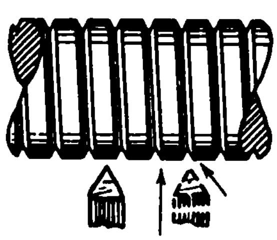
Finishing the End of a Threaded Piece

The end of a thread may be finished by any one of several methods. The 45° chamfer on the end of a thread, as shown in A of figure 9-24, is commonly used for bolts and capscrews. For machined parts and special screws, the end is often finished by rounding with a forming tool, as shown in B of figure 9-24.

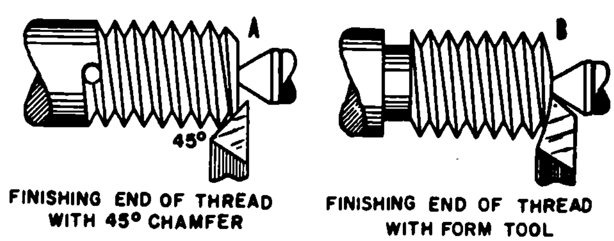
It is difficult to stop the threading tool abruptly, so some provision is usually made for clearance at the end of the cut. In A of figure 9-24, a hole has been drilled at the end of the thread; in B of figure 9-24, a neck or groove has been cut around the shaft. The groove is preferable, as the lathe must be run very slowly to obtain satisfactory results with the drilled hole.

LEFT-HAND SCREW THREADS

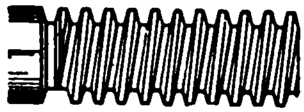
A left-hand screw (fig. 9-25) turns counterclockwise when advancing (looking at the head of the screw), or just the opposite to a right-hand screw. Left-hand threads are used for the crossfeed screws of lathes, the left-hand end



28.154X
Figure 9-23.—Tool must be reset to original groove.



28.155X
Figure 9-24.—Finishing the end of a threaded piece.

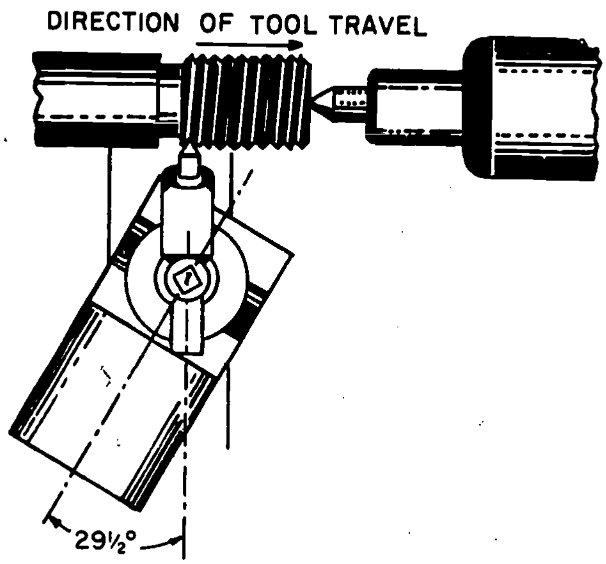


28.156X
Figure 9-25.–A left-hand screw thread.

of axles, one end of a turnbuckle, or wherever an opposite thread is desired.

For cutting a left-hand thread on a lathe, the directions are the same as for cutting a right-hand thread, except you swivel the compound rest to the left instead of to the right. Figure 9-26 shows the correct position for the compound rest. The direction of travel for the tool differs from a right-hand thread in that it moves toward the tailstock as the thread is being cut.

Before starting to cut a left-hand thread, it is good practice, if feasible, to cut a neck or groove in the workpiece. (See fig. 9-25.) Such a groove facilitates running the tool in for each pass, in the same manner as for a right-hand thread.

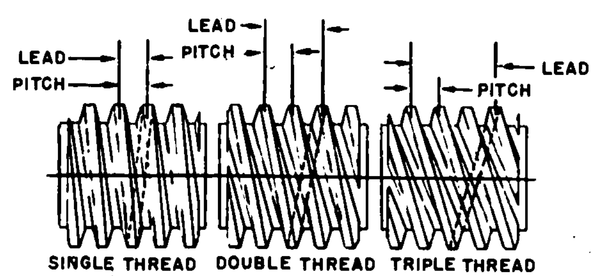


28.356
Figure 9-26.–Setup for left-hand external threads.

MULTIPLE SCREW THREADS

A multiple thread, as shown in figure 9-27, is a combination of two or more threads, parallel to each other, progressing around the surface into which they are cut. If a single thread is thought of as taking the form of a helix, that is, of a string or cord wrapped around a cylinder, a multiple thread may be thought of as several cords lying side by side and wrapped around a cylinder. There may be any number of threads, and they start at equally spaced intervals around a cylinder. Multiple threads are used when rapid movement of the nut or other attached parts is desired and when any weakening of the thread is to be avoided. A single thread having the same lead as a multiple thread would be very deep in comparison to the multiple thread. The depth of the thread is calculated according to the pitch of the thread.

The tool selected for cutting multiple threads has the same shape as that of the thread to be cut and is similar to the tool used for cutting a single thread except that greater side clearance is necessary. The helix angle of the thread increases with an increase in the multiple thread. The general method for cutting multiple threads is about the same as for single screw threads, except that the lathe must be geared to the number of single threads per inch, or with reference to the lead of the thread, and not the pitch, as shown in figure 9-27. Provisions must also be made to obtain the correct spacing of the different thread grooves. You can get the proper spacing by using the thread-chasing dial, setting the compound parallel to the ways, using a faceplate, or using the change gear break up.



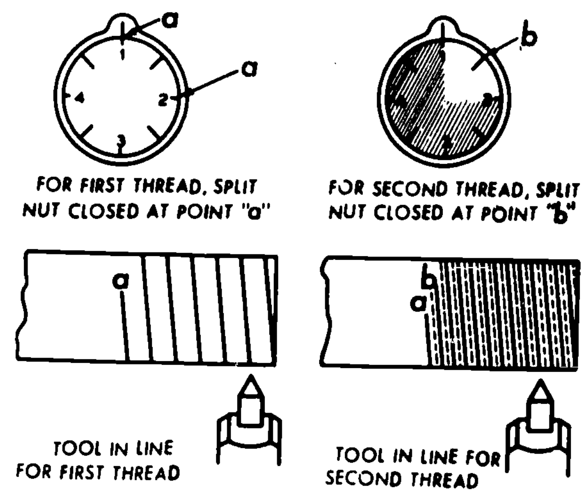
28.357
Figure 9-27.–Comparison of single and multiple-lead threads.

The use of the thread-chasing dial is the most desirable method for cutting 60° multiple threads. With each setting for depth of cut with the compound, you can take successive cuts on each of the multiple threads so that you can use thread micrometers.

To explore the possibility of using the thread-chasing dial, you must first find out if the lathe can be geared to cut a thread having a lead equal to that of one of the multiple threads. For example, if you want to cut 10 threads per inch, double threaded, you divide the number of threads per inch by the multiple (in this case 10/2) to get the number of single threads per inch (in this case 5). Then you gear the lathe for the number of single threads per inch.

To use the thread-chasing dial on a specific machine, you should refer to instructions usually found attached to the lathe apron. If, for 5 threads per inch, you should engage the half-nut at any numbered line on the dial, the same thread would be cut at positions 1 and 2 on the dial, as shown in figure 9-28. If you then cover the dial with your hand, leaving the part uncovered between those adjacent positions that cut the groove (positions 1 and 2 in figure 9-28), make a check to see if there is a point of engagement midway between positions 1 and 2 for the second thread. The second groove of a double thread lies midway of the flat surface between the grooves. There is a point of engagement in this case, position "b" in figure 9-28. For the same depth of cut, engage the half-nut first at one of the "a" positions, then at "b" position so that alternate cuts bring both thread grooves down to size together. In the event positions 1 and 2 indicate the engagement place for groove of a triple thread, you must have two positions of engagement, equally spaced, between positions 1 and 2 in order to cut the other grooves of the triple thread.

Cutting of multiple threads by positioning the compound parallel to the ways should be limited to square and Acme external and internal threads. Set the compound rest parallel to the ways of the lathe and cut the first thread to the finished size. Then feed the compound and tool forward parallel to the thread axis a distance equal to the pitch of the thread and cut the next thread, etc. You can cut any desired multiple threads in this manner if you gear the lathe to the lead of the multiple thread.

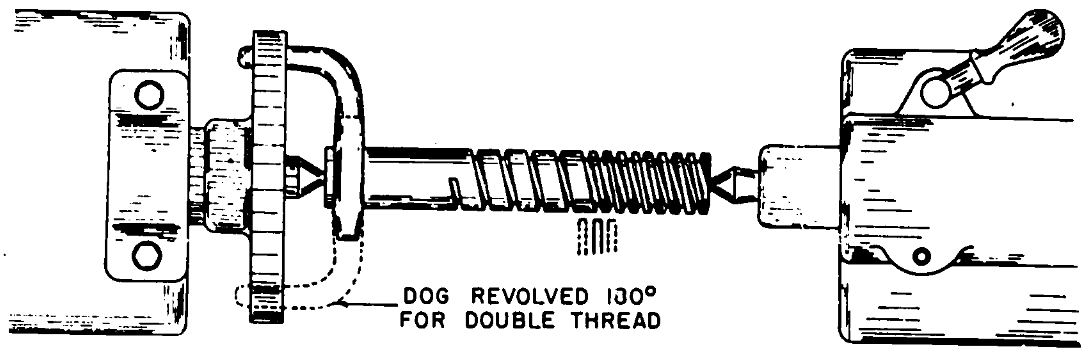


28.358

Figure 9-28.—Cutting multiple threads using the thread-chasing dial.

The faceplate method of cutting multiple threads involves changing the position of the work between centers for each groove of the multiple thread. One method is to cut the first thread groove in the conventional manner. Then, remove the work from between centers and replace it with the tail of the dog in another slot of the drive plate, as shown in figure 9-29. Two slots are necessary for a double thread, three slots for a triple thread, etc. The number of multiples you can cut by this method depends upon the number of equally spaced slots in the driveplate. There are special drive or index plates so that you can accurately cut a wide range of multiples by this method.

Another method of cutting multiple threads is to disengage either the stud or spindle gear from the gear train in the end of the lathe after you cut a thread groove. Then turn the work and spindle the required part of a revolution, and reengage the gears for cutting the next thread. If you are to cut a double thread on a lathe that has a 40-tooth gear on the spindle, cut the first thread groove in the ordinary manner. Then mark one of the teeth on the spindle gear that meshes with the next driven gear. The mark is carried onto the driven gear, in this case the reversing gear. Also mark the tool diametrically opposite the marked spindle gear tooth (the 20th tooth of the 40-tooth gear). Count the



28.359
Figure 9-29.—Use of face plate.

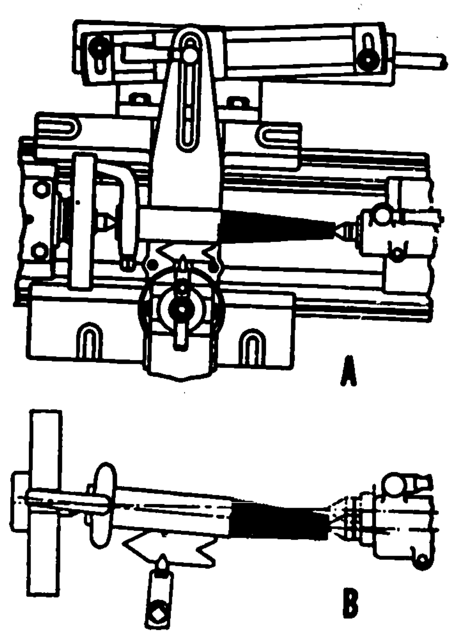
tooth next to the marked tooth as tooth number one. Then disengage the gears by placing the tumbler (reversing) gears in neutral position, turn the spindle one-half revolution or 20 teeth on the spindle gear, and reengage the gear train. The stud gear may be indexed as well as the spindle gear. However, if the lathe does not have a 1 to 1 ratio between the spindle and stud gears, the stud gear instead of being turned as when geared for a 1 to 1 ratio would be given a proportional turn depending upon the ratio of the gearing. The method of indexing the stud or spindle gears is possible only when you can evenly divide the number of teeth in the gear indexed by the multiple desired. Some of the newer type lathes have a sliding sector gear that you can readily engage or disengage with the gear train by shifting a lever. Graduations on the end of the spindle show when to disengage and to reengage the sector gear for cutting various multiples.

THREADS ON TAPERED WORK

The taper attachment should be used when you cut a thread on tapered work. If there is no taper attachment with the lathe, cut the thread on tapered work by setting over the tailstock. The setup is the same as for turning tapers.

Part A of figure 9-30 shows the method of setting the threading tool with the thread gage when you use the taper attachment. Part B of figure 9-30 shows the same operation for using the tailstock setover method.

Note that in both methods illustrated in figure 9-30, you set the threading tool square with the axis by placing the center gage on the straight part of the work, NOT on the tapered section. This is important.



28.158
Figure 9-30.—Cutting thread on tapered work.

CHAPTER 10

TURRET LATHES AND TURRET LATHE OPERATIONS

Horizontal and vertical turret lathes are generally used to produce several identical workpieces. Because turret lathes are designed for production work, they have many automatic features that are not found on engine lathes. For greatest efficiency, a turret lathe must be set up so the operator can perform the machining steps with a minimum amount of control.

In this chapter we shall discuss turret lathes and some of the important factors in the tooling setup.

TURRET LATHE SAFETY

Before learning to operate a turret lathe, you must realize the importance of observing safety precautions. As in all machine operations, you have one guideline: SAFETY FIRST, ACCURACY SECOND, AND SPEED LAST. The safety precautions listed in chapter 8 for engine lathes apply also to turret lathes. Listed below are additional safety precautions that you must observe to safely operate both horizontal and vertical turret lathes.

- Only authorized and qualified personnel will operate turret lathes.

- Wear goggles or a face shield whenever you operate a turret lathe.

- Be sure that long stock extending from the turret lathe is properly guarded and supported.

- Be aware of tools mounted on the turret heads. If you are not careful they will strike you when the turrets rotate to a new station.

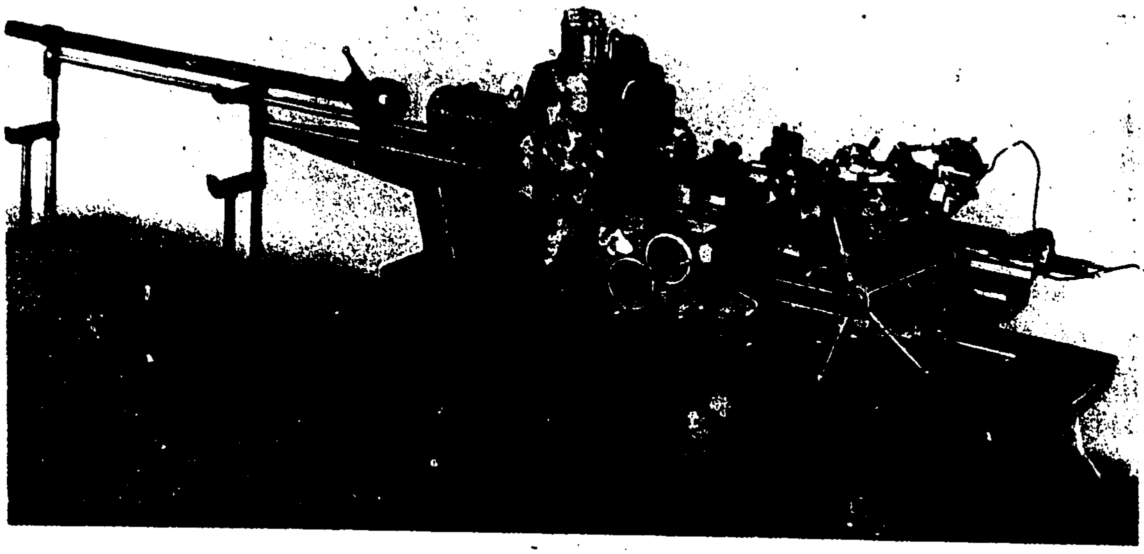
- NEVER completely trust the automatic stops on a turret lathe. Be alert at all times to the progress and movement of the cutting tool(s).

- NEVER exceed the recommended depth of cut, cutting speeds, and feeds.

- Before starting the machine, always be alert for tools, clamping devices, or other materials adrift on the table of a vertical turret lathe.

HORIZONTAL TURRET LATHES

The horizontal turret lathe is a modification of the engine lathe. The biggest difference is that the turret lathe has two multifaced toolholders. One toolholder (or turret head, as it is called) is located where the tailstock is on an engine lathe. In a typical turret lathe, the turret head has six faces, on each of which can be fastened various single tools or groups of cutting tools. The other turret toolholder (usually square and therefore called the square turret) is mounted on a cross-slide found on an engine lathe. A typical cross-slide turret can hold one cutting tool on each face. However, some types can mount two or more tools on one face. Each turret rotates about an upright axis. Thus, if you mount the



28.159

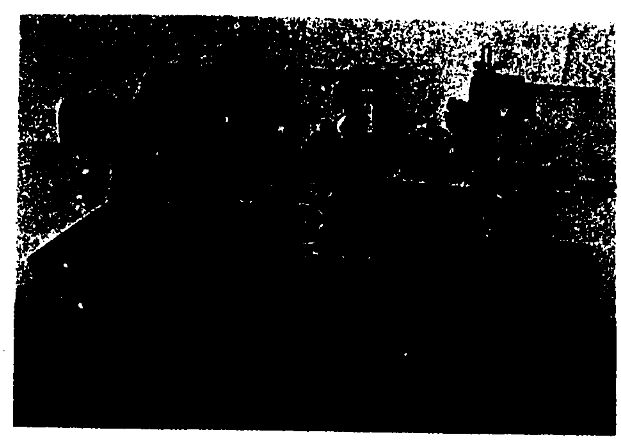
Figure 10-1.—Bar machine.

proper cutting tools on the turrets, you can do several different machining operations in rapid sequence by merely rotating another tool or set of tools into position for feeding into the work. Moreover, you can do simultaneous machining operations. For instance, on a particular job, the cross-slide turret tool may be taking an external cut on the workpiece while a tail-mounted tool on the turret head is performing an internal machining operation on the piece such as boring, reaming, drilling, or tapping.

CLASSIFICATION OF HORIZONTAL TURRET LATHES

Figures 10-1 and 10-2 show two classes of horizontal turret lathes, the bar machine and the chucking machine. One main difference between the two is the size and shape of the work they will machine. Bar machines are used for making parts out of bar stock or for machining castings or forgings of a size and shape similar to bar stock. (Note that the bar machine (fig. 10-1) has a stock feed attachment.) Chucking machines are used for machining castings, forgings, and cut bar stock that must be held in a chuck or fixture because of their large size or odd shape. The other main difference between bar and chucking machines is in the types of turning tools and holders used with the machines.

Since the bar machine is designed to machine pieces that have a relatively small cross section, its hexagonal turret turning tools must



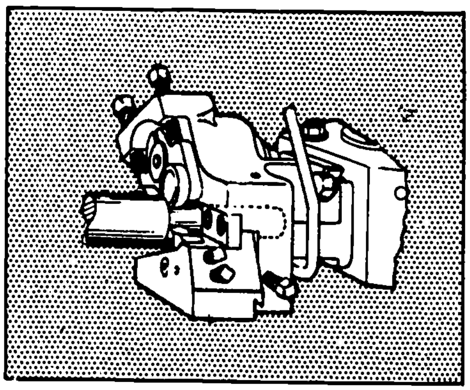
28.160

Figure 10-2.—Chucking machine.

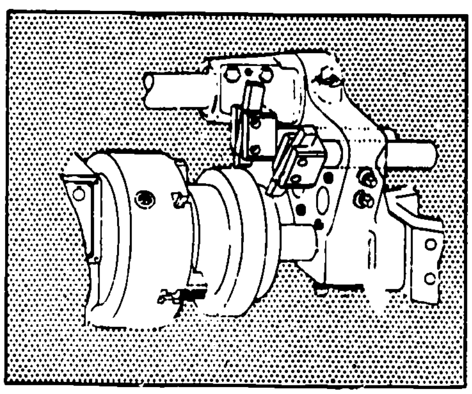
be able to support the work during cutting; otherwise, the workpiece will very likely bend away from the cutting tool.

The stock material which the chucking lathe is designed to machine is usually rigid enough to withstand heavy cutting forces without support. Figure 10-3 illustrates the difference between a bar setup and chucking setup for a hexagonal turret.

Bar machines and chucking machines may be either the ram type (fig. 10-4) or the saddle type (fig. 10-5). On the ram type, the turret head is mounted on a ram slide, which you can move longitudinally on a saddle that is clamped to the bedways of the machine. The ram has both hand and power longitudinal feeds. To make adjustments, you must manually move the saddle upon which the ram is mounted along the bedways. The stroke of the ram is relatively short. For this reason, the ram type is not used for working material that requires long longitudinal machining with hexagonal turret-held tools.



A–BAR TURNING SETUP



B–CHUCKING SETUP

28.161X

Figure 10-3.–Hexagonal turret turning tool setups.

The saddle type lathe has the turret head mounted directly on the saddle which, with its apron or gear box, moves back and forth on the bedways. The length of the longitudinal cut you can make with a hexagonal turret-held tool is limited only by the length of the bedways.

Hexagonal turrets found on-board ship do not normally have cross feed. However, cross feed is available on some saddle type lathes. An example of a cross-sliding hexagonal turret is shown in figure 10-5. The small handwheel just to the left of the large saddle hand feed wheel controls the manual crossfeed. There are levers for engaging power feed. The hexagonal turret realigns with the spindle axis when the cross-slide is returned to its starting position.

Standard toolholders are used to provide cross feed for the ram type and the fixed center turret saddle type.

COMPONENTS

Many of the components of turret lathes are similar to those of engine lathes. We will discuss only the main components of the turret lathe that differ in principle of operation from the engine lathe components. If you clearly understand the construction and functions of an engine lathe, you will have little difficulty in learning the construction and functions of turret lathes.

Headstock

The first important unit of any turret lathe is the headstock. Many lathes have a multiple-speed motor coupled directly to the spindle. Others have all-geared heads which provide an even wider range of spindle or chuck

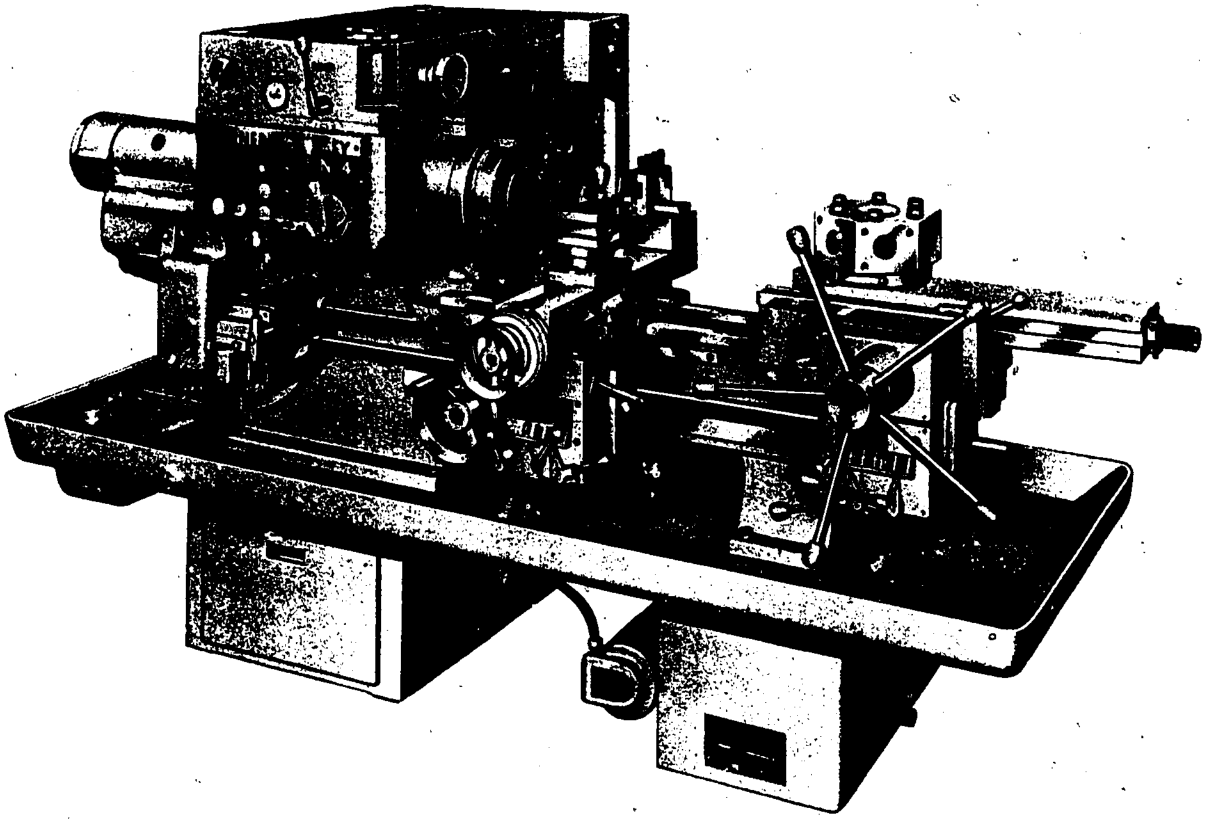


Figure 10-4.—Ram type bar machine. 28.342

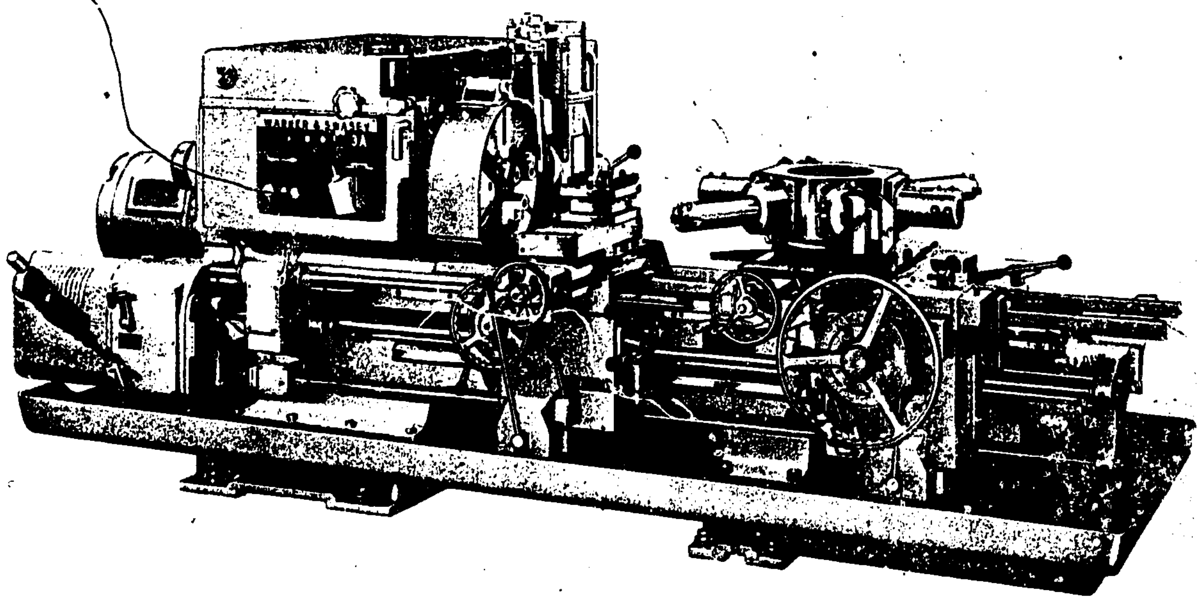


Figure 10-5.—Saddle type chucking machine. 28.343

speeds. The all-geared heads come in a variety of designs, each having a different number of speeds and a different method of selecting and changing the speeds. Some models have a pre-selector that lets you set up the different speeds you will need for a job before you begin. On these machines, speed changes are made through a minimum number of rapid changes without interfering with the timing of the operation.

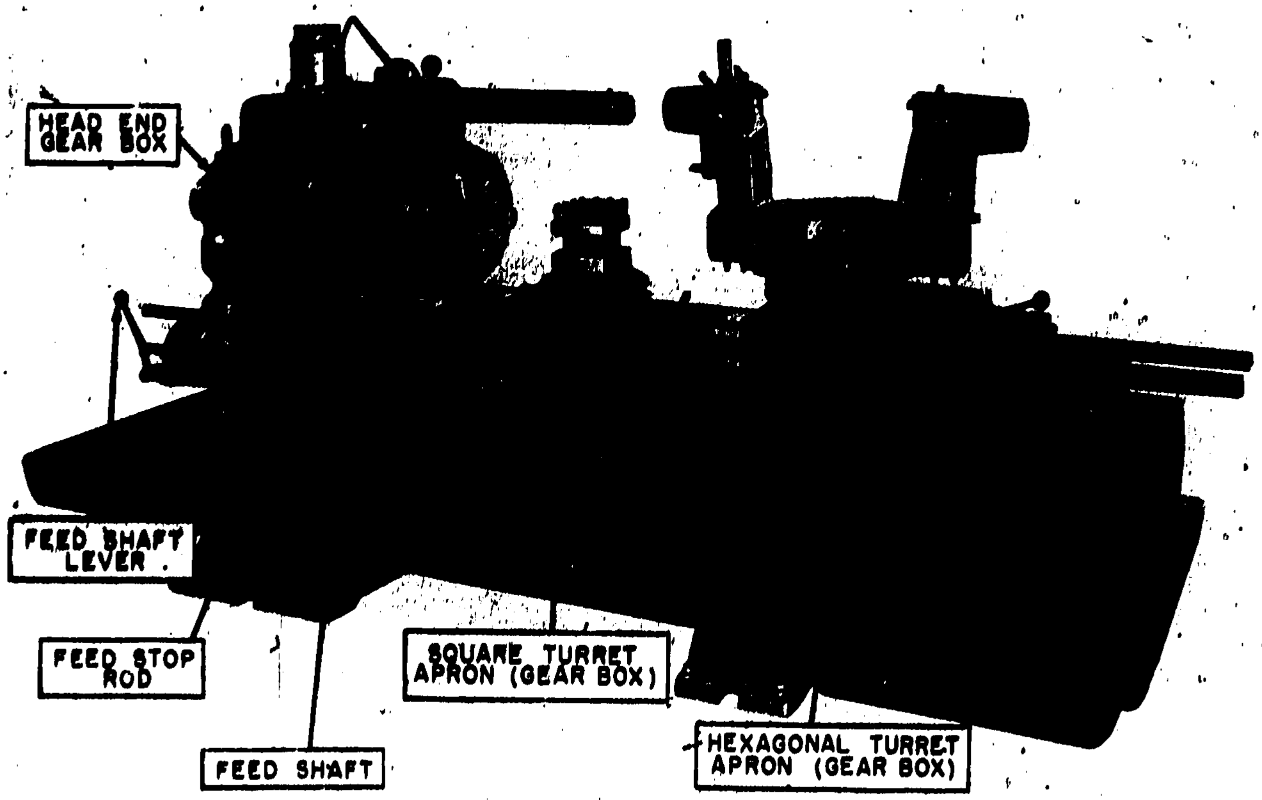
Feed Train

The feed train of a turret lathe (fig. 10-6) transmits power from the spindle of the machine to both the cross-slide and the hexagonal turret. The feed train consists of a head end gear box, a feed shaft, a square turret carriage apron or gear box, and a hexagonal turret apron or gear box.

The number of different feeds varies, depending upon the size and model of the machine. On any machine, first select a range of feeds by shifting or changing the gears in the head end gear box. Then shift the levers in the aprons to select the desired feed.

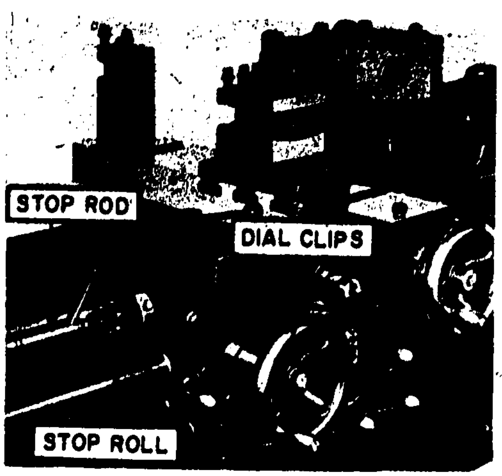
Feed Trips and Stops

To save time in making a number of duplicate parts, many horizontal turret lathes have feed trips and positive stops on the cross-slide unit and the hexagonal turret unit saddle or ram, which when set, eliminate the need for measuring each piece.



28.165

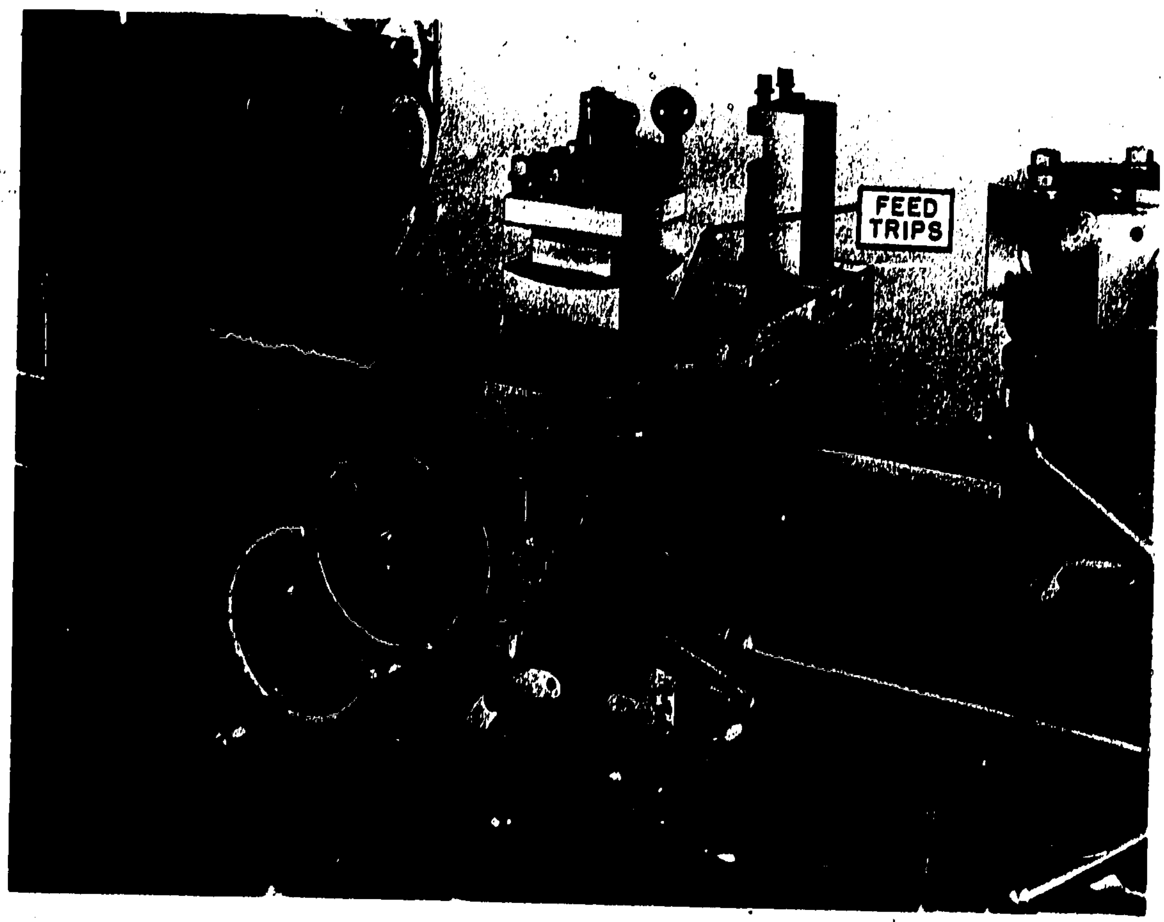
Figure 10-6.—Saddle type turret lathe feed train.



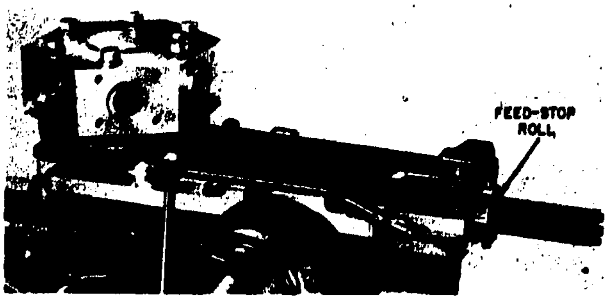
28.166
Figure 10-7.—Typical longitudinal feed stop arrangement for cross-slide.

A 6-station stop roll (fig. 10-7) in the carriage and an adjustable stop rod in the head bracket allow for duplicating sizes cut with a longitudinal movement of the cross-slide carriage. Stop screws in the stop roll let you set the cutoff for any particular operation, and a master adjusting screw in the end of the stop rod lets you make an overall setup adjustment without disturbing the individual stop screws. The dial clips shown in figure 10-7 are used as a reference for accurately sizing a piece by hand feed after the power crossfeed has been knocked off by the crossfeed trips shown in figure 10-8.

Turret stop screws on the ram type machine are mounted in a stop roll (fig. 10-9) carried in the outer end of the turret slide. The screw in the lowest position of the stop roll controls the travel of the working face of the turret. The stop



28.167X
Figure 10-8.—A typical crossfeed trip arrangement.



28.168X
Figure 10-9.–Hexagonal turret feed-stop roll on ram type machine.

roll is connected to the turret so that when a particular face of the turret is positioned for work, its mating stop screw is automatically brought into correct position.

To set the hexagonal turret stops on ram type machines:

1. Run a cut from the turret to get the desired dimensions and length.
2. Stop the spindle, engage the feed lever, and clamp the turret slide.
3. Turn the stop screw in until the feed knocks off; then continue turning the screw in until it hits the dead stop.

On saddle type machines, the stop roll for the hexagonal turret is located under the saddle and between the ways (fig. 10-10). The stop roll does not move endwise; it automatically rotates as the turret revolves. To set the stops:

1. Move all the dogs back to the outer end of the roll, where they will be in a convenient position. Select a turret face and allow the master stop to engage the loosened stop dog. After you take the trial cut, the stop dog will slide ahead of the master stop.

2. After you have taken the proper length of cut, stop the spindle, engage the longitudinal feed lever, and clamp the saddle. Then, adjust the stop dog to the nearest locking position with the screw nearest the master stop. When the end of the dog is flush with the edge of a locking

28.169X
Figure 10-10.–Hexagonal turret feed stops on saddle type machine.

groove on the stop roll, the locking screw nearest the master stop will line up automatically with the next locking groove.

3. Screw down the first lock screw, at the same time pressing the stop dog toward the head end of the machine.

4. Screw down the second lock screw and then adjust the stop screw until it moves the master stop back to a point where the feed lever knocks off. Then tighten the center screw to bind the stop in position.

Threading Mechanisms

There are several different methods for producing screw threads on a turret lathe. The most common method is to use taps and dies attached to the hexagon turret. The design and proper use of these tools will be covered later in this chapter. A thread chasing attachment (fig. 10-11) allows the machining of screw threads on a surface up to about 7 inches long. There are two major parts to this attachment. The leader is a hollow cylindrical shaft that clamps over the feed rod of the turret lathe. You can position it anywhere along the feed rod for alignment with the surface requiring threads. The follower is a half-nut type arrangement, similar to an engine lathe. It is bolted to the carriage and engaged

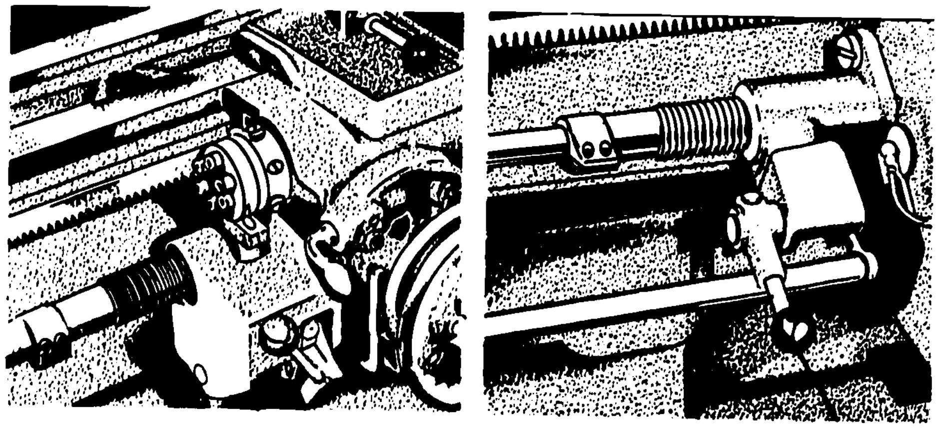


Figure 10-11.—Thread chasing attachments.

28.344

over the threaded part of the leader. Disengagement is either manual or automatic, depending on the model. This attachment can normally be installed on existing equipment. An attachment that requires factory installation is the lead screw threading attachment (fig. 10-5). This attachment gives the turret lathe the same threading capability as an engine lathe. A lead screw extends the working length of the lathe to allow for threading long workpieces. A quick-change gear box on the headstock end of the lathe provides for a wide and rapid selection of a number of threads per inch.

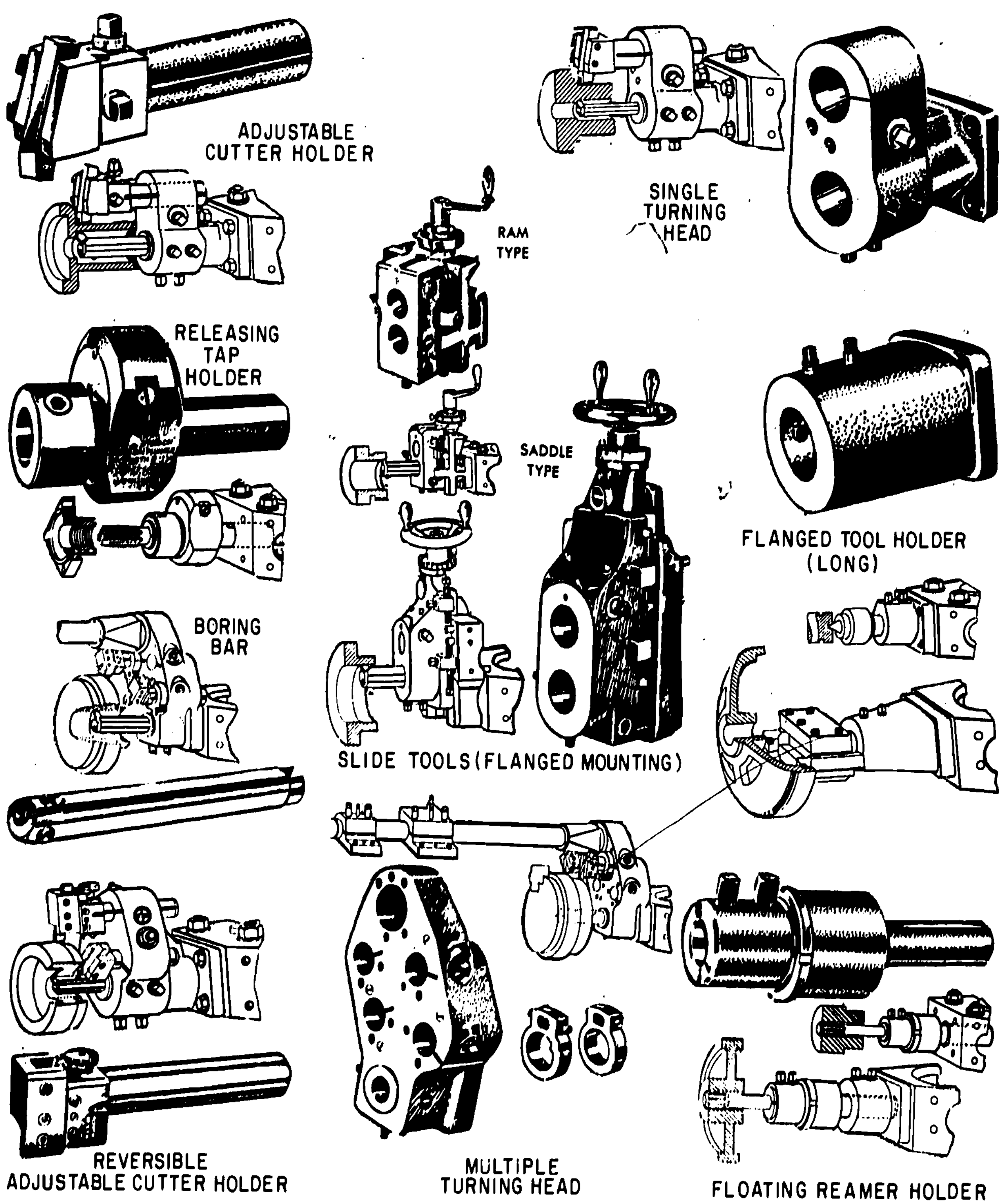
TURRET LATHE OPERATIONS

Aside from additional control levers and additional automatic features, the principal differences between operating an engine lathe and a turret lathe lie in the methods of tooling and in the methods of setting up the work. In this section we will discuss turret lathe tooling principles and methods of doing typical jobs in horizontal and vertical turret lathes.

Proper maintenance is important for efficient production on a turret lathe. Specific maintenance procedures for a specific turret lathe are given in the manufacturer's technical manual. Before starting a lathe, ensure that all bearings are lubricated and that the machine is clean. Turret lathes have pressurized lubrication systems and have peepholes at strategic points in the system so that you can tell at a glance whether oil is being circulated to the areas where it is required. When cleaning a lathe, use a cloth or brush to remove chips. DO NOT use compressed air to clean a lathe. Compressed air is likely to blow foreign matter into the precision fitted parts, causing extensive damage.

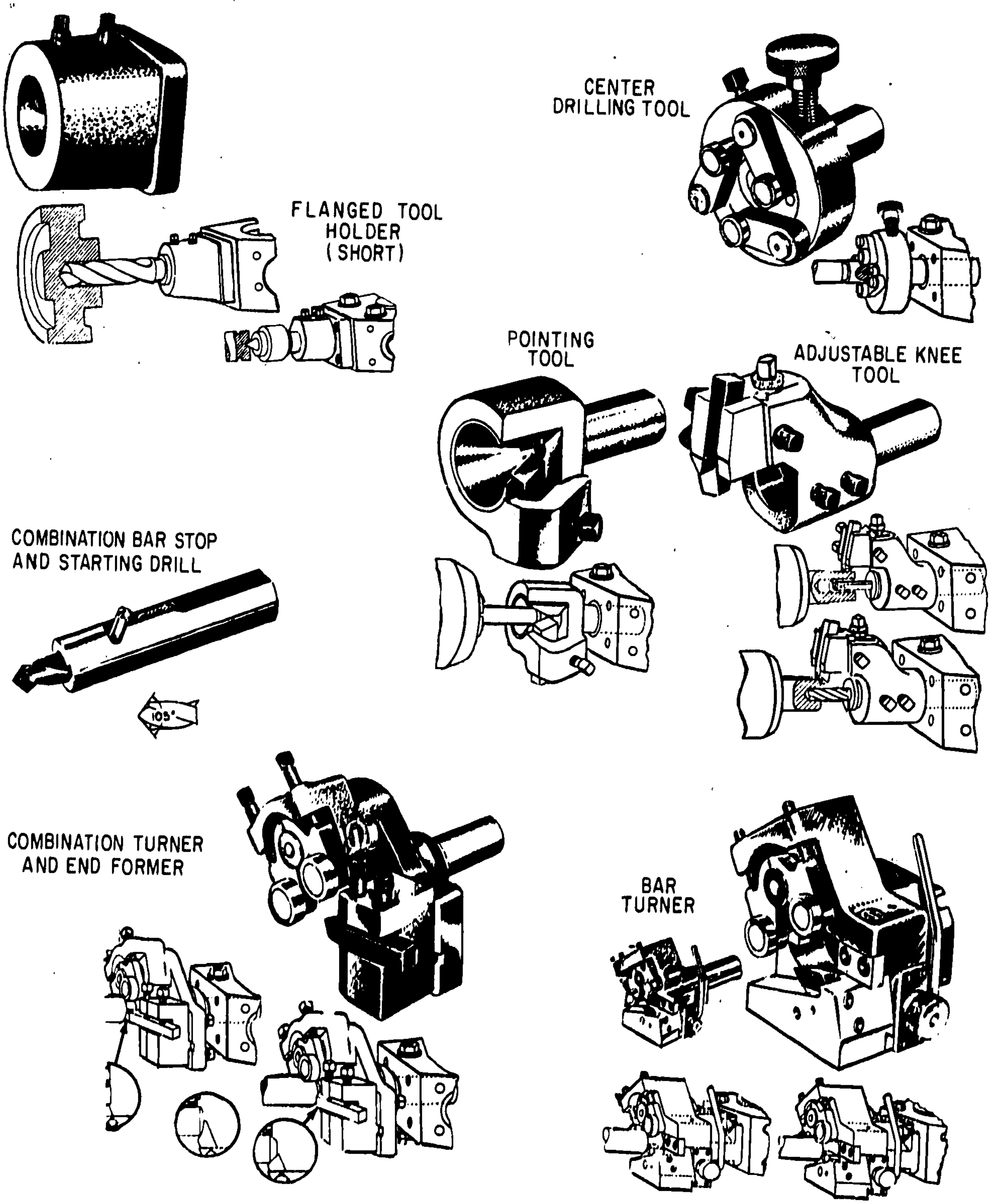
TOOLING HORIZONTAL TURRET LATHES

As previously mentioned, horizontal turret lathes fall into two general classes, the bar machines and the chucking machines. The principal differences between the two classes are in the size and shape of the workpieces they handle, the type of workholding device, and the type of turning tools employed in the hexagonal turret. In the following paragraphs which describe workholding devices, grinding and setting cutters, and various machining procedures, we make no particular distinction as to the class of machine involved, because it will usually be obvious; where it is not obvious, the information applies to horizontal turret lathes in general. The preceding information also applies to the two types of machines, the ram type and the saddle type. Examples of some of the commonly used tools for a chucking machine are shown in figure 10-12 and tools for a bar machine in figure 10-13.



28.345

Figure 10-12.—Turret lathe chucking tools.



28.346

Figure 10-13.—Turret lathe bar tools.

As a good turret lathe operator, your aim should be to tool and operate the machine so as to turn out a job as rapidly and as accurately as possible. Always keep in mind the following factors:

- Keep the total time for a job at a minimum by balancing setup time, work handling time, machine-handling time, and actual cutting time.

- Reduce setup time by using universal equipment and by arranging the heavier flanged type tools in a logical order.

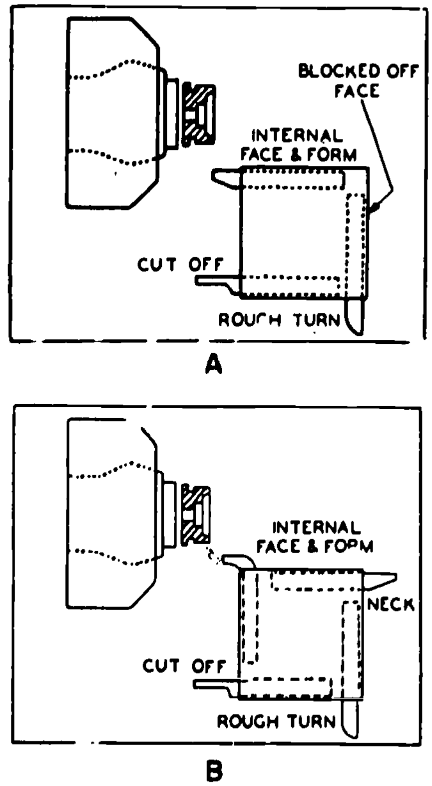
- Select proper standard equipment. Use special equipment only when it is justified by the quantity of work to be produced.

- Reduce machine handling time by using the right size machine and by taking as many multiple cuts as possible.

- Reduce cutting time by the following methods: (1) Take two or more cuts at the same time from one tool station, (2) take cuts from the hexagonal turret and the cross-slide at the same time, and (3) increase feeds by making the setup as rigid as possible by reducing tool overhang and using rigid toolholders.

- Never block off stations on the square turret. (See fig. 10-14).

- Keep the distance that each tool projects from the hex turret as equal as possible. This will minimize the length of travel required to retract each tool for indexing to the next one.



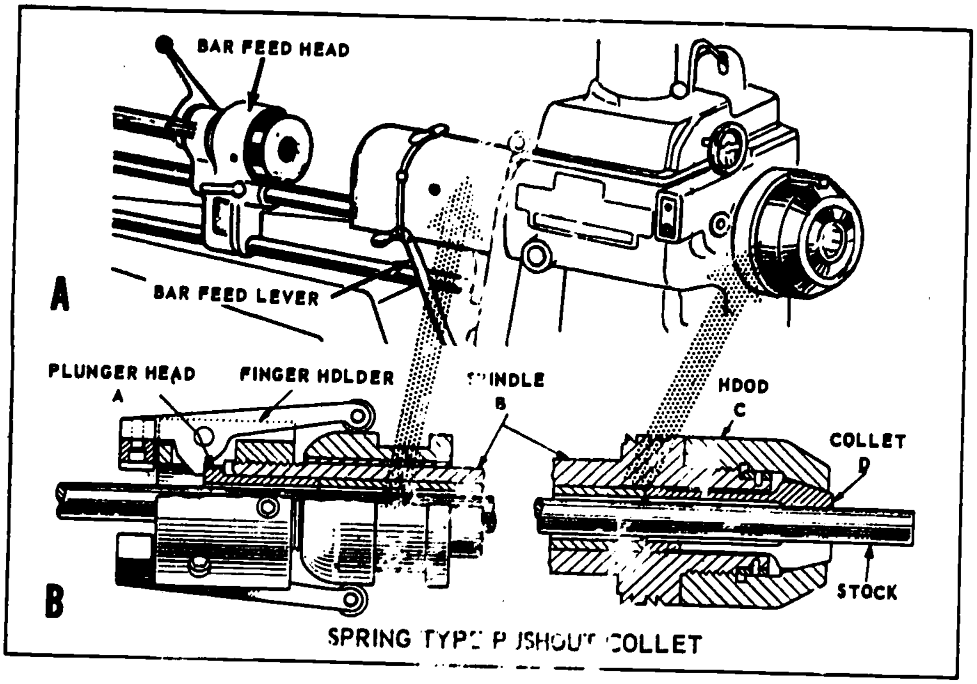
28.171X

Figure 10-14.—Square turret tool positions.

Holding the Work

Horizontal turret lathes are generally used for turning out machine parts rapidly in quantity. The workholding devices must be such that you spend little time in placing stock material in the machine. Moreover, to produce duplicate parts of uniform corresponding dimensions, once you have set the tools, the workholding device must be able to position and hold each succeeding raw workpiece without your having to stop to take measurements or make adjustments. (Remember: SAFETY FIRST, ACCURACY SECOND, AND SPEED LAST.) The semiautomatic collets, arbors, and chucks described in the following sections are able to do this.

COLLETS.—The spring-type pushout collet shown in figure 10-15 is the most widely used. It is made in different sizes for use on bar stock up to 2 1/2 inches in diameter. The principle upon which it works is as follows: When you engage the bar feed lever on the machine to start the bar feed head (fig. 10-15A) to advance the stock, it simultaneously loosens the grip of the collet. When the end of the bar stock butts against a stock stop mounted on one face of the hexagonal turret, the plunger (Part A in fig. 10-15B) forces the partially split tapered end of



28.172X
Figure 10-15.—Bar feed device and spring type pushout collet mechanism.

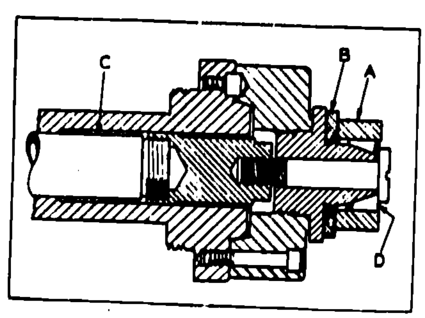
collet D into the taper of the hood C, causing the collet to grip the stock firmly. Your one simple movement automatically set the stock material into position for machining.

There are several variations of the spring-type collet, but they all depend upon the plunger head principle for gripping and releasing the stock, differing only in the direction of taper on the collet.

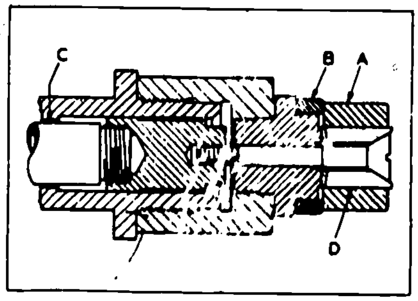
ARBORS.—For mounting small, rough castings or for mounting workpieces of second operations, you will often use quick-acting arbors.

Figure 10-16 is an expanding bushing-type arbor. In this type arbor, as draw bar C is pulled back, the split bushing D climbs the taper of the arbor body, expanding to grip the workpiece A tightly along its entire length and at the same time forcing the workpiece against stop plate B. This type of arbor is suitable for roughing work or first operations, where a firm grip for heavy feeds is more important than accuracy.

The expanding plug-type arbor (fig. 10-17) centers the workpiece more accurately and is usually used for second or finishing operations. In this type, when the taperheaded screw is pulled to the left by the action of the draw bar C, it expands the outer end of the partially split plug D enough to grip the workpiece A internally and at the same time forces the workpiece tightly against the stop plate B. This



28.173X
Figure 10-16.—Expanding bushing-type arbor.



28.174X
Figure 10-17.–Expanding plug-type arbor.

type or arbor is used for holding workpieces that have been bored or reamed to size internally, rough machined to size externally, and need only a light finishing cut as a final operation.

CHUCKS.–These workholding devices fall into three classes: (1) univeral chucks of the geared scroll, geared screw, or box type, where all three jaws move at the same time; (2) independent chucks, where each of the jaws is operated independently; and (3) combination chucks, where each jaw may be operated independently, but where all jaws can be moved as a group.

The 2-jaw chuck is used mostly for holding small work or irregular shape. The jaw screw operates both jaws at the same time. Use an adapter to attach chuck jaws of various shapes to the master jaws.

The 3-jaw geared scroll chuck is used more than any other type. With standard jaw equipment, it holds work of regular shape. It can be adapted to hold irregularly shaped work.

Figure 10-18 shows a 4-jaw combination chuck which has two-piece master-jaw construction and an independent jaw screw between sections. The bottom or master part of the jaw is moved by the scroll, and the top part is moved by the independent jaw screw. Chucks of this type are used mostly to hold irregularly shaped work or when a jaw needs to be offset from a true circle. On the combination chuck, you use the independent movable jaws to true

28.175X
Figure 10-18.–A 4-jaw combination chuck.

the work in the first chuckings. You can then use the same chuck for second operations by using the geared scroll to operate the jaws when gripping on a finished diameter. Soft metal (such as copper shims) is often used with chuck jaws for chucking second operation work to prevent marring the finish of the workpiece.

Some machines have a power chuck wrench that you use with 3-jaw chucks. This attachment lets you open and close the chuck by a lever located on the headstock. There is a control knob for adjusting the pressure of the chuck to allow for gripping different workpieces. An example of such an attachment can be seen on the turret lathe in figure 10-5.

Grinding and Setting Turret Lathe Tools

The angles to which a turret lathe tool is ground and the position at which it is set can change the angle that the cutting edge of the tool forms with the work. The angles ground and the position set affect the chip flow, pressure exerted on the tool, and the amount of feed and depth of cut that can be used. Consequently, accurate tool angles and proper tool position are essential to production when you use a turret lathe.

GRINDING.–Some important points to keep in mind when grinding turret lathe tools are:

- Do NOT estimate the tool angles you should use, nor the tool angle you should grind. Consult a tool angle chart (see chapter 6 of this manual) to determine the correct angles. To check the angle when you grind a cutter, use the cutter grinding gage or the bar turner cutter grinding gage, as appropriate. These gages are part of your lathe equipment.

- Some cutters are ground wet; others are ground dry. High-speed steel cutters are usually ground wet, while Stellite and carbide cutters are usually ground dry. When grinding wet, keep the cutter well-flooded to prevent heating; nothing will ruin a cutter quicker than a wet grinding that is partially dry. On the other hand, if the cutter should be ground dry, do not dip the tip in coolant. Sudden cooling will cause surface cracks, which once started will eventually cause the cutter tip to fail.

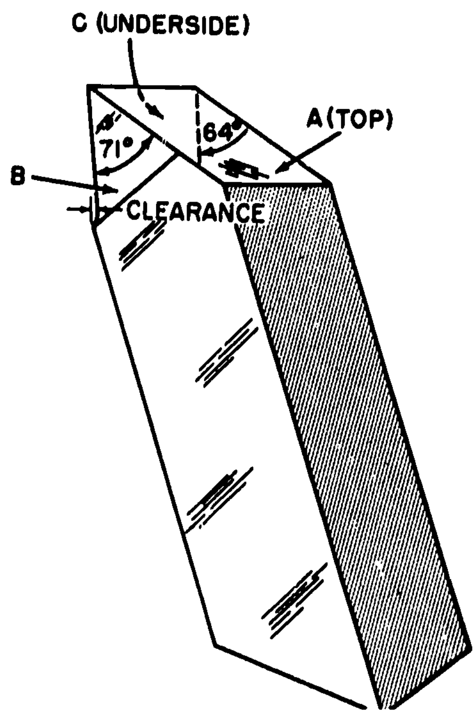
- When a carbide-tipped cutter requires sharpening, use the grinder specified in your shop for that purpose. Grinding wheels suitable for high-speed steel will ruin carbide cutters.

- When you are grinding a carbide-tipped cutter, manipulate it so that pressure of the grinding will be toward the seat of the carbide tip rather than away from it.

The tool angles of single cutters and multiple turning head cutters for the square turret and hexagonal turret, respectively, are quite similar to those employed in engine lathe tool bits or turning tools. But the cutters themselves are usually much larger than those used on an engine lathe because the turret lathe is designed to remove large quantities of metal rapidly. Bar turner cutters, or box tools as they are often called, are ground in a different manner.

Bar turner cutters are usually held in a semivertical position. That is, the cutting edge or tool point, which is located near the center of the cutter end, points slightly toward the cut and toward the center of the work. In this position, the pressure of the cut is downward through the shank of the cutter.

Bar turner cutters are ground to form the tool point on the end of the cutter, near the centerline, somewhat like a chisel point. The bar turner cutter in figure 10-19 is in the position it would be held in the holder. Normally, in sharpening, you grind only angle surfaces A (the top). You hone angle surfaces B and C to remove burrs which result from grinding surface A. After repeated sharpenings, angle surfaces B and C will become too small and you must then grind them. The tool angles for a bar turner cutter are the same as those on a cross-slide mounted cutter, but they appear to be vastly

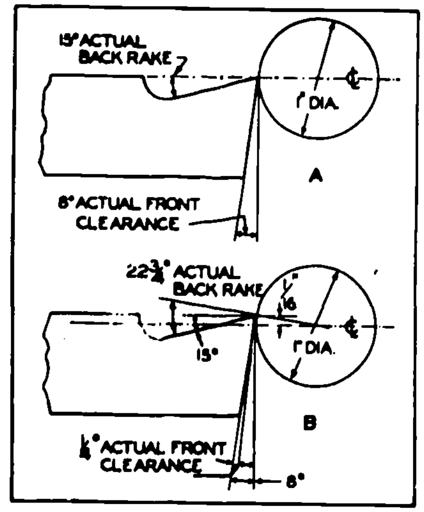


28.176X

Figure 10-19.—Bar turner cutter tool point.

different because of the difference in tool point location.

CONTROLLING CHIPS.–Chips can be controlled in one of two ways: (1) get the right combination of back and side rake angles in combination with speeds and feeds or (2) grind on the back rake face of the cutter a chip breaker groove which will curl and break chips into short lengths. Method (1) is usually the best way. By changing the angle slightly, it is possible to throw chips in one direction or the other. If you use method (2), start the chip breaker groove just behind the cutting edge; be careful not to carry it through the point of the cutter. A chip breaker groove through the point of the cutter will tend to break down the cutting point, produce a poor quality of finish, and may produce a double chip (fig. 10-20).

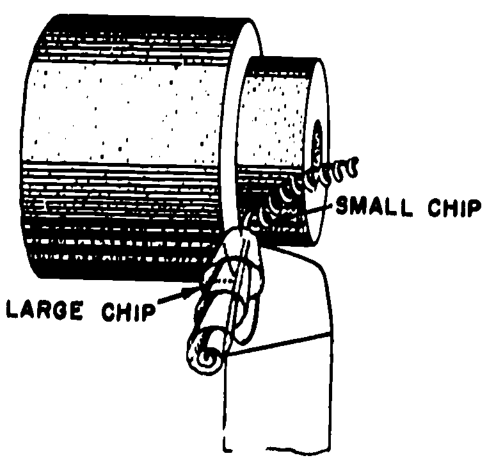


28.178X

Figure 10-21.–Keep cutters on center.

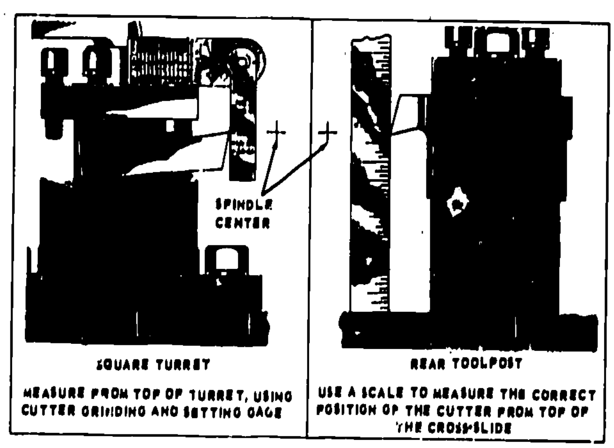
SETTING SINGLE AND MULTIPLE TURNING CUTTERS.–To retain all of its small front clearance angle, a turret lathe cutter must be set in its holder so that its active cutting edge is on the same plane as the centerline of the work, and not above center as tool bits are often set in engine lathe operation. Part A of figure 10-21 shows a cutter in correct position. This cutter-workpiece relation is very important when the workpiece diameter is small. Observe in part B of figure 10-21 the effect of raising the cutter above center. A cutter set in the position shown has only a fraction of the amount of front clearance needed under its cutting lip and has an unnecessarily large back rake angle. On the other hand, if a cutter is set below center for cutting small diameter work, the work is very likely to climb the cutter, or at least cause violent chatter.

Figure 10-22 shows how to set a square turret and a "reach over" or rear-tool station



28.177

Figure 10-20.–Double chip caused by grinding a chip breaker groove too close to cutting edge.



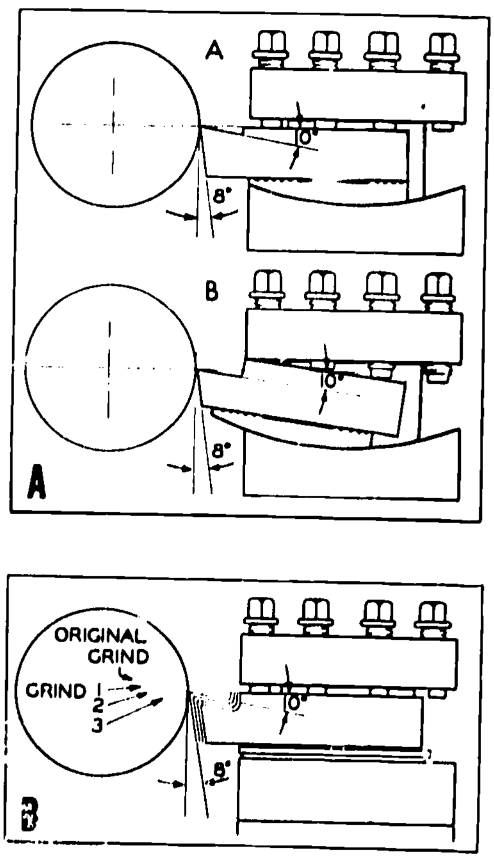
28.179

Figure 10-22.–Setting square turret and "reach over" toolpost cutters on center.

cutter on center. Notice that the cutter in the "reach over" toolpost is inverted; the reason for this is that the work surface rolls up from underneath.

In square turrets, you can raise or lower the cutter to correct position by either shims or rockers, depending upon the type of base plate (fig. 10-23).

Another factor to consider in setting a cutter is the amount of its overhang from the holder. Too much overhang will cause the cutter to chatter, and insufficient overhang will cause the holding device to foul the work. When possible, you should keep the amount of overhang equal to or slightly less than twice the thickness of the cutter shank.



28.180
Figure 10-23.—A. Use of rockers. B. Use shims.

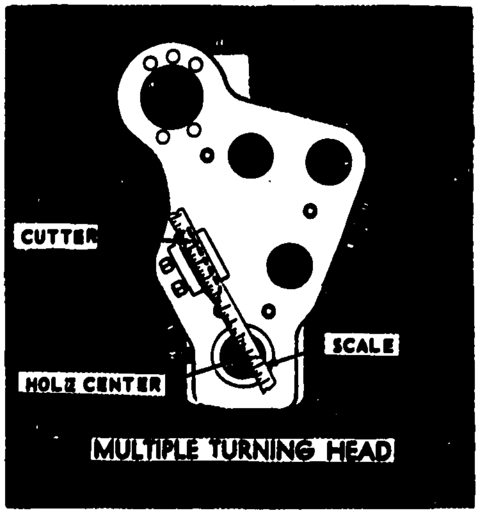
Each time you regrind a cutter (other than a carbide-tipped type), the height of the tool point and the length of the cutter itself are reduced; therefore, after each grinding you must reposition the cutter in its holder to place the tool point on center. If you use a shim type holder, raise the cutter to center by adding a shim of appropriate thickness (fig. 10-23B). When using a rocker arrangement, you need an entirely different approach; elevating the reground tool point to center by adjusting the rocker will cause the clearance and rake angle to change. The best way to maintain the proper angles and yet keep the tool point on center, when using the rocker arrangement, is to decrease the top (back and side) rake angles and increase the front clearance angle slightly at each grinding in anticipation of the change in cutter position caused by removal of metal from the tool point. Figure 10-23A shows how this is done.

The dimensions of carbide-tipped cutters are relatively unaffected by grinding; therefore, the cutters seldom require alteration in holder setup after they have been reground. The shim type holder presents a stable horizontal base for the cutter shank and is best for holding carbide-tipped cutters. The cutters can be taken out, reground, and placed back in and on center without undue manipulation.

The overhead turning cutters, which are mounted on the hexagonal turret, must also be on center in relation to the work. The principle involved in setting these cutters is not different from that involved in setting the square turret—mounted cutters, though at first it may appear to be different. In order to assure yourself that this is so, look at figure 10-21 and turn the book so that the cutters point toward the work from above rather than from the side.

Figure 10-24 shows how to set an overhead turning cutter on center by using a scale for reference in bringing the shank and tool point of the cutter into radial line with the center of the turning head, which is an alignment with the center of the spindle.

SETTING BAR TURNER CUTTERS.—Bar turners are held on the hexagonal turret and



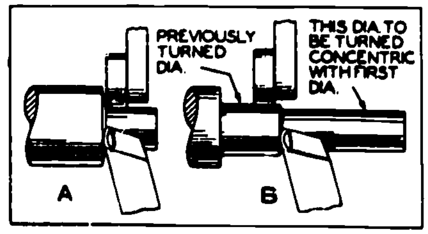
28.181X
Figure 10-24.—Setting overhead turning cutter on center.

combine in one unit a cutter holder and backrest that travels with the cutter and supports the workpiece. The backrest holds the work against the cutter so that deep cuts can be taken at heavy feeds.

Backrests on bar turners usually have rollers to eliminate wear and to make high-speed operation possible. Bar turners that have V-backrests are used for turning brass where there is no problem of wear and where small chips might get under rollers and mar the workpiece.

The rollers on a ROLLER-TYPE TURNER may be either ahead of or behind the cutter. If they are behind the cutter, they burnish the workpiece. This burnishing is often an important factor; it may eliminate the need for polishing or grinding operations. When a diameter is turned so that it is concentric with a finished diameter, the rollers are run ahead of the cutter on the previously finished surface. Figure 10-25 illustrates rollers behind and ahead of a cutter.

The rollers on a UNIVERSAL TURNER are set ahead of or behind the cutter by adjusting the movable cutter with the rolls remaining in fixed position. The universal bar turner is



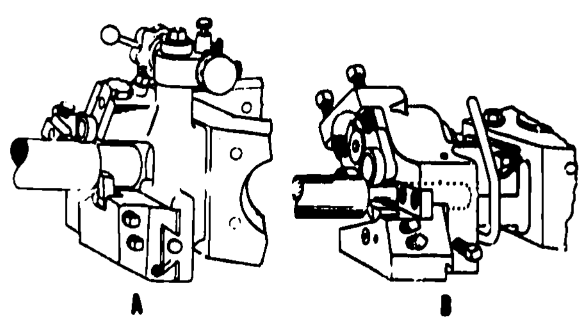
28.182X
Figure 10-25.—Rolls. A. Behind cutter. B. Ahead of cutter.

illustrated in figure 10-26A. Another type, the single-bar turner (fig. 10-26B), has adjustable roller arms; the cutter is fixed, and the rollers can be moved ahead of or behind the cutter.

Use the following steps in setting up a SINGLE BAR TURNER:

1. Extend the bar stock about 1 1/2 to 2 inches from the collet. Then with a cutter in the square turret on the cross-slide, turn the bar to 0.001 inch under the size desired for a length of 1/2 to 1 inch.

2. With the roller jaw swung out of position (fig. 10-27A) and with the cutter set above center, adjust the cutter slide of the turner against the turned portion of the bar stock and rub a shine mark on the turned portion, as indicated in fig. 10-27B.



28.183X
Figure 10-26.—A. Universal bar turner. B. Single bar turner.

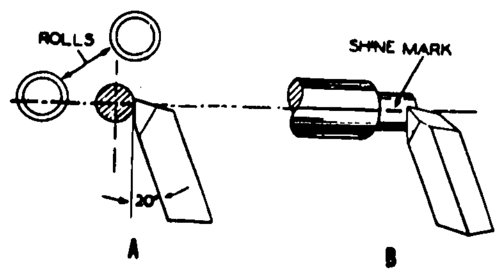
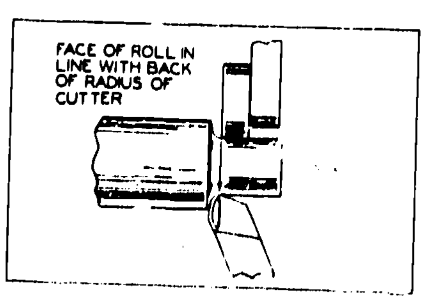


Figure 10-27.—Rubbing a shine mark to establish center. A. Roll jaws out of position. B. Shine mark on turned portion.

3. Set the cutter at the center of the shine mark, clamp the cutter tightly in its slide, turn the spindle to move the shine mark away from the cutter point, and adjust the slide until the cutter is 0.0015 inch from the turned diameter. You now have the cutter set, and the rollers should be positioned endwise and adjusted to size.

4. Align the rollers with the back of the point radius of the cutter, as shown in figure 10-28. Adjust the rollers with the clamping screws, and then clamp tightly. The rollers are in proper adjustment when LIGHT PRESSURE WILL STOP THEM FROM TURNING as the bar stock is revolved.

5. Push the cutter to cutting position with the withdrawal lever and take a trial cut. If you have a proper setup, the size will be accurate to ±0.001 inch.



28.185

Figure 10-28.—Roll aligned with cutter.

BAR TURNING.—The following pointers will be helpful in bar turning:

- To prevent making marks on the work as you bring back the turret, always use the withdrawal lever before the return stroke of the turret.

- When rollers are set to follow the cutter, it is usually true that the heavier the cut the better the finish. The heavier the cut the greater is the pressure against the rollers, and the greater is the burnishing action.

- If you are using light cuts, special rollers with a steep taper will sometimes produce a better finish.

- Regardless of the depth of cut, there are three factors that you must watch to get a high grade finish: (1) the faces of the two rollers must be in line, (2) the leading corners of the rollers must be perfectly round and exactly equal, and (3) end play in the rollers should not exceed 0.003 inch.

Selecting Speeds and Feeds

The general rules for feeds and speeds in chapter 8 of this manual for engine lathe operation apply also to turret lathes. However, since the cutters and the machine itself are designed for production work, you can take heavier roughing cuts that you ordinarily would with an engine lathe.

Bear in mind that the spindle speed of the turret lathe must be governed by the surface speed at the point of work of the cutter farthest from the rotating axis. That is, if you are going to use two cutters on a workpiece with one cutter to turn a small diameter and the other to cut a much larger diameter, the headstock rpm you select must be based on the surface speed at the large diameter. Disregard the fact that the cutter at the small diameter will not be cutting at anywhere near its permissible rate.

Using Coolants

Using coolants makes it possible to run the lathe at higher speeds, take heavier cuts, and use cutters for longer periods without regrinding, thus getting maximum service from the lathe. Coolants flush away chips, protect machined parts against corrosion, and help give a better finish to the work. A coolant also helps in greater accuracy by keeping the work from overheating and becoming distorted. Figure 10-29 shows the incorrect and the correct methods of applying cutting oil or coolant.

Some coolants and the materials with which they are used are listed below:

CAST IRON–Soluble oil 1 to 30 ratio, or mineral lard oil, or dry

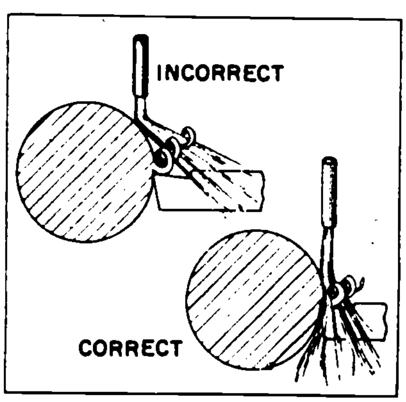
ALLOY STEEL–Soluble oil 1 to 10 ratio, or mineral lard oil

LOW/MEDIUM CARBON STEEL–Soluble oil 1 to 20 ratio, or mineral lard oil

BRASSES AND BRONZES–Soluble oil 1 to 20 ratio, or mineral lard oil

STAINLESS STEEL–Soluble oil 1 to 5 ratio, or mineral lard oil

ALUMINUM–Soluble oil 1 to 25 ratio, or dry



28.186

Figure 10-29.–Correct and incorrect methods of applying coolant.

MONEL/NICKEL ALLOYS–Soluble oil 1 to 20 ratio, or a sulfur-based oil

The selection of the best coolant or cutting fluid depends on the cutting tool materials, the toughness of the metal being machined and the type of operation being performed. Simple turning may require a coolant that keeps the temperature down and flushes chips away. A mixture of soluble oil that has a low oil ratio will do this very efficiently. An operation such as threading or heavy turning requires something that not only cools but also lubricates. A heavier soluble oil mixture or mineral lard oil satisfies these requirements.

BORING

Two general types of boring cutters are used–tool bits held in boring bars and solid forged boring cutters. Tool bits held in boring bars are most common. This combination allows great flexibility in sizes and types of work that can be done. Solid forged cutters, however, are used to bore holes too small to be cut with a boring bar and inserted cutter.

The cutter in a STUD BORING BAR is held either at right angles to the bar or extended beyond the end of the bar at an angle. This extension of the cutter makes it possible to bore up to shoulders and in blind holes. The angular cutting bar has the added advantage of an adjusting screw behind the cutter.

When the stub boring bar or forged boring bar is used, the overhang should be as short as the hole and the setup will permit. You should always select the largest possible size of boring bar to give the cutter as rigid a mounting as possible. You should never extend the boring cutter farther than is actually necessary. You can use sleeves to increase the rigidity of small stub boring bars and to reduce the effect of overhang. The increased rigidity helps to make the work more accurate and allows for heavier feeds.

The HEXAGON TURRET is ordinarily used in making boring cuts, although, if necessary, boring tools can be held on the cross-slide. The advantages of taking a boring cut from the hexagon turret are:

1. You can take turning or facing cuts with the cross-slide at the same time you take a boring cut with the turret.

2. You can combine boring cutters with turning cutters in multiple or single turning heads.

3. You can mount various size cutters eliminating the need for adjusting the cutter as the bore size increases.

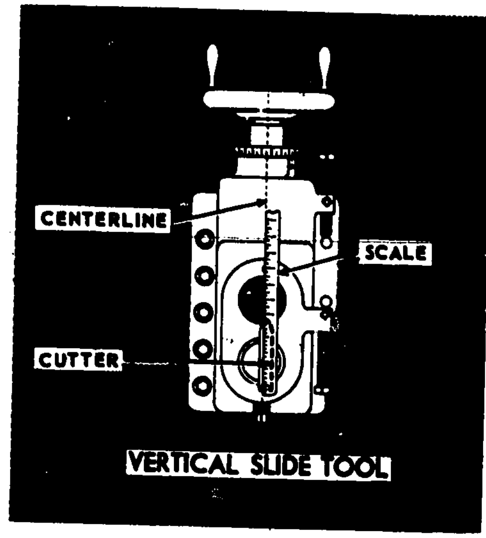
4. When a quantity of like pieces is required, you can increase boring feed by using a boring bar with two cutters. It is good practice when using double cutters to rough bore with a piloted boring bar to obtain rigidity for heavy feeds and then finish the hole with a stub boring bar held in a slide tool.

Piloted boring bars require a machine with a long stroke—the saddle type—so that the turret can be moved far enough to pull the piloted bar from the pilot bushing and the work before indexing the turret. Usually, when the pilot bushing is mounted in the chuck close to the work, the effective travel of the turret must be about 2 1/2 times the length of the workpiece.

Grinding Boring Cutters

Boring cutters are ground in the same manner as other types of cutters, with one major difference. The clearance angles of boring cutters must be greater to prevent rubbing since a boring tool cuts on the inside instead of on the outside of the work. However, the clearance angle must not be too great, or the cutting edge will break down because of insufficient support. The exact amount of front clearance angle will depend on the size of the hole you are boring. The smaller the hole, the more clearance required. There are no set rules for exact clearance angles; knowledge of what will be the best angle comes with experience in operating the lathe.

Figure 10-30 shows how to center a vertical slide tool-held boring cutter.



28.187X
Figure 10-30.—Setting a boring cutter on center.

Forming

One of the fastest methods of producing a finished diameter or shape is by forming. In planning a setup, you should study the work to determine if forming tools can be used. It is possible, on many jobs, to combine two or more cuts into one operation by using a specially designed forming cutter. Forming cutters are also used to produce irregular and curved shapes that are difficult to produce in any other way. There are three types of forming cutters you will use—forged, dovetail, and circular.

FORGED FORMING CUTTERS are ordinarily mounted directly in the square turret or toolpost and are the least expensive to make. They have, however, the shortest productive life.

DOVETAIL FORMING CUTTERS are held in toolholders mounted on the cross-slide. Their shape or contour is machined and ground the full length of the face, and the cutters are set in the holder at an angle to provide front clearance. When the cutter wears, you need to regrind only the top. Dovetail cutters cost more than forged cutters, but they have a longer productive life, are more easily set up, maintain their form after grinding, are more rigid, and can be operated under heavier feeds.

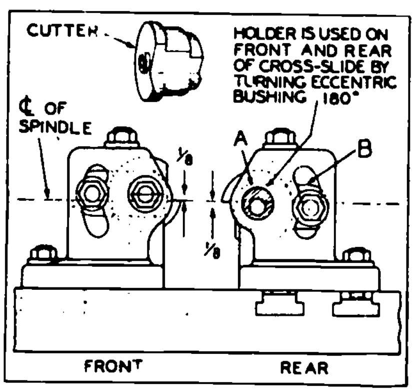
CIRCULAR FORMING CUTTERS have an even longer life than dovetail cutters. The shape of circular cutters is ground on the entire circumference and, as the cutting edge wears away, you regrind only the top. After grinding, move the cutter to a new cutting position by rotating the cutter about its axis. (See fig. 10-31.)

NEVER regrind circular forming cutters on a bench grinder. Regrind them on a toolroom grinder where they can be rigidly supported and ground to maintain the original relief angles.

Threading

For turret lathe operations, dies and taps provide a means of cutting threads easily and quickly and, usually, in only one pass over the work. Dies and taps for turret lathes are divided into three general types: Solid, solid adjustable, and collapsing or self-opening.

Solid taps and dies are usually held in a positive drive holder that has an automatic release (fig. 10-12). A longitudinal floating action (not to be confused with a floating die holder) allows the tap or die to follow the natural lead of the thread, which prevents having to depend on a delicate "feel" to produce results. Solid dies are used only when the thread to be cut is too coarse for the self-opening die head or a solid adjustable die head, or when the tool interferes with the setup.



28.188X

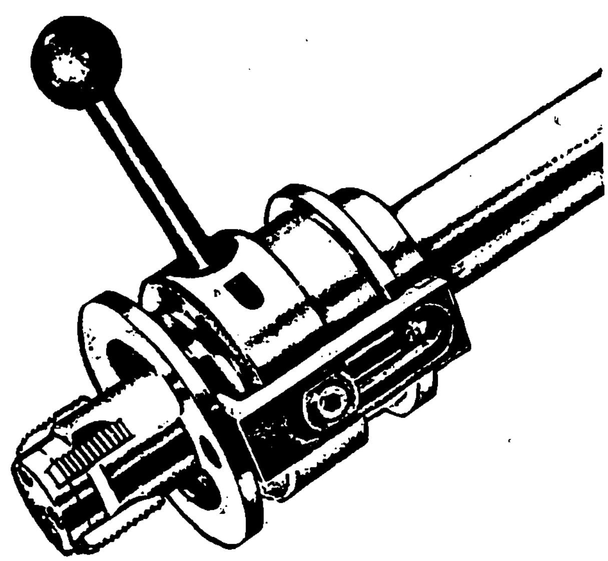
Figure 10-31.—Circular forming cutter diagram.

The solid adjustable taps and dies should be used in place of collapsing taps and self-opening die heads only when speed is low and when time required for backing out is not important.

Collapsing taps (fig. 10-32) are used for internal threading. They are timesavers because you do not have to reverse the spindle to withdraw the tap. The pull-off trip type, which is collapsed by simply stopping the feed, is the most frequently used.

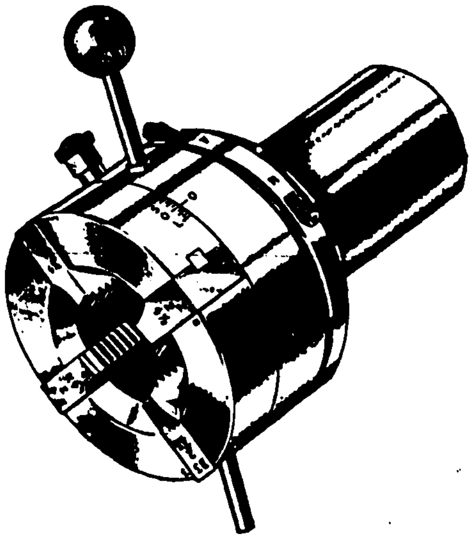
Various types of self-opening die heads are used. One type is shown in figure 10-33. Some have flanged backs for bolting directly to the turret face; others have shanks which fit into a holder. The die heads are fitted with several different types of chasers. The tangential and circular type chasers can be ground repeatedly without destroying the thread shape. They are a bit more difficult to set, but they are better adapted than flat chasers for long runs of identical threads.

Die heads come with either a longitudinal float or a rigid mounting. They may be the pull-off, outside, or inside trip type. The floating type die head should be used for heavy duty turret lathe work, for fine pitch threading, and for finishing rough-cut threads.

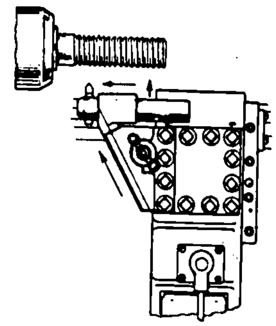


28.347

Figure 10-32.—Universal collapsing tap.



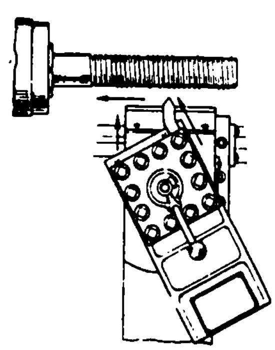
28.348
Figure 10-33.—Pull-off trip self-opening dig head.



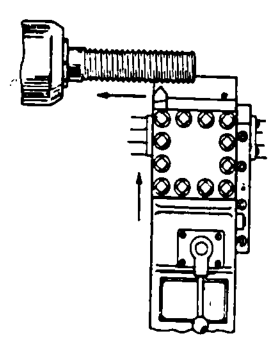
28.190X
Figure 10-35.—Angular feed-in with adjustable threading toolholder.

On some types of work it is necessary to take both roughing and finishing cuts. They are normally taken when threading a tough material or when a smooth finish is required. Some types of die heads have both roughing and finishing attachments. If such die heads are not available, roughing and finishing cuts can be taken with separate dies or taps set up on different turret stations.

As mentioned earlier in this chapter, some horizontal turret lathes can cut or chase threads with a single-point tool. In such machines, there are two methods of feeding the threading tool into the work. The first method is to get an angular feed to the cutter by means of the compound cross-slide (fig. 10-34) or by using the angular threading toolholder (fig. 10-35). By the first method, the cutter is fed into the work at an angle until the final polishing passes are made. For the final polishing passes, the cutter is fed straight in by means of the cross-slide. The second method is to feed the cutter straight into the work for each pass, as indicated in figure 10-36. With this latter method you apply by hand a slight drag to the carriage or saddle during the roughing cut and remove the drag during the final polishing passes. It takes more skill to use the second method, but it produces better threads.



28.189X
Figure 10-34.—Compound cross-slide angular feed-in for thread cutting.



28.191X
Figure 10-3[illegible]ight-in feeding method of threading.

Taper Turning

Tapers may be produced on a turret lathe with (1) forming cutters, (2) roller rest taper turners, or (3) taper attachments.

Forming cutters of the forged, circular, or straight dovetail type may be used to produce tapers when the workpiece is rigid enough or can be supported in such a way that it will withstand the heavy forming cut. If work cannot be formed, other methods (described later) must be used.

Work should be shaped with forming cutters only when the following conditions are met:

1. The work is either self-supporting or is supported by a center rest so that chatter is prevented.

2. The finish must meet requirements.

3. The taper angle must be accurate.

It is best to use the roller rest taper turner for long taper bar jobs. You can quickly set this tool for size by means of the graduated dial and then control the angle of taper accurately by the taper guide bar.

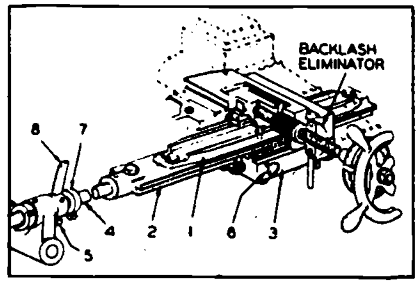
Taper attachments are provided for the cross-slide of most turret lathes of both ram and saddle type. These attachments can be quickly set to produce either internal or external tapers. They do not interfere with normal operation when not in use. Most taper attachments are movable and can be quickly located at any position on the bed.

Taper attachments all have a pivoting guide plate which can be adjusted to any taper angle. Figure 10-37 shows a saddle-type taper attachment in detail.

Figure 10-37 is broken down for study as follows:

a. The guide plate (1) pivots on the base plate (2), which slides in carriage plate (3).

b. When you plan to use the attachment, clamp the extension rod (4) to the machine with



1. GUIDE PLATE
2. BASE PLATE
3. CARRIAGE PLATE
4. EXTENSION ROD
5. SETSCREW
6. BINDER SCREW
7. STOP COLLAR
8. LATCH

28.192X

Figure 10-37.—Detail of cross-slide taper attachment for saddle type machine.

the setscrew (5), and loosen the binder screw (6).

c. You can use the stop collar (7) and the latch (8) for locating the cross-slide unit on the bed of the machine. To use the stop collar and the latch, move the cross-slide unit to the left until the stop collar comes in contact with the latch. This locates the entire unit.

Taper attachments are fitted with a backlash eliminator nut (fig. 10-37) for the slide screws. Tightening this nut against the feed screw removes all play between the feed screw and the nut.

To duplicate and maintain accurate sizes when using a taper attachment with other tools in a setup, remember these three things; (1) you must accurately locate the attachment in the same position in relation to the cross-slide each time it is used, (2) you must locate the cross-slide in exactly the same spot on the bed when you clamp the extension rod with the setscrew, tighten the binder screw, and loosen the extension rod, and (3) be sure the cross-slide is in exactly the same position as in (1) above.

You can produce either internal or external threads with the taper attachment in

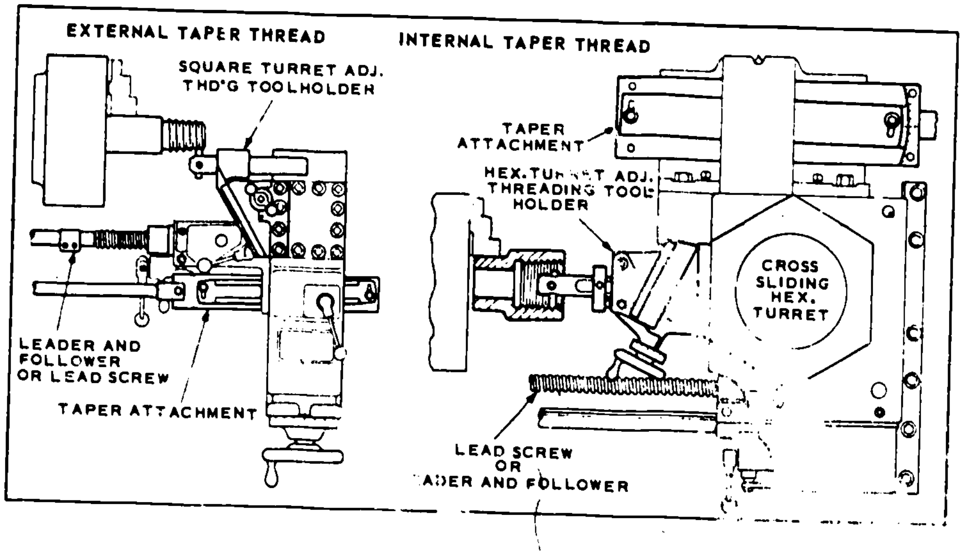


Figure 10-38.—External and internal taper threading.

28.193X

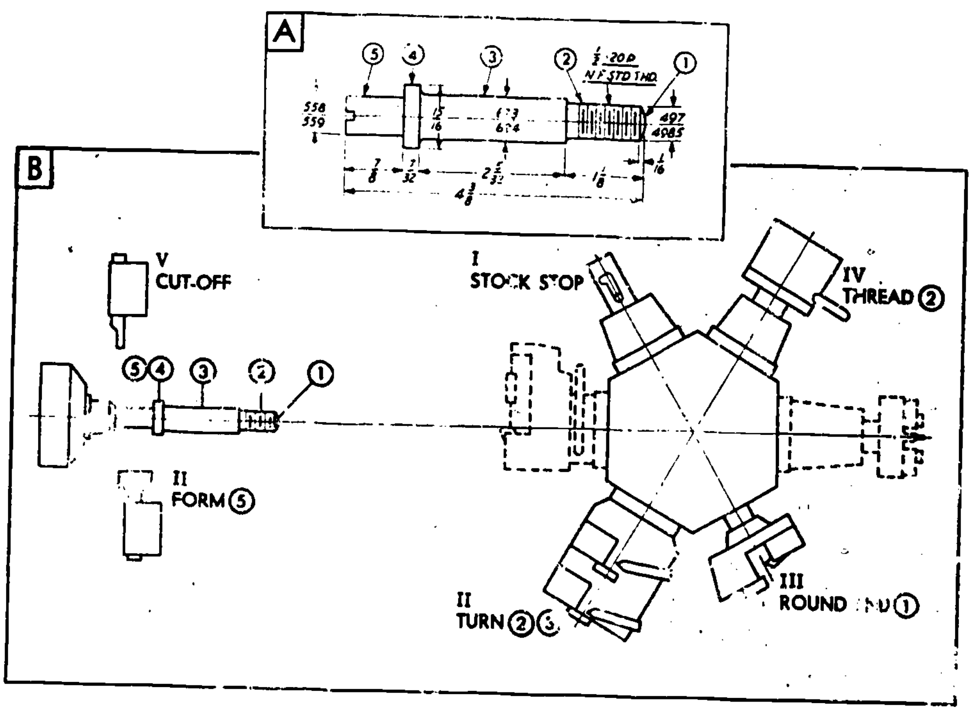


Figure 10-39.—A. A shoulder stud. B. Tooling setup for shoulder stud.

126.10X

260

conjunction with a lead screw thread chasing attachment. (See fig. 10-38). Notice, however, that taper cutting with hexagonal turret held cutters is possible only on lathes that have a cross-sliding hexagonal turret.

HORIZONTAL TURRET LATHE TYPE WORK

The following information is included to give you some examples of turret lathe type work. Regardless of the job, your aim as a good turret lathe operator is to tool up the machine and operate it so that the job can be turned out as rapidly and as accurately as possible. The following examples show you how.

A Shoulder Stud Job

A shoulder stud, shown in part A of figure 10-39, is a typical bar job (universal bar equipment is used) for a small ram type turret lathe that has a screw feed cross-slide. The tooling setup for the shoulder stud is shown in part B of figure 10-39. The diameter (5), which must be held to a clearance of 0.001-inch tolerance, is formed with a cutter on the front of the cross-slide. Diameters (2) and (3) are turned from the hexagon turret with cutters held in the multiple cutter turner. After this operation, the radius on the end of the workpiece is machined in a combination end facer and turner, then the thread is cut, and the piece is cut off.

A Tapered Stud Job

A tapered stud, shown in part B of figure 10-40, does not offer much opportunity for taking multiple cuts. However, cuts from the cross-slide can be combined with cuts taken by the hexagon turret. The tooling setup for the taper stud, shown in part A of figure 10-40, is used for small lot production. The almost identical tooling layout in part C of figure 10-40 shows the setup for medium quantity production.

In both small and medium lot production, the turning of diameter (6) and the forming of diameter (7) can be combined with the turning of diameter (3). In addition, the facing and chamfering of the end (2) can be combined with the turning of diameter (7).

For small lot production (part A of fig. 10-40) the taper is generally formed with a standard wide cutter, ground to the proper angle, and then the cut is matched. These matched cuts will not be very accurate, but as the taper will be ground in a later operation, the job will be satisfactory if sufficient stock is left for grinding. If a forming tool wide enough to cut the taper in one cut is available, it should be used.

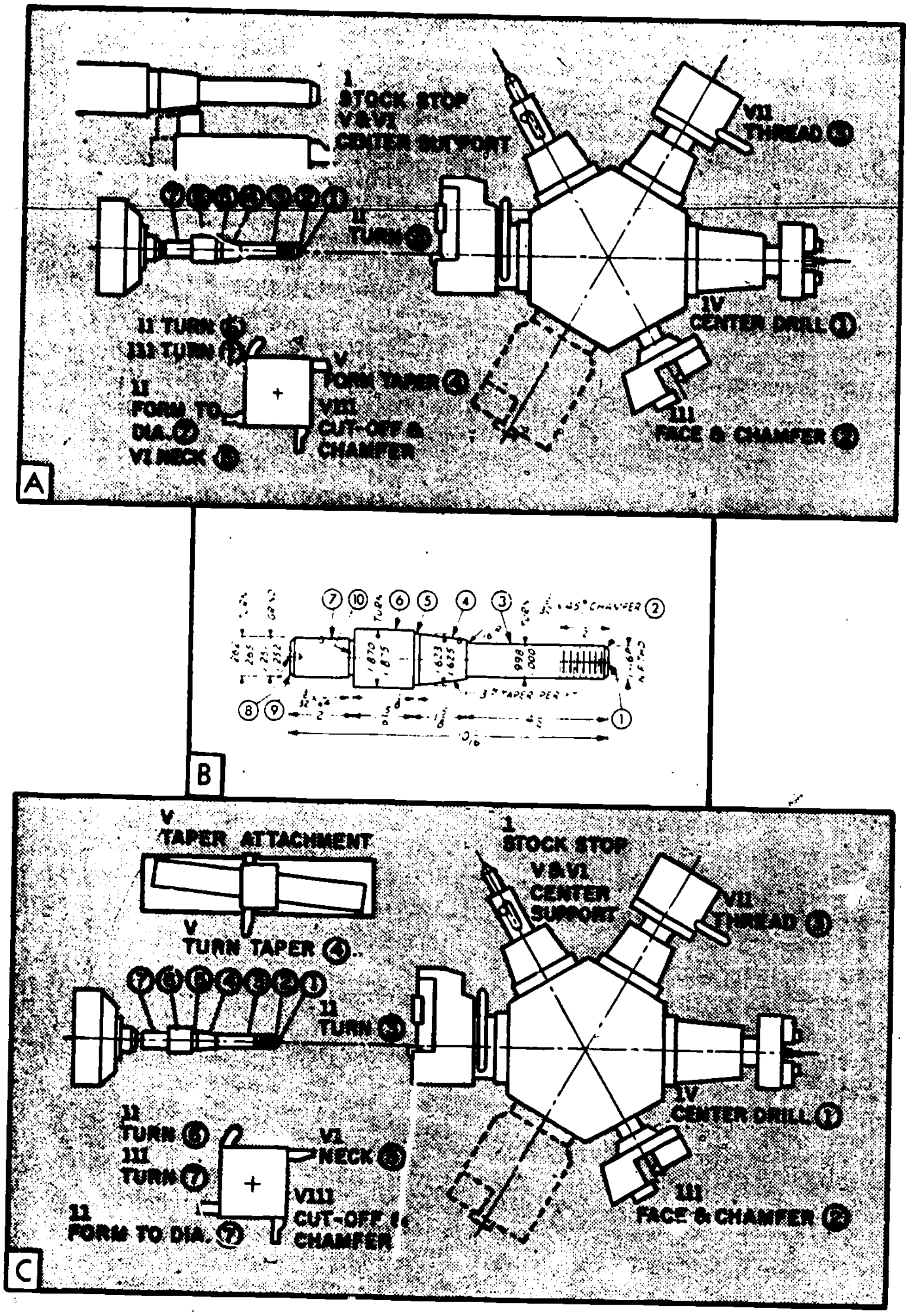
For medium lot production (part C of fig. 10-40) the cross-slide taper attachment may be set up and used for single point turning of the taper. The same amount of time will probably be required to turn the taper (part C. fig. 10-40) as to form the taper (part A, fig. 10-40). However, the turned taper will be more accurate and require less stock for grinding. In addition, the grinding operation will take less time.

Figure 10-41 shows a simple setup for the second operation of the taper stud. The setup is the same for producing either a small or a medium size quantity.

VERTICAL TURRET LATHES

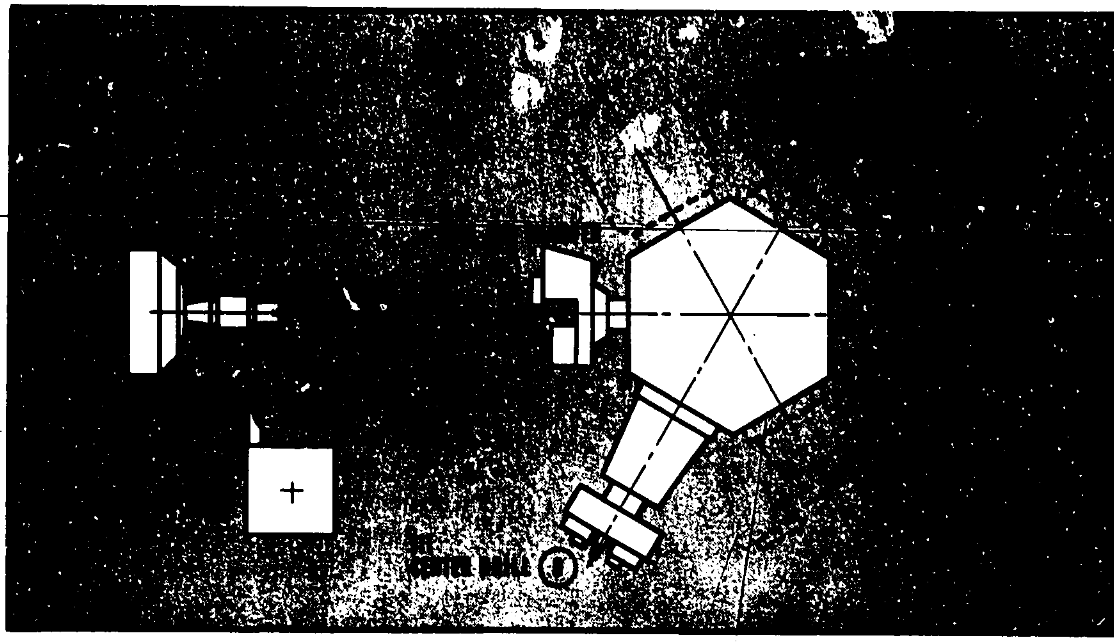
A vertical turret lathe works much like an engine lathe turned up on end. You can perform practically all of the typical lathe operations in a vertical turret lathe, including turning, facing, boring, machining tapers, and cutting internal and external threads.

The characteristic features of this machine are: (1) a horizontal table or faceplate that holds the work and rotates about a vertical axis; (2) a side head that can be fed either horizontally or vertically; and (3) a turret slide, mounted on a crossrail that can feed nonrotating tools either vertically or horizontally.



126.11X

Figure 10-40.—A. Tooling setup for taper stud—small lot production. B. A taper stud. C. Tooling setup for taper stud—medium lot production.



126.12X

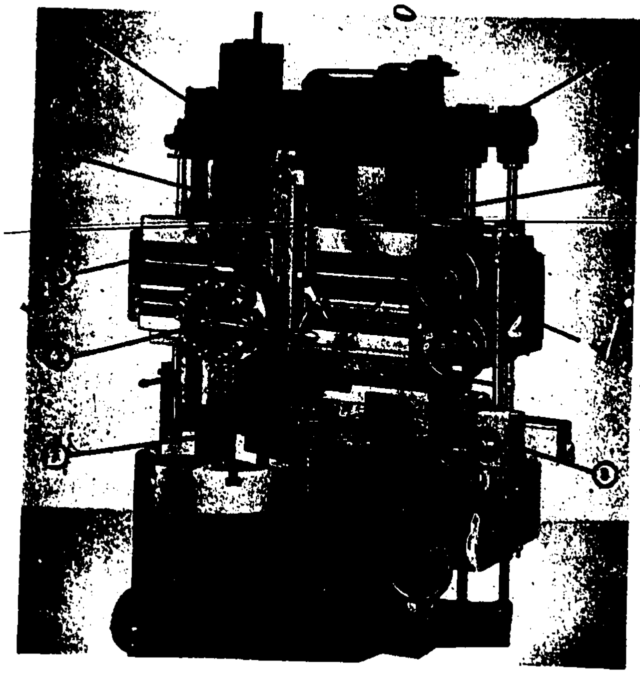
Figure 10-41.–Tooling setup for second chucking on taper stud.

Figures 10-42 and 10-43 show vertical turret lathes similar to those generally found in repair ships and tenders. The main advantage of the vertical turret lathe over the engine lathe is that heavy or awkward parts are easier to set up on the vertical turret lathe and, generally, the vertical turret lathe will handle much larger workpieces than the engine lathe. The size of the vertical turret lathe is designated by the diameter of the table. For instance, a 30-inch lathe has a table 30 inches in diameter. The capacity of a specific lathe is related to but not necessarily limited to the size of the table. A 30-inch vertical lathe (fig. 10-42) can hold and machine (using both the main and the side turrets) a workpiece up to 34 inches in diameter. If only the main turret is used, the workpiece can be as large as 44 inches in diameter.

The main difference between the vertical turret lathe and the horizontal turret lathe is in the design and operating features of the main turret head. Refer to figure 10-42. Note that the turret slide (2) is mounted on a swivel plate (3) which is attached to the saddle (4). The swivel plate allows the turret slide to be swung up to 45° to the right or left of the vertical, depending on the machine model. The saddle is carried on and can traverse, the main rails (5). The main rails are gibbed and geared to the upright bedways (6) for vertical movement. This arrangement allows main turret tools to be fed either vertically or horizontally, as compared to the single direction movement of the hexagonal turret on the horizontal turret lathe. Also, tapers can be cut by setting the turret slide at a suitable angle.

The side turret and side head of the vertical turret lathe correspond to the square turret and cross-slide of the horizontal turret lathe. A typical vertical turret lathe has a system of feed trips and stops which function similarly to those on a horizontal turret lathe. In addition, the machine has feed disengagement devices to prevent the heads from going beyond safe maximum limits and bumping into each other.

Vertical turret lathes have varying degrees of capabilities, including feed and speed ranges,



(1) Main turret head
(2) Turret slide
(3) Swivel plate
(4) Saddle
(5) Main rails
(6) Upright bedways
(7) Side turret
(8) Side head.

28.170X
Figure 10-42.—A 30-inch vertical turret lathe.

angular turning limits and special features, such as threading.

You can expect to find a more coarse minimum feed on the earlier models of vertical turret lathes. Some models have a minimum of 0.008 inch per revolution of the table or chuck, while other models will go as low as 0.001 inch per revolution. The maximum feeds obtainable vary considerably also; however, this is usually less of a limiting factor in job application.

The speeds available on any given vertical turret lathe tend to be much slower than those available on a horizontal lathe. This reduction of speed is often required due to the large and odd-shaped sizes of work done on vertical turret lathes in Navy machine shops. A high speed could cause a workpiece to be thrown out of the machine, causing considerable equipment damage and possible injury to the machine operator or bystanders.

One of the major differences in operator controls between the earlier (fig. 10-42) and the later (fig. 10-43) model vertical turret lathes is in the method used to position the cutter to the work. Earlier models have a handwheel for manually positioning the cutter; later models have an electric drive controlled by a lever. When the feed control lever is moved to the creep position, the turret head moves in the direction selected in increments as low as 0.0001 inch per minute. This creep feed is independent of table revolution and can be made with the table stopped.

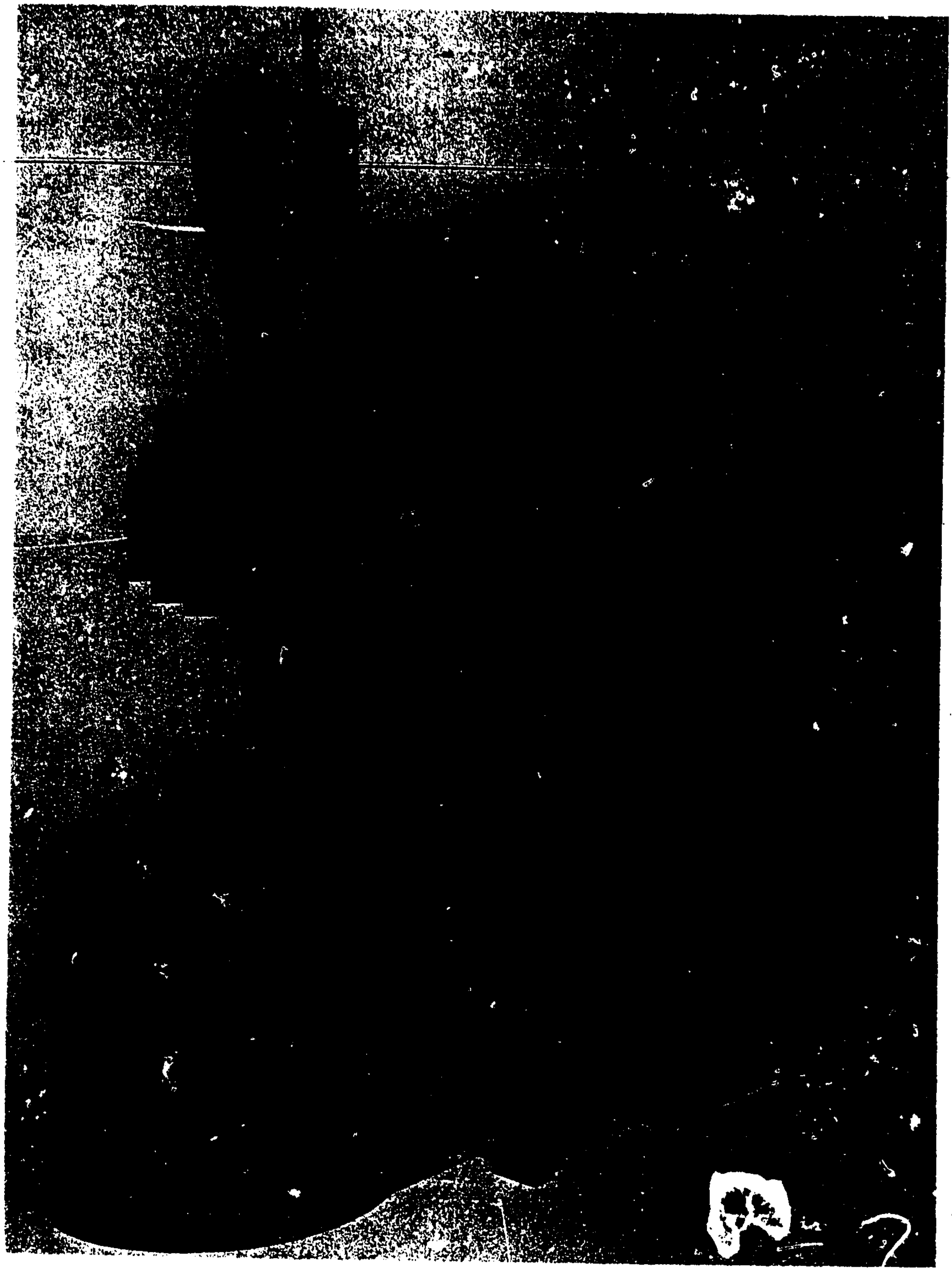
An attachment available on some machines permits threading with a single point tool of up to 32 threads per inch. The gears, as specified by the lathe manufacturer, are positioned in the attachment to provide a given ratio between the revolutions per minute of the table and the rate of advance of the tool.

The same attachment also lets the operator turn or bore an angle of 1° to 45° in any quadrant by positioning certain gears in the gear train. The angle is then cut by engaging the correct feed lever.

Details for turning tapers on a vertical turret lathe without this attachment are given later in the chapter.

TOOLING VERTICAL TURRET LATHES

The principles involved in the operation of a vertical turret lathe are not very different from those just described for the horizontal turret lathe. The only significant difference between the machines, aside from the machine being vertical, is in the main turret. As previously mentioned, you can feed the main head, which corresponds to the hexagonal turret of the horizontal machine, vertically toward the headstock (down); horizontally; or at an angle,



28.349

Figure 10-43.–A 36-inch vertical turret lathe.

either by engaging both the horizontal and vertical feeds or by setting the turret slide at an angle from the vertical and using the vertical feed only.

The tool angles for the cutters of the vertical machine correspond to those used on cutters in the horizontal turret lathe and are an important factor in successful cutting. Also, the same importance is attached to setting cutters on center and maintaining the clearance and rake angles in the process. Again, we cannot overemphasize the importance of holding the cutters rigidly.

In vertical turret lathe work, you must often use offset or bent shank cutters, special sweep tools, and forming tools, particularly when machining odd-shaped pieces. Many such cutting tools are designed to take advantage of the great flexibility of operation provided in the main head.

In a repair ship, the vertical turret lathe is normally used for jobs other than straight production work. For example, a large valve can be mounted on the horizontal face of its worktable or chuck much more conveniently than in almost any other type of machine used to handle large work. Figure 10-44 shows a typical valve seat refacing job in progress in a vertical turret lathe. Figure 10-45 shows the double tooling principle applied to a machining operation.

The tooling principles and the advantage of using coolants for cutting as previously described for horizontal turret lathes apply equally to vertical machines.

TAPER TURNING ON A VERTICAL TURRET LATHE

The following information regarding taper turning on a vertical lathe is based on a Bullard vertical turret lathe. (See fig. 10-42).

There are several ways to cut a taper on a vertical turret lathe. You can cut a 45° taper with either a main turret-held cutter or a side head-held cutter by engaging the vertical and horizontal feeds simultaneously. To cut a taper of less than 30° with a main turret-held tool, set the turret slide for the correct degree of taper and use only the vertical feed for the slide. The operation corresponds to cutting a taper by using the compound rest on an engine lathe; the only difference is that you use the vertical

28.194

Figure 10-44.—Refacing a valve seat in a vertical turret lathe.

28.195

Figure 10-45.—Double tooling.

power feed instead of advancing the cutter by manual feed.

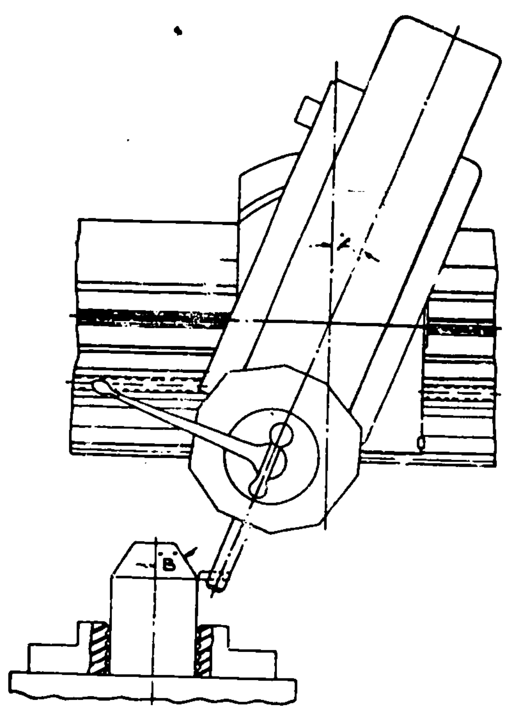
By swiveling the main turret head, you can cut 30° to 60° angles on the vertical turret lathe without the use of special attachments. To machine angles greater than 30° and less than 60° from the vertical, engage both horizontal feed and the vertical feed simultaneously and swivel the head. The angle to which you swivel the head is determined in the following manner. For angles between 30° and 45°, swivel the head in the direction opposite to the taper angle being turned, as illustrated in figure 10-46. The formula for determining the proper angle is A = 90° - 2B°. A sample problem from figure 10-46 follows:

Formula	A = 90° - 2B°
Example	B = 35°
Therefore	A = 90° - (2 X 35)
	A = 90° - 70°
ANGLE	A = 20°

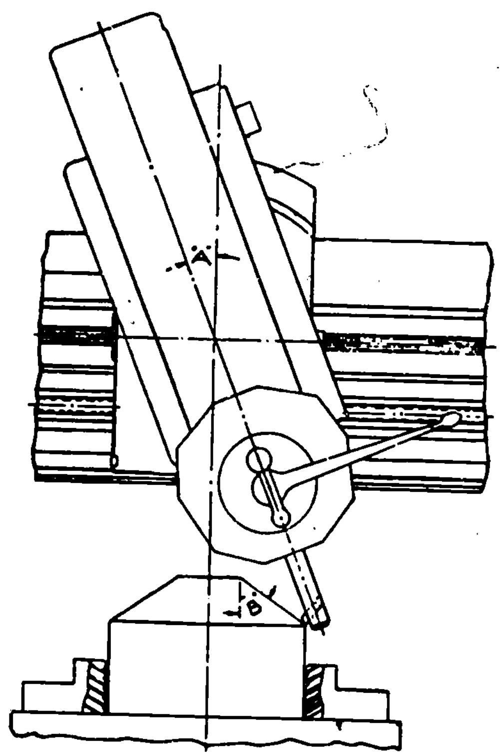
For angles greater than 45° and not more than 60°, swivel the head in the same direction as the taper angle being turned. (See fig. 10-47). The formula for determining the proper angle is ANGLE A = 2B° - 90°. A sample problem from figure 10-47 follows:

Formula	A = 2B° - 90°
Example	B = 56
Therefore	A = (2 X 56°) - 90°
	A = 112° - 90°
ANGLE	A = 22°

In connection with taper turning by the main turret slide swiveling method, you must use great care to set the slide in a true vertical position after completing the taper work and



28.316X
Figure 10-46.—Head setting for 30° to 45° angles.



28.317X
Figure 10-47.—Head setting for 45° to 60° angles.

before using the main head for straight cuts. A very small departure of the slide from the true vertical will produce a relatively large taper on straight work. Unless you are alert to this, you may inadvertently cut a dimension undersize before you are aware of the error.

Still another means of cutting tapers with either a main head-held or side head-held tool is to use a sweep-type cutter ground and set to the desired angle. Then feed it straight to the work to produce the desired tapered shape. This, of course, is feasible only for short taper cuts.

CHAPTER 11

MILLING MACHINES AND MILLING OPERATIONS

The milling machine removes metal with a revolving cutting tool called a milling cutter. With various attachments, milling machines can be used for boring, slotting, circular milling, dividing, and drilling; cutting keyways, racks, and gears; and fluting taps and reamers.

Bed type and knee and column type milling machines are generally found in most Navy machine shops. The bed type milling machine has a vertically adjustable spindle. The horizontal boring mill discussed later in this chapter is representative of the bed type. The knee and column milling machine has a fixed spindle and a vertically adjustable table. There are several classes of milling machines within these types but only those classes with which you will be concerned are discussed in this chapter.

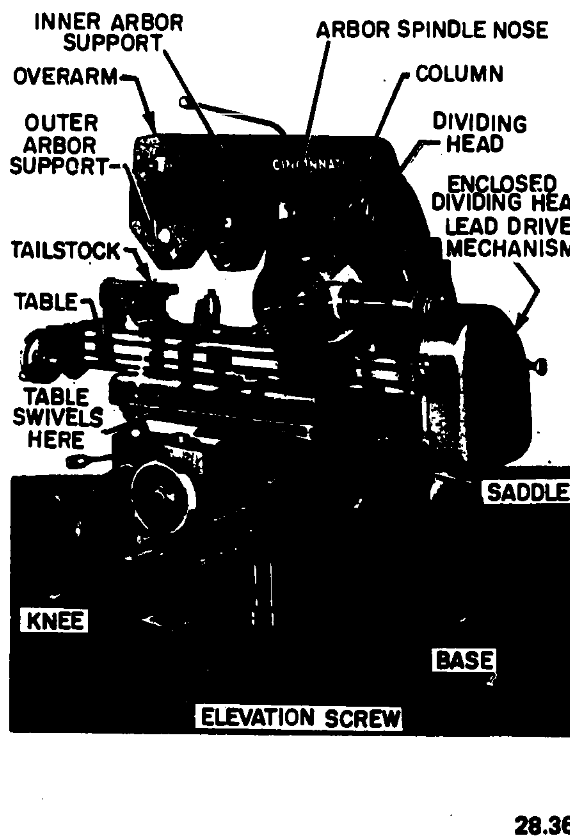
You must be able to set up the milling machine to machine flat, angular, and formed surfaces. Included in these jobs are milling keyways, milling hexagonal and square heads on nuts and bolts, milling T-slots and dovetails, and milling spur gear teeth. To set up a milling machine, you must compute feeds and speeds, select and mount the proper holding device, and select and mount the proper cutter to handle the job.

Like other machines in the shop, milling machines have manual and power feed systems, a selective spindle speed range and a coolant system.

KNEE AND COLUMN MILLING MACHINES

The Navy uses three types of knee and column milling machines: the universal type, the plain type, and the vertical spindle type. Wherever only one type of machine can be installed, the universal type is usually selected.

The UNIVERSAL MILLING MACHINE (fig. 11-1) has all the principal features of the other types of milling machines. It can handle practically all classes of milling work. You can take vertical cuts by feeding the table up or down. You can move the table in two directions in the horizontal plane: either at right angles to



28.362

Figure 11-1.—Universal milling machine.

the axis of the spindle or parallel to the axis of the spindle. The principal advantage of the universal mill over the plain mill is that you can swivel the table on the saddle. Thus, you can move the table in the horizontal plane at an angle to the axis of the spindle. This machine is used to cut most types of gears, milling cutters, and twist drills, and is used for various kinds of straight and taper work.

The PLAIN MILLING MACHINE is the simplest milling machine since it has only a few of the features found on the other machines. You can move the table in three directions: longitudinally (at right angles to the spindle), transversely (parallel to the spindle), and vertically (up and down). The ability of this machine to take heavy cuts at fast speeds is its chief value which is made possible by the machine's rigid construction.

The VERTICAL SPINDLE MILLING MACHINE (fig. 11-2) has the spindle in a vertical position and at a right angle to the

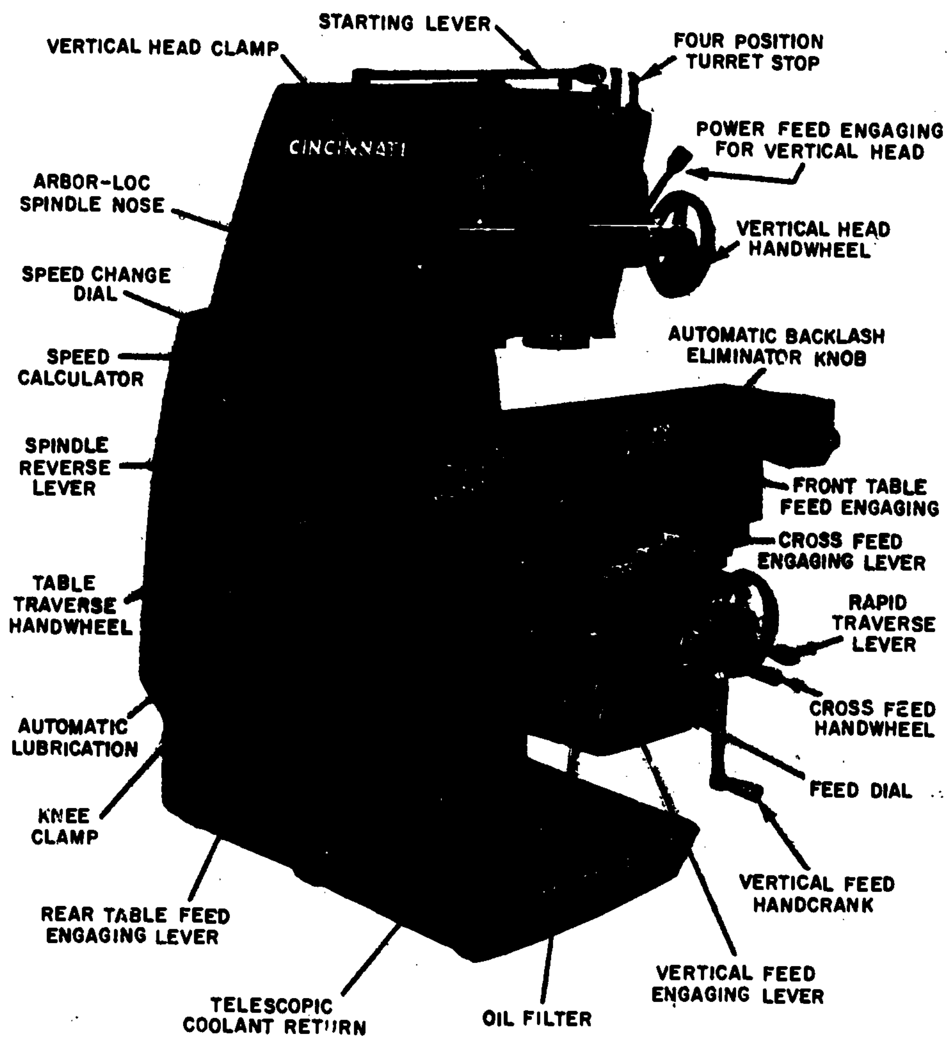
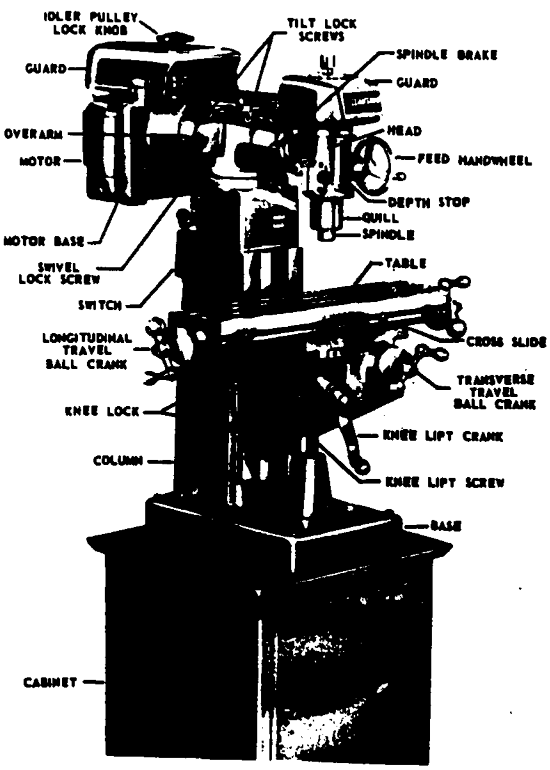


Figure 11-2.—Vertical spindle milling machine.

surface of the table. The spindle has a vertical movement, and the table can be moved vertically, longitudinally, and transversely. Both the spindle and table movement can be controlled manually or by power. The vertical-spindle milling machine can be used for face milling, profiling, die sinking, and for various odd-shaped jobs; it can also be used to advantage in boring holes. Various small vertical spindle milling machines (fig. 11-3) are also available for light precision milling operations.

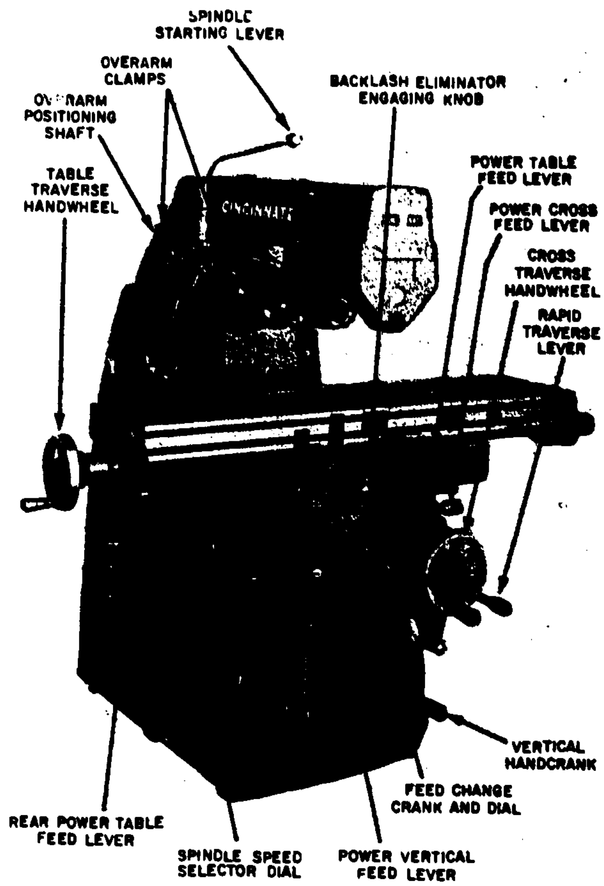
MAJOR COMPONENTS

You must know the name and purpose of each of the main parts of a milling machine to understand the operations discussed later in this chapter. Keep in mind that, although we are discussing a knee and a column milling machine, you can apply most of the information to tne other types. Figure 11-4, which illustrates a



28.364

Figure 11-3.—Small vertical milling machine (Atlas Press).



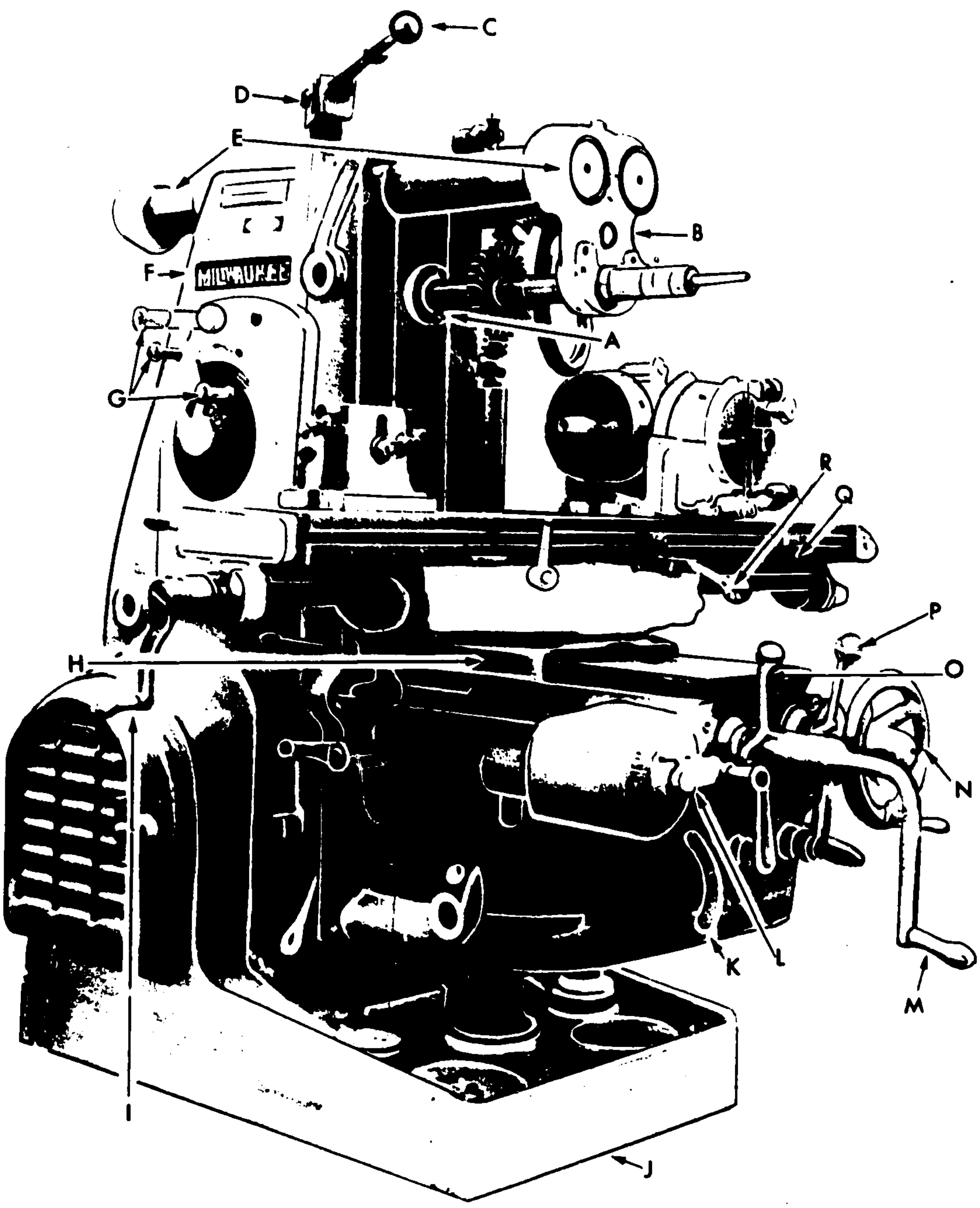
28.365

Figure 11-4.—Plain milling machine, showing operation controls (Cincinnati Milling Machine Co.).

plain knee and column milling machine and figure 11-5, which illustrates a universal knee and column milling machine, will help you to become familiar with the location of the parts.

COLUMN: The column, including the base, is the main casting which supports all the other parts of the machine. An oil reservoir and a pump in the column keep the spindle lubricated. The column rests on a base that contains a coolant reservoir and a pump that you can use when performing any machining operation that requires a coolant.

KNEE: The knee is the casting that supports the table and saddle. The feed change gearing is



A. SPINDLE
B. ARBOR SUPPORT
C. SPINDLE CLUTCH LEVER
D. SWITCH
E. OVERARM
F. COLUMN
G. SPINDLE SPEED SELECTOR LEVERS
H. SADDLE AND SWIVEL
I. LONGITUDINAL HANDCRANK
J. BASE
K. KNEE
L. FEED DIAL
M. KNEE ELEVATING CRANK
N. TRANSVERSE HANDWHEEL
O. VERTICAL FEED CONTROL
P. TRANSVERSE FEED LEVER
Q. TABLE FEED TRIP DOG
R. LONGITUDINAL FEED CONTROL

28.366

Figure 11-5.—Universal knee and column milling machine with horizontal spindle.

enclosed within the knee. It is supported and can be adjusted by the elevating screw. The knee is fastened to the column by dovetail ways. You can raise or lower the knee by either hand or power feed. You usually use hand feed to take the depth of cut or to position the work and power feed to move the work during the machining operation.

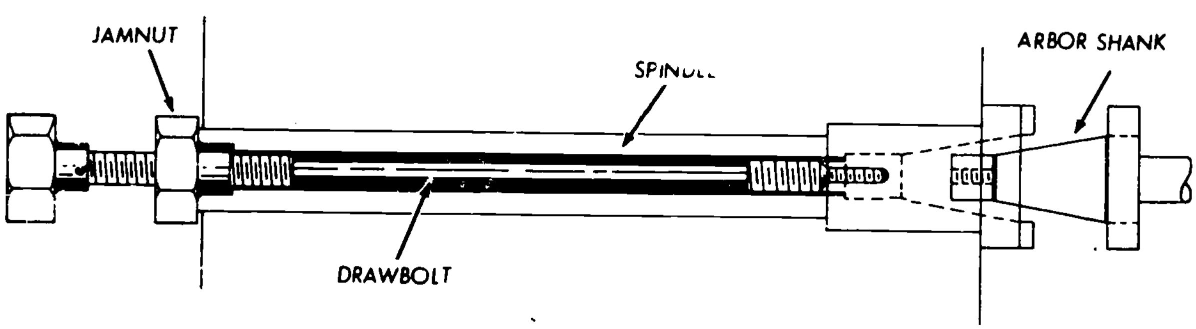
SADDLE and SWIVEL TABLE: The saddle slides on a horizontal dovetail (which is parallel to the axis of the spindle) on the knee. The swivel table (on universal machines only) is attached to the saddle and can be swiveled approximately 45° in either direction.

POWER FEED MECHANISM: The power feed mechanism is contained in the knee and controls the longitudinal, transverse (in and out) and vertical feeds. You can obtain the desired rate of feed on machines, such as the one shown in figure 11-4, by positioning the feed selection levers as indicated on the feed selection plate. On machines such as the one in figure 11-5, you get the feed you want by turning the speed selection handle until the desired rate of feed is indicated on the feed dial. Most milling machines have a rapid traverse lever that you can engage when you want to temporarily increase the speed of the longitudinal, transverse, or vertical feeds. For example, you would engage this lever when positioning or aligning the work. NOTE: For safety reasons, you must exercise extreme caution while using rapid traverse controls.

TABLE: The table is the rectangular casting located on top of the saddle. It contains several T-slots for fastening work or workholding devices to it. You can move the table by hand or by power. To move the table by hand, engage and turn the longitudinal handcrank. To move it by power, engage the longitudinal directional feed control lever. You can position the longitudinal directional feed control lever to the left, to the right, or in the center. Place the end of the directional feed control lever to the left to feed the table toward the left. Place it to the right to feed the table toward the right. Place it in the center position to disengage the power feed or to feed the table by hand.

SPINDLE: The spindle holds and drives the various cutting tools. It is a shaft mounted on bearings supported by the column. The spindle is driven by an electric motor through a train or gears all mounted within the column. The front end of the spindle, which is near the table, has an internal taper machined in it. The internal taper (3 1/2 inches per foot) permits you to mount tapered-shank cutter holders and cutter arbors. Two keys, located on the face of the spindle, provide a positive drive for the cutter holder, or arbor. You secure the holder or arbor in the spindle by a drawbolt and jamnut, as shown in figure 11-6. Large face mills are sometimes mounted directly to the spindle nose.

OVERARM: The overarm is the horizontal beam to which you fasten the arbor support. The overarm (fig. 11-5) may be a single casting



28.367

Figure 11-6.—Spindle drawbolt.

that slides in dovetail ways on the top of the column. It may consist of one or two cylindrical bars that slide through holes in the column, as shown in figure 11-4. To position the overarm on some machines, first unclamp locknuts and then extend the overarm by turning a crank. On others, you move the overarm by simply pushing on it. You should extend the overarm only far enough to position the arbor support to make the setup as rigid as possible. To place arbor supports (B in fig. 11-5) on an overarm such as the one shown in figure 11-5, extend one of the bars approximately 1 inch farther than the other bar. Tighten the locknuts after positioning the overarm. On some milling machines the coolant supply nozzle is fastened to the overarm. You can mount the nozzle with a split clamp to the overarm after you have placed the arbor support in position.

ARBOR SUPPORT: The arbor support is a casting that contains a bearing which aligns the outer end of the arbor with the spindle. This helps to keep the arbor from springing during cutting operations. Two types of arbor supports are commonly used. One type has a small diameter bearing hole usually 1-inch maximum diameter. The other type has a large diameter bearing hole usually up to 2 3/4 inches. An oil reservoir in the arbor support keeps the bearing surfaces lubricated. You can clamp an arbor support any place you want on the overarm. Small arbor supports give additional clearance below the arbor supports when you are using small diameter cutters. Small arbor supports can provide support only at the extreme end of the arbor. For this reason they are not recommended for general use. You can position large arbor supports at any point on the arbor. Therefore, they can provide support near the cutter, if necessary. For this reason, you should position the large arbor support as close to the cutter as possible to provide a rigid tooling setup. Although arbor supports are not classified, a general rule of thumb can be used by the size of the arbor used. For example, old reference type A is small bearing diameter and old reference type B is large bearing diameter. NOTE: Before loosening or tightening the arbor nut, you must install the arbor support. This will prevent bending or springing of the arbor.

SIZE DESIGNATION: All milling machines are identified by four basic factors: size, horsepower, model, and type. The size of a milling machine is based on the longitudinal (from left to right) table travel in inches. Vertical, cross, and longitudinal travel are all closely related as far as overall capacity. For size designation, only the longitudinal travel is used. There are six sizes of knee type milling machines with each number representing the number of inches of travel.

Standard Size	Longitudinal Table Travel
No. 1	22 inches
No. 2	28 inches
No. 3	34 inches
No. 4	42 inches
No. 5	50 inches
No. 6	60 inches

If the milling machine in your shop is labeled No. 2HL, it has a table travel of 28 inches; if it is labeled No. 5LD, it has a travel of 50 inches. The MODEL designation is determined by the manufacturer, and features vary with different brands. The TYPE of milling machine is designated as plain or universal, horizontal or vertical, and knee and column or bed. In addition, machines may have other special type designations.

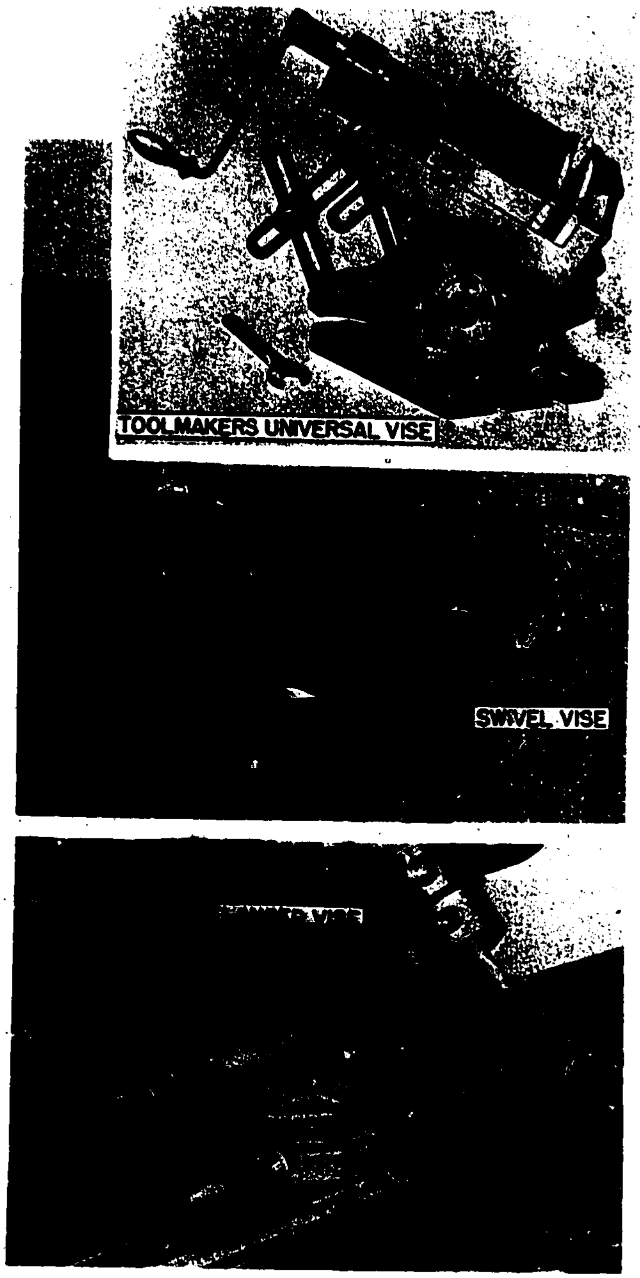
Standard equipment with milling machines used in Navy ships includes workholding devices, spindle attachments, cutters, arbors, and any special tool needed for setting up the machine for milling. This equipment allows for holding and cutting the great variety of milling jobs you will encounter in Navy repair work.

WORKHOLDING DEVICES

The following material describes workholding devices which you may be required to use.

VISES

The vises commonly used on milling machines are the flanged plain vise, the swivel vise, and the toolmaker's universal vise (fig. 11-7). The flanged vise provides a rigid workholding setup. It is used when the surface to be machined must be parallel to the surface seated in the vise. The swivel vise is used similarly to the flanged vise, but the setup is less rigid; the workpiece can be swiveled in a horizontal plane to any required angle. The toolmaker's universal vise is used when the workpiece must be set up at a complex angle in relation to the axis of the spindle and to the table surface.



28.199X

Figure 11-7.—Milling machine vises.

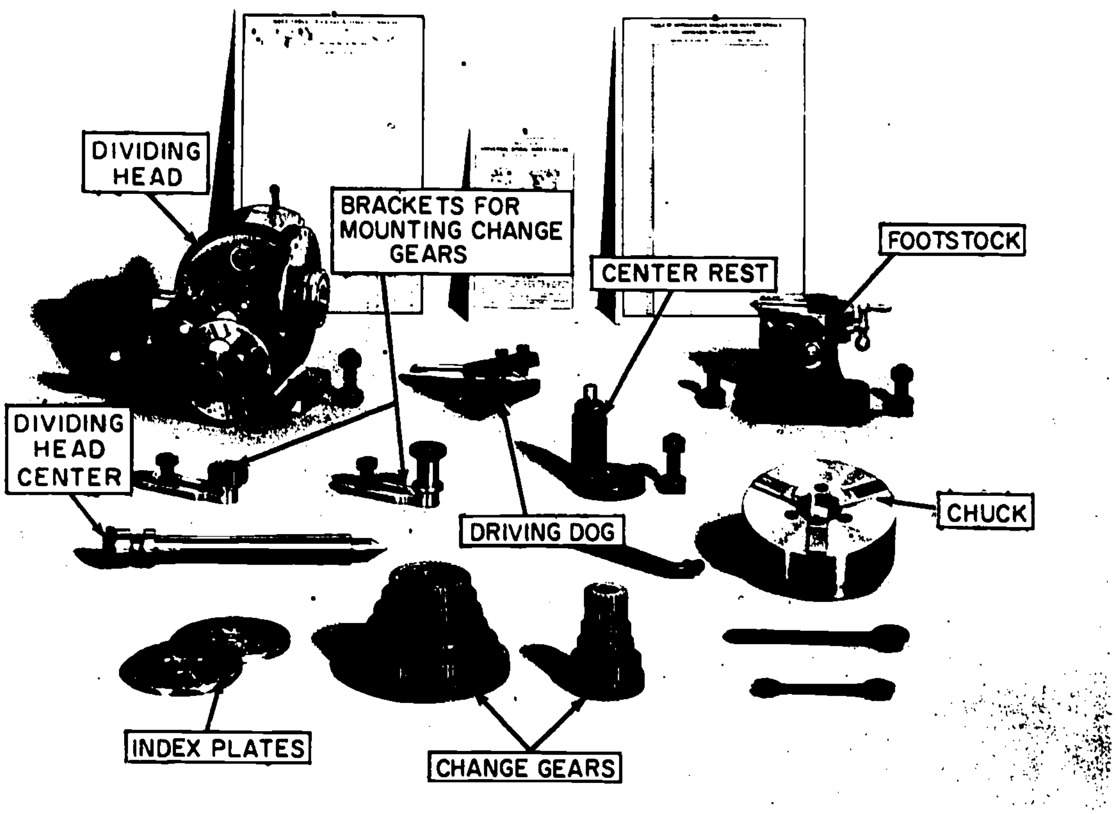
INDEXING EQUIPMENT

Indexing eqiupment (fig. 11-8) is used to hold the workpiece and to provide a means of turning the workpiece so that a number of accurately spaced cuts can be made (gear teeth for example). The workpiece is held in a chuck, attached to the dividing head spindle, or between a live center in the dividing index head and a dead center in the footstock; collets are also used. The center rest can be used to support long slender work. You can raise or lower the center of the footstock for setting up tapered workpieces.

Dividing Head

The internal components of the dividing head are shown in figure 11-9. The ratio between the worm and gear is 40 to 1. By turning the worm one turn, you rotate the spindle 1/40 of a revolution. The index plate has a series of concentric circles of holes. With the index plate you can accurately gage partial turns of the worm shaft and turn the spindle accurately in amounts smaller than 1/40 of a revolution. You can secure the index plate either to the dividing head housing or to the worm shaft. You can adjust the crankpin radially for use in any circle of holes. You can set the sector arms to span any number of holes in the index plate to provide a guide for rotating the index crank for partial turns. You can turn the dividing head spindle directly by hand by disengaging the worm and drawing the plunger back, or by the index crank through the worm and worm gear.

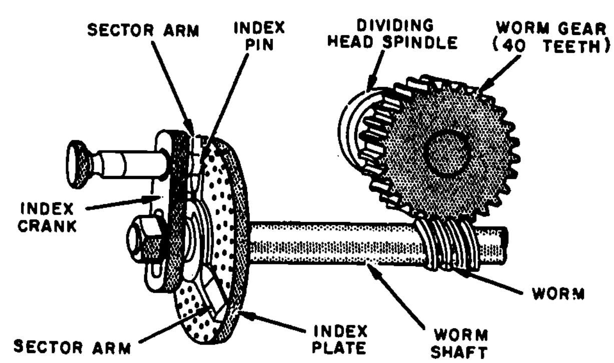
The spindle is set in a swivel block so that you can set the spindle at any angle from



28.200X

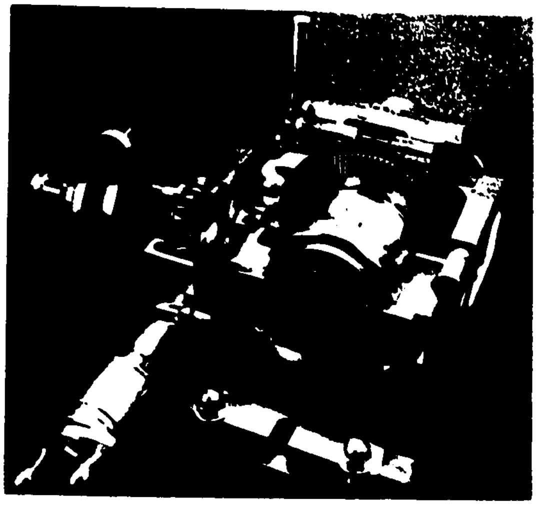
Figure 11-8.—Indexing equipment.

slightly below horizontal to slightly past vertical. There is an index plate that usually has a 24-inch hole circle. Place it back of the chuck or center so that you can index the spindle rapidly by hand for commonly required divisions. Most index heads have a 40-1 ratio. One well-known exception has a 5 to 1 ratio (see fig. 11-10). This ratio is made possible by a 5 to 1 hypoid bevel gear ratio between the index crank and the dividing head spindle. The faster movement of the spindle with one turn of the index crank permits speedier production. It is also an advantage in truing work or testing work for run out with a dial indicator. Although made to a high standard of accuracy, the 5 to 1 ratio dividing head does not permit as wide a selection of divisions by simple indexing operation.



28.201

Figure 11-9.—Dividing head mechanism.



28.368
Figure 11-10.–Universal spiral dividing head with 5 to 1 ratio between spindle and index crank. (Kearney & Trecker Corp.)

Differential indexing can be used on the 5 to 1 ratio dividing head by means of a differential indexing attachment.

The dividing head may also be geared to the lead screw of the milling machine by a driving mechanism and will impart a rotary motion to the work–as required for helical and spiral milling. The index head may have one of several driving mechanisms. The most common of these is the ENCLOSED DRIVING MECHANISM which is standard equipment on some makes of the plain and universal knee and column milling machines. The enclosed driving mechanism has a lead range of 2 1/2 to 100 inches and is driven directly from the lead screw.

Gearing Arrangement

Figure 11-11 illustrates the gearing arrangement used on most milling machines. The gears are marked as follows:

A = Gear on the worm shaft (driven)

B = First gear on the idler stud (driving)

C = Second gear on idler stud (driven)

D = Gear on lead screw (driving)

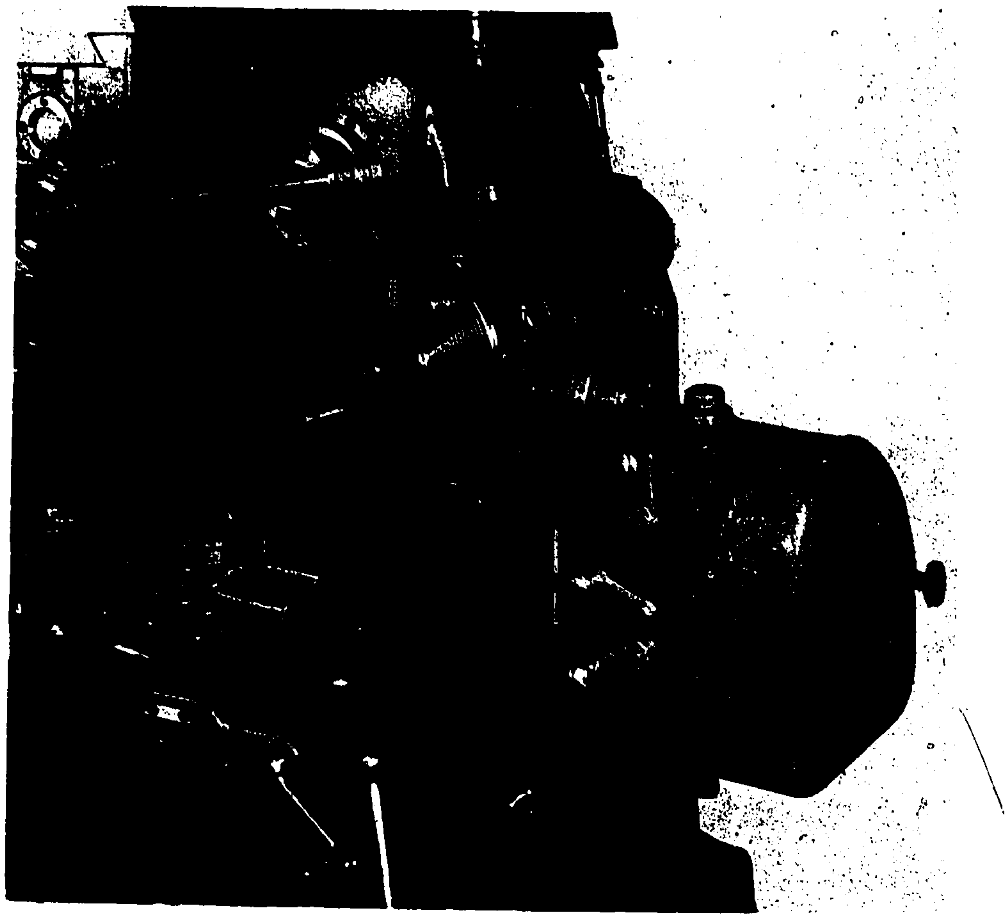
E and F = Idler gears

28.307
Figure 11-11.–Enclosed driving mechanism.

LOW LEAD DRIVE.–For some models and makes of milling machines a low lead driving mechanism is available; however, additional parts must be built into the machine at the factory. This driving mechanism has a lead range of 0.125 to 100 inches.

LONG AND SHORT LEAD DRIVE.–When an extremely long or short lead is required, you can use the long and short lead attachment (fig. 11-12). As with the low lead driving mechanism, the milling machine must have certain parts built into the machine at the factory. In this attachment, an auxiliary shaft in the table drive mechanism supplies power through the gear train to the dividing head. It also supplies the power for the table lead screw which is disengaged from the regular drive when the attachment is used. This attachment provides leads in the range between 0.010 and 1000 inches.

CIRCULAR MILLING ATTACHMENT.–The circular milling attachment, or rotary table



126.27X

Figure 11-12.—The long and short lead attachment.

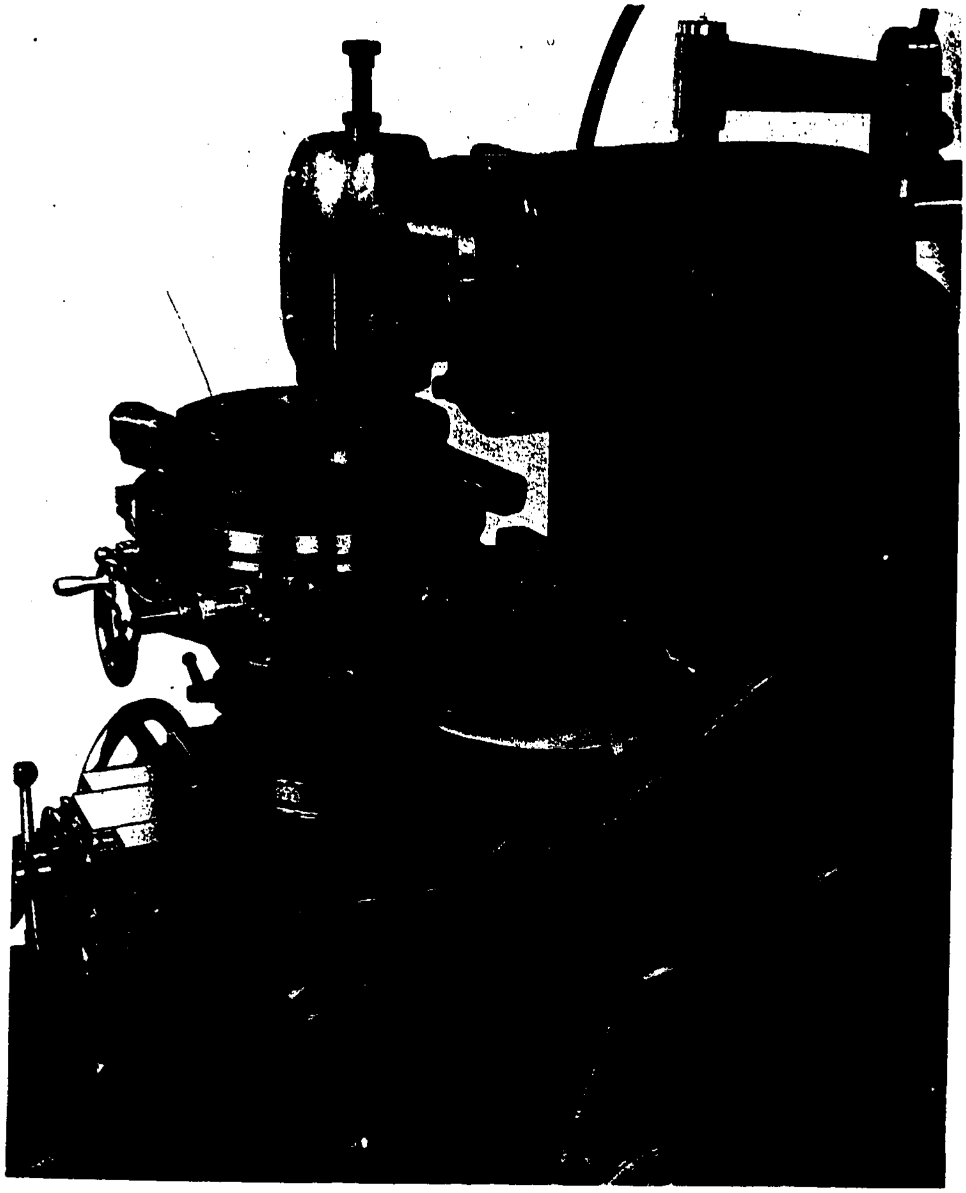
(fig. 11-13), is used for setting up work that must be rotated in a horizontal plane. The worktable is graduated (1/2° to 360°) around its circumference. You can turn the table by hand or by the table feed mechanism through a gear train (fig. 11-13). An 80 to 1 worm and gear drive contained in the rotary table and index plate arrangement makes this device useful for accurate indexing of horizontal surfaces.

SPECIAL ATTACHMENTS

The universal milling (head) attachment, shown in figure 11-14, is clamped to the column of the milling machine. The cutter can be secured in the spindle of the attachment and then can be set by the two rotary swivels so that the cutter will cut at any angle to the horizontal or the vertical plane. The spindle of the universal milling attachment is driven by gearing connected to the milling machine spindle.

SLOTTING ATTACHMENT

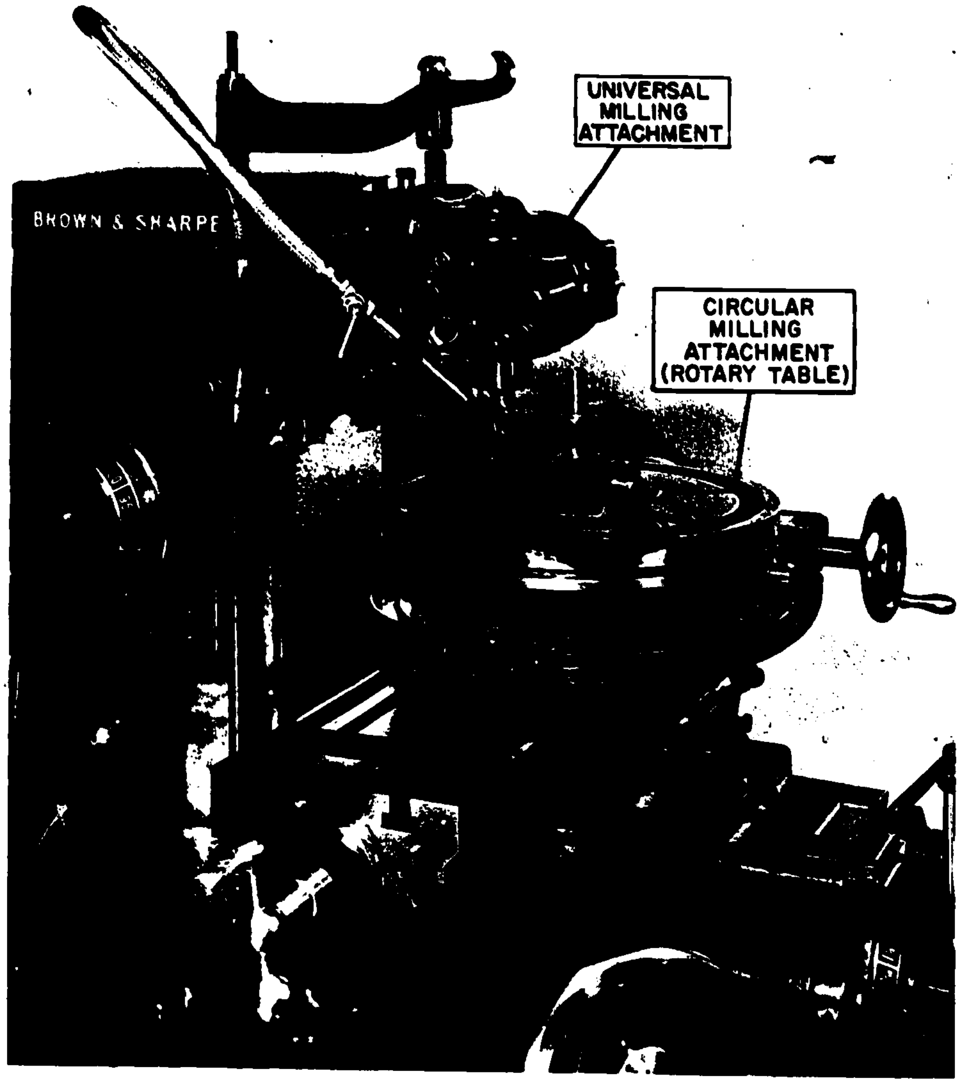
Although special machines are designed for cutting slots (such as keyways and splines), this type machine frequently is not available. Consequently, the machinist must devise other



126.29X
Figure 11-13.—Circular milling attachment, power feed mechanism, and vertical milling attachment.

means. The slotting attachment in figure 11-15, when mounted on the column and spindle of a plain or universal milling machine, will perform such operations.

The attachment is designed so that the rotating motion of the spindle is changed to reciprocating motion of the tool slide on the slotter, similar to the ram on a shaper. A single point cutting tool is used. Since the tool slide can be swiveled through 360°, slotting can be done at any angle, and the stroke can be set from 0 to 4 inches.



28.202X

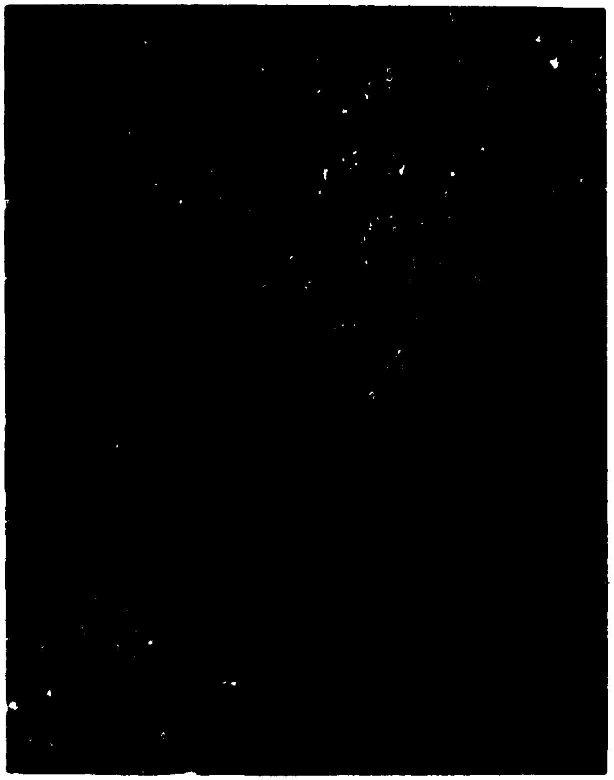
Figure 11-14.—Circular milling attachments (rotary table) and universal (head) attachment.

INDEXING THE WORK

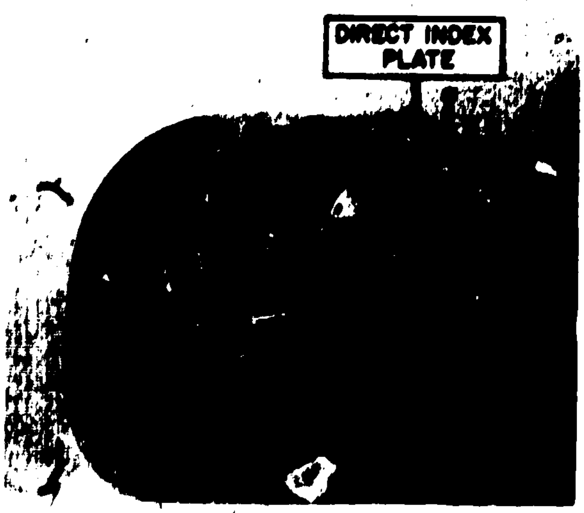
Indexing is done by the direct, plain, compound, or differential methods. The direct and plain methods are the most commonly used; the compound and differential are used only when the job cannot be done by plain or direct indexing.

DIRECT INDEXING

Direct indexing, sometimes referred to as rapid indexing, is the simplest method of indexing. Figure 11-16 shows the front index plate attached to the work spindle. The front index plate usually has 24 equally spaced holes. These holes can be engaged by the front index



28.369
Figure 11-15.–Slotting a bushing using a slotting attachment. (Cincinnati Milling Machine Co.)



28.209X
Figure 11-16.–Direct index plate.

pin, which is spring-loaded and moved in an out by a small lever. Rapid indexing requir that the worm and worm wheel be disengaged that the spindle can be moved by han Numbers that can be divided into 24 can l indexed in this manner. Rapid indexing is use when a large number of duplicate parts are to l milled.

To find the number of holes to move in th index plate, divide 24 by the number divisions required.

Number of holes to move = $\frac{24}{N}$ where N required number of N divisions

Example: Indexing for a hexagon head bol

because a hexagon head has six flats,

$$\frac{24}{N} = \frac{24}{6} = 4 \text{ holes}$$

IN ANY INDEXING OPERATIO ALWAYS START COUNTING FROM TH HOLE ADJACENT TO THE CRANKPIN During heavy cutting operations, clamp th spindle by the clamp screw to relieve strain o the index pin.

PLAIN INDEXING

Plain indexing, or simple indexing, is use when a circle must be divided into more part than is possible by rapid indexing. Simpl indexing requires that the spindle be moved by turning an index crank, which turns the worm that is meshed with the worm wheel. The ratio between worm and worm wheel is 1 to 40 (1:40). One turn of the index crank turns the index head spindle 1/40 of a complete turn Forty turns of the index crank will revolve the spindle chuck and job one complete turn. To determine the number of turns or fractiona parts of a turn of the index crank necessary to

301

cut any required number of divisions, divide 40 by the number of divisions required.

$$\text{Number of turns of the index crank} = \frac{40}{N}$$

where N = number of divisions r᷊ ired

Example (1): Index for five divisions

$$\frac{40}{N} = \frac{40}{5} = 8 \text{ turns}$$

There are eight turns of the crank for each division.

Example (2): Index for eight divisions

$$\frac{40}{N} = \frac{40}{8} = 5 \text{ turns}$$

Example (3): Index for ten divisions

$$\frac{40}{N} = \frac{40}{10} = 4 \text{ turns}$$

When the number of divisions required does not divide evenly into 40, the index crank must be moved a fractional part of a turn with index plates. A commonly used index head comes with three index plates. Each plate has six circles of holes which we shall use as an example.

Plate one:	15-16-17-18-19-20
Plate two:	21-23-27-29-31-33
Plate three:	37-39-41-43-47-49

The previous examples of the use of the indexing formula $\frac{40}{N}$ gave results in complete turns of the index crank. This seldom happens on the typical indexing job. For example, indexing for 18 divisions

$$\frac{40}{N} = \frac{40}{18} = 2\frac{4}{18} \text{ turns}$$

The whole number indicates the complete turns of the index crank, the denominator of the fraction represents the index circle, and the numerator represents the number of holes to use on that circle. Because there is an 18-hole index circle, the mixed number 2 4/18 indicates that the index crank will be moved 2 full turns plus 4 holes on the 18-hole circle. The sector arms are positioned to include 4 holes and the hole in which the index crank pin is engaged. The number of holes (4) represents the movement of the index crank; the hole that engages the index crank pin is not included.

When the denominator of the indexing fraction is smaller or larger than the number of holes contained in any of the index circles, change it to a number representing one of the circle of holes. Do this by mulitplying or dividing the numerator and the denominator by the same number. For example, to index for the machining of a hexagon (N = 6):

$$\frac{40}{N} = \frac{40}{6} = 6\frac{4}{6} \text{ turns}$$

In its simplest form, this is 6 2/3 turns. The denominator 3 will divide equally into the following circles of holes:

Plate one	15 and 18
Plate two	21 and 33
Plate three	39

If plate 3 is conveniently on the index head, you should use it. You must multiply the denominator 3 by 13 to equal 39. In order not to change the value of the original indexing fraction, you must multiply both the numerator and the denominator by 13

$$\frac{2}{3} \times \frac{13}{13} = \frac{26}{39} \qquad 6\frac{2}{3} = 6\frac{26}{39}$$

Thus, to mill each side of a hexagon, you must move the index crank 6 full turns and 26 holes on a 39-hole circle.

When the number of divisions exceeds 40, you may divide both terms of the fraction by a common divisor to obtain an index circle that is available. For example, if 160 divisions are required, N = 160; the fraction to be used is

$$\frac{40}{N} \ \frac{40}{160}$$

Because there is no 160-hole circle this fraction must be reduced.

$$\frac{40/10}{160/10} = \frac{4}{16}$$

Turn 4 holes on the 16-hole circle.

It is usually more convenient to reduce the original fraction to its lowest terms and multiply both terms of the fraction by a factor that will give a number representing a circle of holes.

$$\frac{40/40}{160/40} = \frac{1}{4}$$

$$\frac{1}{4} \times \frac{4}{4} = \frac{4}{16}$$

The following examples will further clarify the use of this formula.

Example 1: Index for 9 divisions.

$$\frac{40}{N} = \frac{40}{9} = 4\frac{4}{9}$$

If an 18-hole circle is used, the fraction becomes $\frac{4}{9} \times \frac{2}{2} = \frac{8}{18}$. For each division turn the crank 4 turns and 8 holes on an 18-hole circle.

Example 2: Index for 136 divisions.

$$\frac{40}{N} = \frac{40}{136} = \frac{5}{17}$$

There is a 17-hole circle, so for each division turn the crank 5 holes on a 17-hole circle.

CAUTION: In setting the sector arms to space off the required number of holes on the index circle, do not count the hole that the index crank pin is in.

Most manufacturers provide different plates for indexing. Late model Brown and Sharpe index heads use two plates with the following circle of holes. Plate one: 15, 16, 19, 23, 31, 37, 41, 43, 47, Plate two: 17, 18, 20, 21, 27, 29, 33, 39, 47. The standard index plate supplied with the Cincinnati index head is provided with 11 different circles of holes on each side.

Side one: 24-25-28-30-34-37-38-39-41-42-43
Side two: 46-47-49-51-53-54-57-58-59-62-66

ANGULAR INDEXING

When you must divide work into degrees or fractions of degrees by plain indexing, remember that one turn of the index crank will rotate a point on the circumference of the work 1/40 of a revolution. Since there are 360° in a circle, one turn of the index crank will revolve the circumference of the work 1/40 of 360°, or 9°. Hence, in using the index plate and fractional parts of a turn, 2 holes in an 18-hole circle equal 1°, 1 hole in a 27-hole circle equals 2/3°, 3 holes in 54-hole circle equal 1/3°. To determine the number of turns and parts of a turn of the index crank for a desired number of degrees, divide the number of degrees by 9. The quotient will represent the number of complete turns and fractions of a turn that you should rotate the index crank. For example, the calculation for determining 15° when an index plate with a 54-hole circle is available, is as follows:

$$\frac{15}{9} = 1\frac{6}{9} \times \frac{6}{6} = 1\frac{36}{54}$$

or one complete turn plus 36 holes on the 54-hole circle. The calculation for determining 13 1/2° when an index plate with an 18-hole circle is available, is as follows:

$$\frac{13.5}{9} = 1\frac{4.5}{9} \times \frac{2}{2} = 1\frac{9}{18}$$

or one complete turn plus 9 holes on the 18-hole circle.

When indexing angles are given in minutes and approximate divisions are acceptable, movement of the index crank and the proper index plate may be determined by the following calculations. You can determine the number of minutes represented by one turn of the index crank by multiplying the number of degrees covered in one turn of the index crank by 60:

$$9° \times 60' = 540'$$

Therefore, one turn of the index crank will rotate the index head spindle 540 minutes.

The number of minutes (540) divided by the number of minutes in the division desired, indicates the total number of holes there should be in the index plate used. (Moving the index crank one hole will rotate the index head spindle through the desired number of minutes of angle.) This method of indexing can be used only for approximate angles since ordinarily the quotient will come out in mixed numbers or in numbers for which there are no index plates available. However, when the quotient is nearly equal to the number of holes in an available index plate, the nearest number of holes can be used and the error will be very small. For example, the calculation for 24 minutes would be:

$$\frac{540}{24} = \frac{22.5}{1}$$

or one hole on the 22.5-hole circle. Since there is no 22.5-hole circle on the index plate, a 23-hole circle plate would be used.

If a quotient is not approximately equal to an available circle of holes, multiply by any trial number which will give a product equal to the number of holes in one of the available index circles. You can then move the crank the required number of holes to give the desired division. For example, the calculation for determining 54 minutes when an index plate that has a 20-hole circle is available, is as follows:

$$\frac{54}{540} = \frac{1}{10} \times \frac{2}{2} = \frac{2 \text{ (no. of holes)}}{20 \text{ (20-hole circle)}}$$

or 2 holes on the 20-hole circle.

COMPOUND INDEXING

Compound indexing is a combination of two plain indexing procedures to index a number of divisions beyond the range of plain indexing. One number of divisions is indexed using the standard plain indexing method; another number of divisions is indexed by turning the index plate (leaving the crank pin engaged in the hole as set in the first indexing operation) by a required amount. The difference between the amount indexed in the first operation and the amount indexed in the second operation results in the spindle turning the required amount for the number of divisions. Compound indexing is seldom used because (1) differential indexing is easier, (2) high number index plates are usually available to provide any range of divisions normally required and (3) the computation and actual operation are quite complicated making it easy for errors to be introduced.

Compound indexing is briefly described in the following example. To index 99 divisions proceed as follows:

1. Multiply the required number of divisions by the difference between the number of holes in two circles selected at random. Divide this product by 40 (ratio of spindle to crank) times the product of the two index hole circles. Assume that the 27-hole circle and

22-hole circle have been selected. The resulting equation is:

$$\frac{99 \times (33 - 27)}{40 \times 33 \times 27} \times \frac{99 \times 6}{40 \times 33 \times 27}$$

2. To make the problem easier to solve, factor each term of the equation into its lowest prime factors and cancel where possible. For example:

$$\frac{(\cancel{3} \times \cancel{3} \times \cancel{11})(\cancel{3} \times \cancel{2})}{(\cancel{2} \times 2 \times 2 \times 5)(\cancel{11} \times \cancel{3})(\cancel{3} \times \cancel{3} \times 3)} = \frac{1}{60}$$

The result of this process must be in the form of a fraction as given (that is, 1 divided by some number). Always try to select the two circles which will have factors that cancel out the factors in the numerator of the problem. When the numerator of the resulting fraction is greater than 1, divide it by the denominator and use the quotient (to nearest whole number) instead of the denominator of the fraction.

3. The denominator of the resulting fraction derived in step two is the term used to find the number of turns and holes for indexing the spindle and index plate. To index for 99 divisions, turn the spindle by an amount equal to 60/33 or one complete turn plus 27 holes in the 33-hole circle; turn the index plate by an amount equal to 60/27 or two complete turns plus 6 holes in the 27-hole circle. If the index crank is turned clockwise, the index plate must be turned counterclockwise and vice versa.

DIFFERENTIAL INDEXING

Differential indexing is similar to compound indexing except that the index plate is turned during the indexing operation by gears connected to the dividing head spindle. Because the index plate movement is caused by the spindle movement, only one indexing procedure is required. The gear train between the dividing head spindle and the index plate provides the correct ratio of movement between the spindle and index plate.

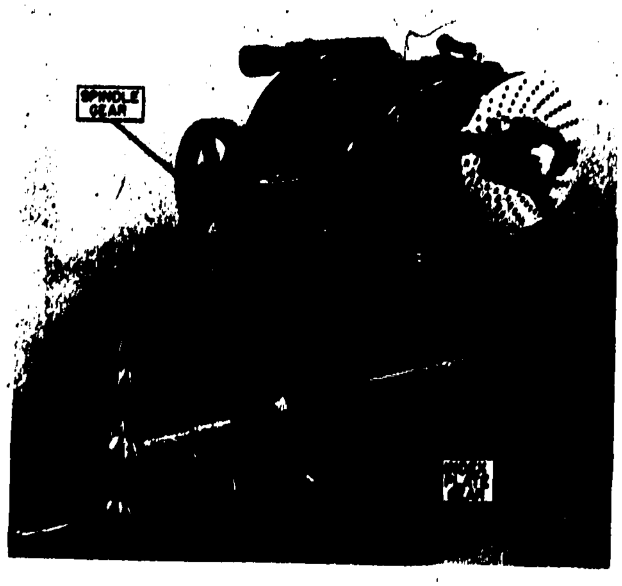
Figure 11-17 shows a dividing head set up for differential indexing. The index crank is turned as for plain indexing, thus turning the spindle gear and then the compound gear and idler to drive the gear which turns the index plate. Specific procedures for installing the gearing and arranging the index plate for differential indexing (and compound indexing) are given in manufacturers' technical manuals.

To index 57 divisions for example, take the following steps:

1. Select a number greater or lesser than the required number of divisions for which an available index plate can be used (60 for example).

2. The number of turns for plain indexing 60 divisions is: 40/60 or 14/21 which will require 14 holes in a 21-hole circle in the index plate.

3. To find the required gear ratio, subtract the required number of divisions from the selected number or vice versa (depending on



28.210X

Figure 11-17.—Differential indexing.

which is larger), and multiply the result by 40/60 (formula for indexing 60 divisions). Thus:

$$\text{gear ratio} = (60 - 57) \times \frac{40}{60} = 3 \times \frac{40}{60} = \frac{2}{1}$$

The numerator indicates the spindle gear; the denominator indicates the driven gear.

4. Select two gears that have a 2 to 1 ratio (for example a 48-tooth gear and a 24-tooth gear).

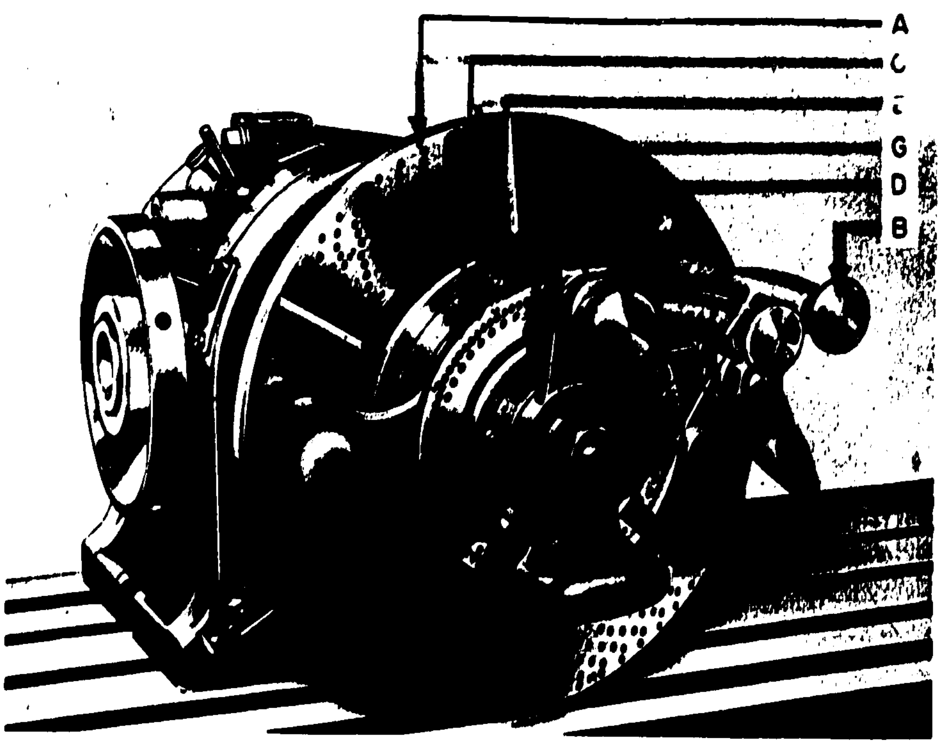
5. If the selected number is greater than the actual number of divisions required, use one or three idlers in the simple gear train; if the selected number is smaller, use none or two idlers. The reverse is true for compound gear trains. Since the number is greater in this example, use one or three idlers.

6. Now turn the index crank 14 holes in the 21-hole circle of the index plate. As the crank turns the spindle, the gear train turns the index plate slightly faster than the index crank.

Wide Range Divider

In the majority of indexing operations, the desired number of equally spaced divisions may be obtained by either direct or plain indexing. By using one or the other of these methods, you may index up to 2,640 divisions. To increase the range of divisions, use the high number index plates in place of the standard index plate. These high number plates have a greater number of circles of holes and a greater range of holes in the circles than the standard plates. This increases the range of divisions obtainable from 1,040 to 7,960.

In some instances, you may need to index beyond the range of any of these methods. To further increase the range, use a universal dividing head that has a wide range divider. This type of indexing equipment enables you to index divisions from 2 to 400,000. The wide range divider (fig. 11-18) consists of a large index plate with sector arms and crank and a



128.28X

Figure 11-18.—The wide range divider.

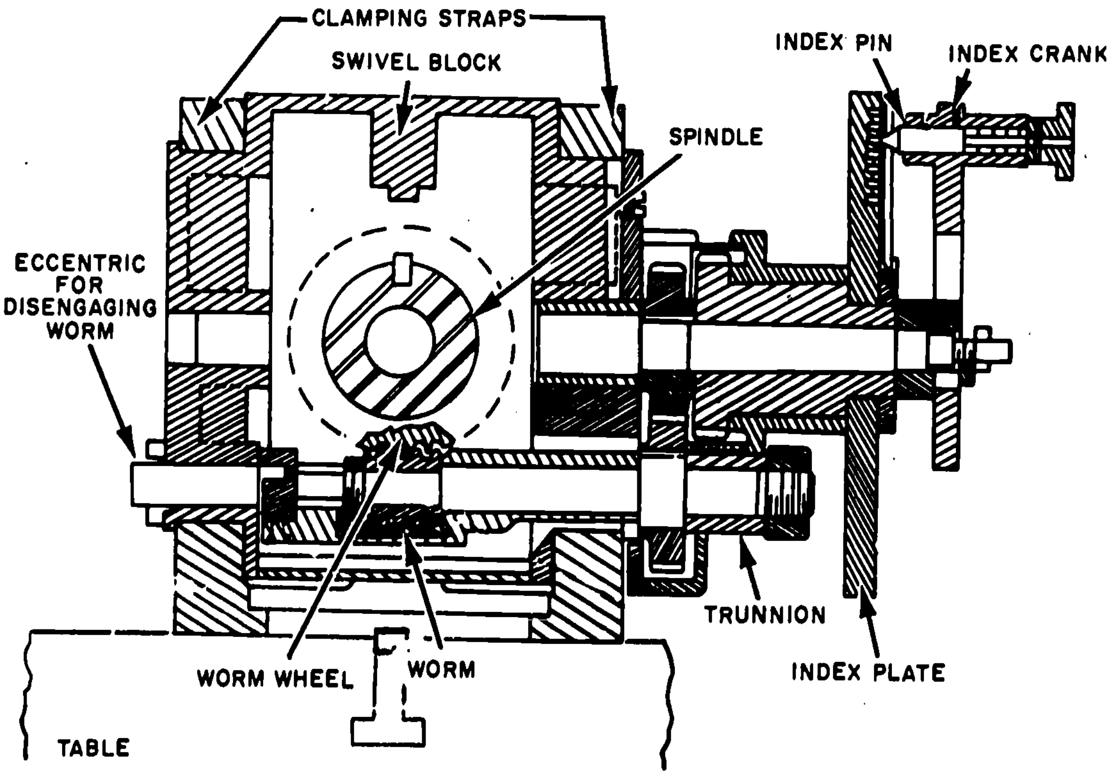
306

small index plate with sector arms and crank. The large index plate (A, fig. 11-18) has holes drilled on both sides and contains eleven circles of holes on each side of the plate. The number of holes in the circles on one side are 24, 28, 30, 34, 37, 38, 39, 41, 42, 43, and 100. The other side of the plate has circles containing 46, 47, 49, 51, 53, 54, 57, 58, 59, 62, and 66 holes. The small index plate has two circles of holes and is drilled on one side only. The outer circle has 100 holes and the inner circle has 54 holes.

The small index plate (C, fig. 11-18) is mounted on the housing of the planetary gearing (G, fig. 11-18) which is built into the index crank (B, fig. 11-18) of the large plate. As the index crank of the large plate is rotated, the planetary gearing assembly and the small index plate and crank rotate with it.

As with the standard dividing head, the large index crank rotates the spindle in the ratio of 40 to 1. Therefore, one complete turn of the large index crank rotates the dividing head spindle 1/40 of a turn, or 9°. By using the large index plate and crank, you can index in the conventional manner. Machine operation is the same as it is with the standard dividing head.

When the small index crank (D, fig. 11-18) is rotated, the large index crank remains stationary but the main shaft which drives the work revolves in the ratio of 100 to 1. This ratio, superimposed on the 40 to 1 ratio between the worm and worm wheel (see fig. 11-19), causes the dividing head spindle to rotate in the ratio of 4,000 to 1. This means, then, that one complete revolution of the spindle will require 4,000 turns of the small index crank. Turning the small crank one complete turn will rotate the dividing head spindle 5 minutes, 24 seconds of a degree. If one hole of the 100-hole circle on the small index plate were to be indexed, the dividing head spindle would make 1/400,000 of a turn, or 3.24 seconds of a degree.



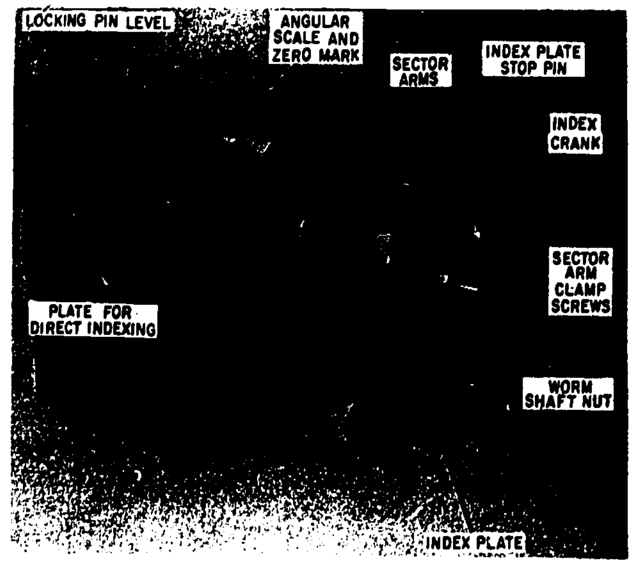
28.370
Figure 11-19.—Section through a dividing head showing worm, worm wheel, and worm shaft. (Cincinnati Milacron Co.)

307

You can get any whole number of divisions, up to and including 60, and hundreds of others, by using only the large index plate and crank. The dividing head manufacturer provides tables listing many of the settings which may be read directly from the table with no further calculations necessary. If the number of divisions required is not listed in the table or if there are no tables, calculate the required settings in the following manner: Divide 400,000 by the number of divisions desired. The result gives you a whole number quotient and a fraction. Set the sector arm of the large index plate to the first two hole numbers of the quotient, and set the sector arm of the small index plate to the last two numbers of the quotient. If the quotient results in a five-digit number (instead of the usual four digits), the first number of the quotient determines the number of full turns of the large index crank. After you index both the large and the small cranks, you must compensate for the remaining fraction. Add one space to the indexing movement of the small crank at intervals equal to the first whole number you get by dividing 1 by the remaining fraction. The indexing of the fraction will be slightly in error o.er a small span of divisions; however, the error is so slight as to be negligible.

Adjusting the Sector Arms

To use the index head sector arms, turn the left-hand arm to the left of the index pin which is inserted into the first hole in the circle of holes that is to be used. Then loosen the setscrew (fig. 11-18E) and adjust the right-hand arm of the sector so that the correct number of holes will be contained between the two arms (fig. 11-20). After making the adjustments, lock the setscrew to hold the arms in position. When setting the arms, count the required number of holes from the one in which the pin is inserted, considering this hole as zero. By subsequent use of the index sector, you will not need to count the holes for each division. When using the index crank to revolve the spindle, you must unlock the spindle clamp screw; however, before cutting work held in or on the index head, lock the spindle again to relieve the strain.



28.371

Figure 11-20.—Principal parts of a late model Cincinnati universal spiral index head (Cincinnati Milacron Co.).

CUTTERS AND ARBORS

When performing a milling operation, you move the work into a rotating cutter. On most milling machines, the cutter is mounted on an arbor that is driven by the spindle. However, the spindle may drive the cutter directly. We will discuss cutters in the first part of this section and arbors in the second part.

CUTTERS

There are many different milling machine cutters. Some cutters can be used for several operations, while others can be used for only one operation. Some cutters have straight teeth and others have helical teeth. Some cutters have mounting shanks and others have mounting holes. You must decide which cutter to use. To make this decision, you must be familiar with the various milling cutters and their uses. The information in this section will help you to select the proper cutter for the various operations you will be performing. In this section we will cover cutter types and cutter selection.

Standard milling cutters are made in many shapes and sizes for milling both regular and irregular shapes. Various cutters designed for specific purposes also are available; for example, a cutter for milling a particular kind of curve on some intermediate part of the workpiece.

Milling cutters generally take their names from the operation which they perform. Those commonly recognized are: (1) plain milling cutters of various widths and diameters, used principally for milling flat surfaces which are parallel to the axis of the cutter; (2) angular milling cutters, designed for milling V-grooves and the grooves in reamers, taps, and milling cutters; (3) face milling cutters, used for milling flat surfaces at right angles to the axis of the cutter; and (4) forming cutters, used for the production of surfaces with some form of irregular outline.

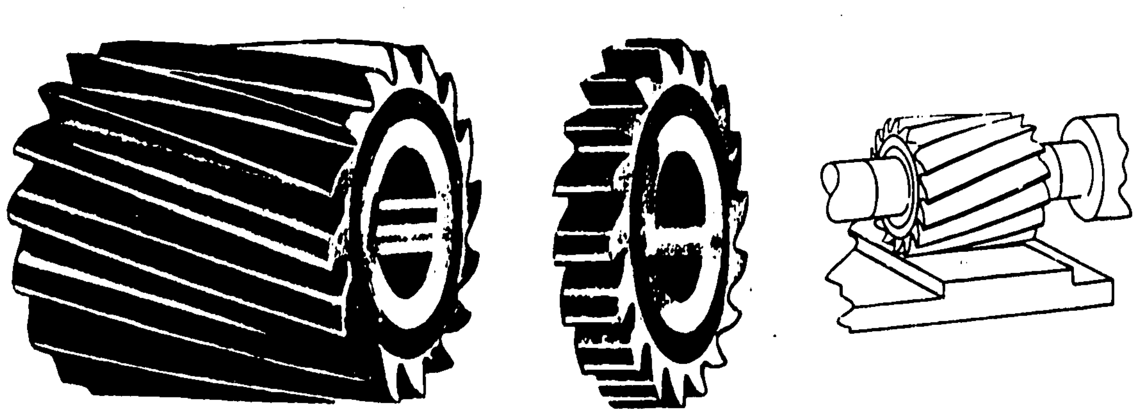
Milling cutters may also be classified as arbor-mounted, or shank-mounted. ARBOR-MOUNTED CUTTERS are mounted on the straight shanks of arbors. The arbor is then inserted into the milling machine spindle. We will discuss the methods of mounting arbors and cutters in greater detail later in this chapter.

Milling cutters may have straight, right-hand, left-hand, or staggered teeth. Straight teeth are parallel to the axis of the cutter. If the helix angle twists in a clockwise direction (viewed from either end), the cutter has right-hand teeth. If the helix angle twists in a counterclockwise direction, the cutter has left-hand teeth. The teeth on staggered-tooth cutters are alternately left-hand and right-hand.

Types and Uses

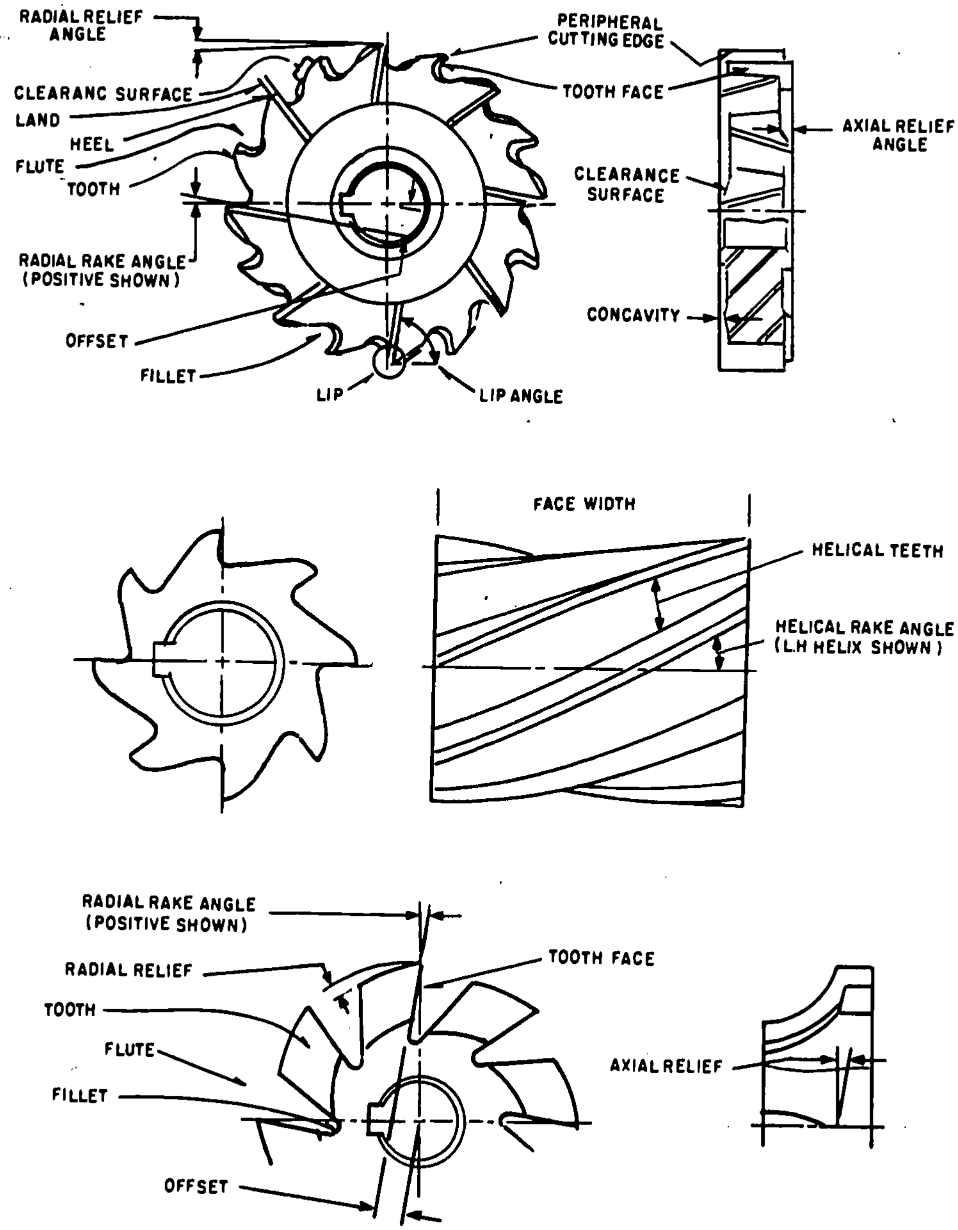
There are many different types of milling cutters. We will now discuss these types and their uses.

PLAIN MILLING CUTTER.–You will use plain milling cutters to mill flat surfaces that are parallel to the cutter axis. As you can see in figure 11-21, a plain milling cutter is a cylinder with teeth cut on the circumference only. Plain milling cutters are made in a variety of diameters and widths. Note in figure 11-22, that the cutter teeth may be either straight or helical. When the width is more than 3/4 inch, the teeth are usually helical. A straight cutter tool is parallel to its axis. It cuts along its entire width at the same time, causing a shock as the tooth starts to cut. Helical teeth eliminate this shock and produce a free cutting action. A helical tooth begins the cut at one end and continues across the work with a smooth shaving action. Plain milling cutters usually have radial teeth. On some coarse helical tooth cutters the tooth face is undercut to produce a smoother cutting



28.372

Figure 11-21.–Plain milling cutters.



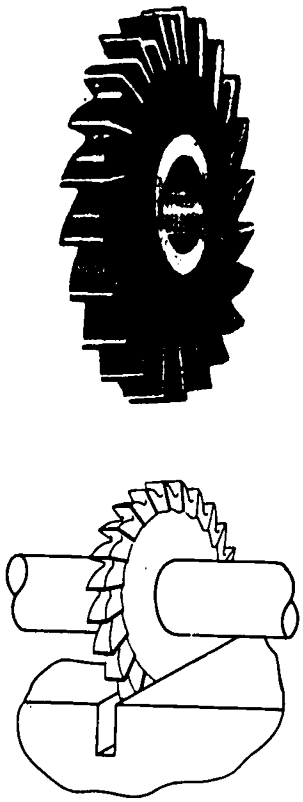
28.373

Figure 11-22.—Milling cutter terms.

action. Coarse teeth decrease the tendency of the arbor to spring and give the cutter greater strength.

A plain milling cutter has a standard size arbor hole for mounting on a standard size arbor. The size of the cutter is designated by the diameter of the cutter, width of the cutter, and diameter of the hole in the cutter.

SIDE MILLING CUTTER.—The side milling cutter (fig. 11-23) is a plain milling cutter with teeth cut on both sides as well as on the



28.374
Figure 11-23.–Side milling cutter.

periphery or circumference of the cutter. You can see that the portion of the cutter between the hub and the side of the teeth is thinner to give more chip clearance. These cutters are often used in pairs to mill parallel sides. This process is called straddle milling. Cutters more than 8 inches in diameter are usually made with inserted teeth. The size designation is the same as for plain milling cutters.

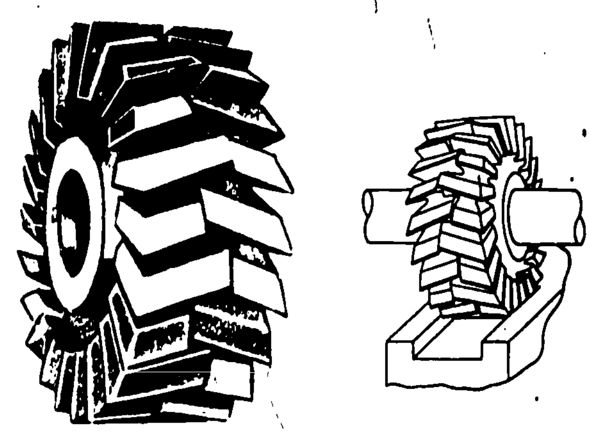
HALF-SIDE MILLING CUTTER.–Half-side milling cutters (fig. 11-24) are made particularly for jobs where only one side of the cutter is needed. These cutters have coarse, helical teeth on one side only so that heavy cuts can be made with ease.

SIDE MILLING CUTTER (INTERLOCKING).–Side milling cutters whose teeth interlock (fig. 11-25) can be used to mill

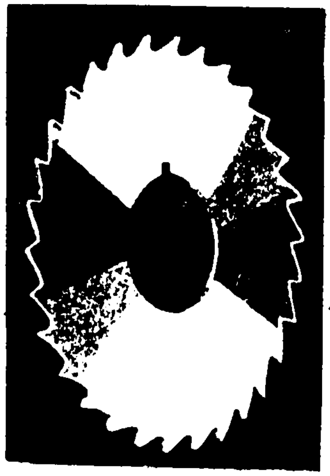
28.375
Figure 11-24.–Half-side milling cutter.

standard size slots. The width is regulated by thin washers inserted between the cutters.

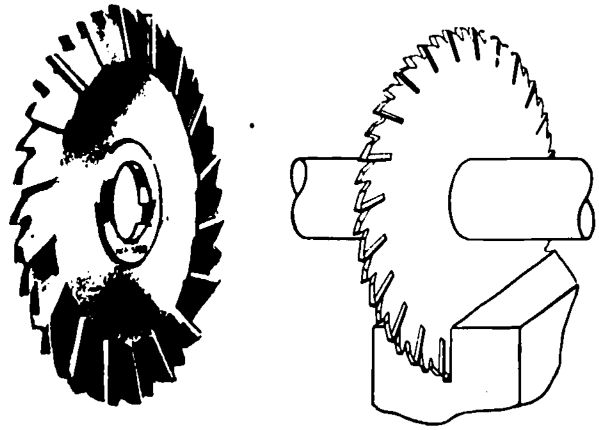
METAL SLITTING SAW.–You can use a metal slitting saw to cut off work or to mill narrow slots. A metal slitting saw is similar to a plain or side milling cutter. The face width is usually less than 3/16 inch. This type of cutter usually has more teeth for a given diameter than



28.376
Figure 11-25.–Interlocking teeth side milling cutter.



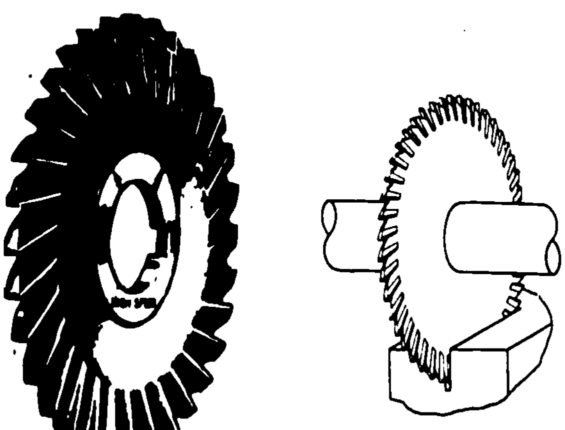
28.377
Figure 11-26.—Metal slitting saw.



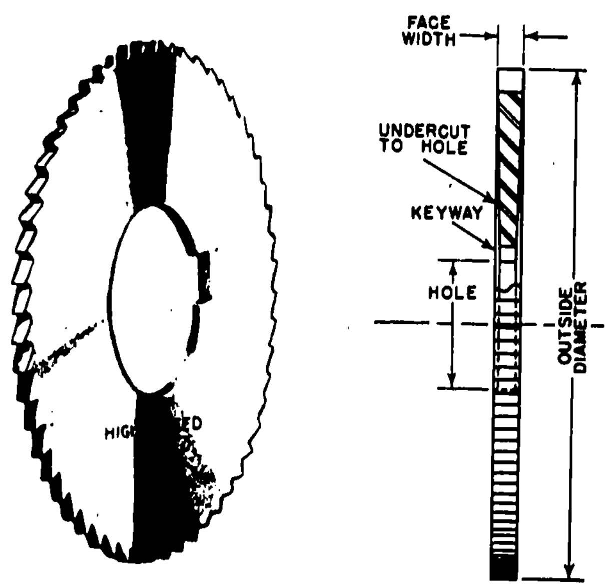
28.379
Figure 11-28.—Slitting saw with staggered teeth.

plain cutter. It is thinner at the center than at he outer edge to give proper clearance for nilling deep slots. Figure 11-26 shows a metal litting saw with teeth cut in the circumference f the cutter only. Some saws, such as the one in igure 11-27, have side teeth which achieve etter cutting action, break up chips, and revent dragging when you cut deep slots. For eavy sawing in steel, there are metal slitting saws with staggered teeth, as shown in figure 11-28. These cutters are usually 3/16 inch to 3/8 inch in thickness.

SCREW SLOTTING CUTTER.—The screw slotting cutter (fig. 11-29) is used to cut shallow slots, such as those in screw heads. This cutter



28.378
Figure 11-27.—Slitting saw with side teeth.



28.380
Figure 11-29.—Screw slotting cutter.

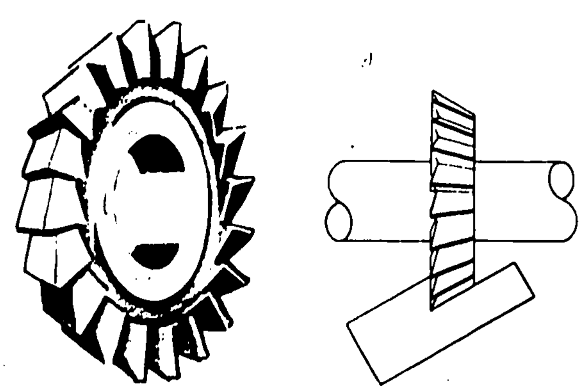
has fine teeth cut on its circumference. It is made in various thicknesses to correspond to American Standard gage wire numbers.

ANGLE CUTTER.–Angle cutters are used to mill surfaces that are not at a right angle to cutter axis. You can use angle cutters for a variety of work, such as milling V-grooves and dovetail ways. On work such as dovetailing, where you cannot mount a cutter in the usual manner on an arbor, you can mount an angle cutter that has a threaded hole or is constructed like a shell end mill on the end of a stub or shell end mill arbor. When you select an angle cutter for a job you should specify type, hand, outside diameter, thickness, hold size, and angle.

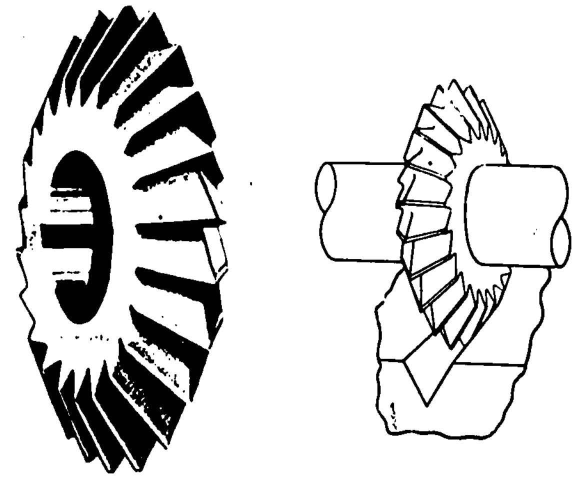
There are two types of angle cutters: single and double. The single angle cutter, shown in figure 11-30, has teeth cut at an oblique angle with one side at an angle of 90° to the cutter axis and the other usually at 45°, 50°, or 80°.

The double angle cutter (fig. 11-31) has two cutting faces, which are at an angle to the cutter axis. When both faces are at the same angle to the axis, you obtain the cutter you want by specifying the included angle. When they are different angles, you specify the angle of each side with respect to the plane of intersection.

FLUTING CUTTER.–A fluting cutter is a double angle form tooth cutter with the points



28.381
Figure 11-30.–Single angle cutter.



28.382
Figure 11-31.–Double angle cutter.

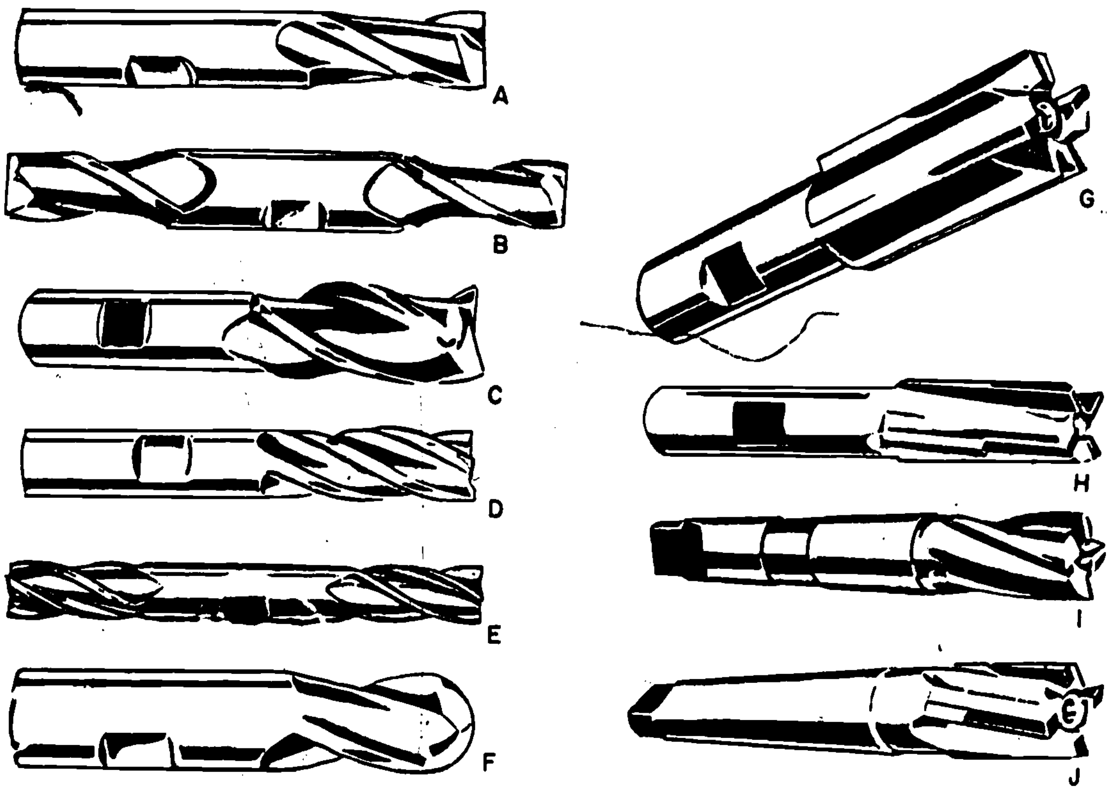
of the teeth well rounded. It is generally used to mill flutes in reamers. Fluting cutters are marked with the range of diameters they are designed to mill.

END MILL CUTTERS.–End milling cutters may be the SOLID TYPE with the teeth and the shank as an integral part (fig. 11-32). They also may be the SHELL TYPE (fig. 11-33) in which the cutter body and the shank or arbor are separate. End milling cutters have teeth on the circumference and on the end. Those on the circumference may be straight or helical.

Except for the shell type, all end mills have either a straight shank or a tapered shank which is mounted into the spindle of the machine for driving the cutter. There are various types of adapters for securing end mills to the machine spindle.

End milling involves the machining of surfaces (horizontal, vertical, angular, or irregular) with end milling cutters. Common operations include the milling of slots, keyways, pockets, shoulders, and flat surfaces, and the profiling of narrow surfaces. (See fig. 11-34.)

End milling cutters are used most often on vertical milling machines. However, they also are

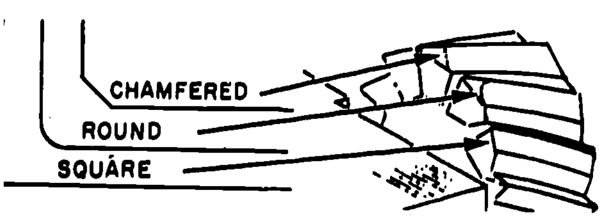


(A) Two-flute single-end; (B) Two-flute double-end; (C) Three-flute single-end; (D) Multiple-flute single-end; (E) Four-flute double-end; (F) Two-flute ball-end; (G) Carbide-tipped, straight flutes; (H) Carbide-tipped, RH helical flutes; (I) Multiple-flute with taper shank; (J) Carbide-tipped with taper shank and helical flutes.

28.383

Figure 11-32.—End mill cutters.

used frequently on machines with horizontal spindles. Many different types of end milling cutters are available in sizes ranging from 1/64 inch to 2 inches. They may be made of high-speed steel, or they may have cemented carbide teeth or they may be of the solid carbide type.



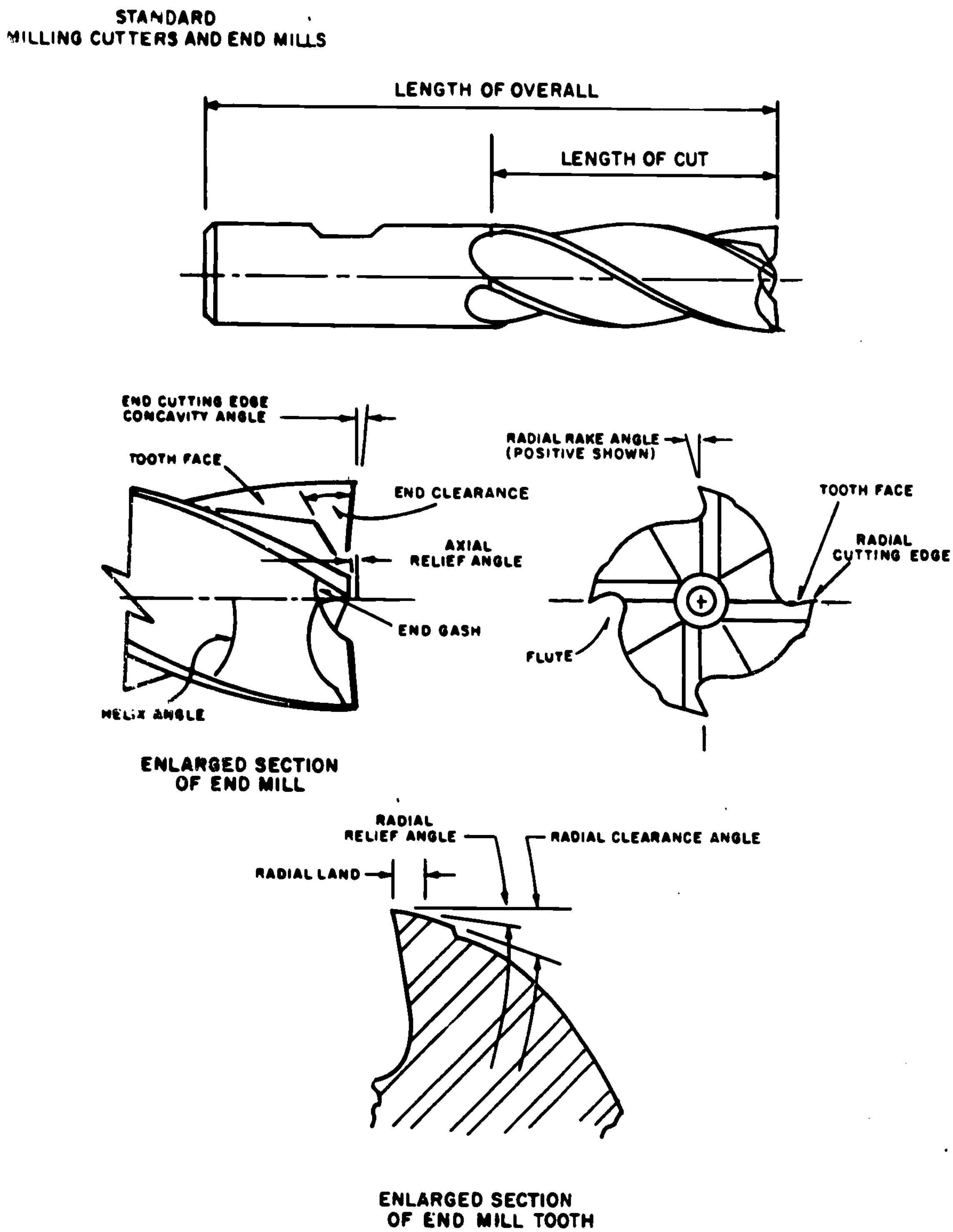
28.384

Figure 11-33.—Shell End Mill (National Twist Drill).

TWO-FLUTE END MILLS have only two teeth on the circumference. The end teeth can cut to the cutter. Hence, they may be used to feed into the work like a drill; they can then be fed lengthwise to form a slot. These mills may be either the single end type with cutter on one end only, or they may be the double-end type. (See fig. 11-32.)

MULTIPLE-FLUTE END MILLS have three, four, six, or eight flutes and normally are available in diameters up to 2 inches. They may be single- or double-end (fig. 11-32).

BALL END MILLS (fig. 11-32) are used for milling fillets or slots with a radius bottom, for



28.385

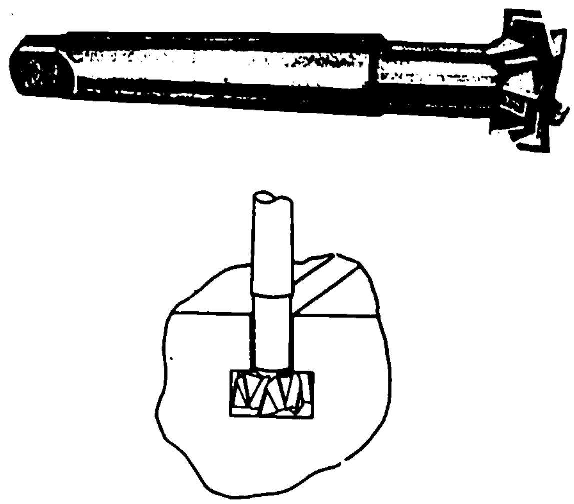
Figure 11-34.—End mill terms.

rounding pockets and the bottom of holes, and for all-round die sinking and die making work. Two-flute end mills with end cutting lips can be used to drill the initial hole as well as to feed longitudinally. Four-fluting ball end mills with center cutting lips also are available. These work well for tracer milling, fillet milling and die sinking.

SHELL END MILLS (fig. 11-33) have a hole for mounting the cutter on a short (stub) arbor. The center of the shell is recessed for the screw or nut which fastens the cutter to the arbor. The teeth are usually helical. These mills are made in larger sizes than solid end mills. Normally, they are available in diameters from 1 1/4 to 6 inches. Cutters of this type are intended for slabbing or surfacing cuts, either face milling or end milling.

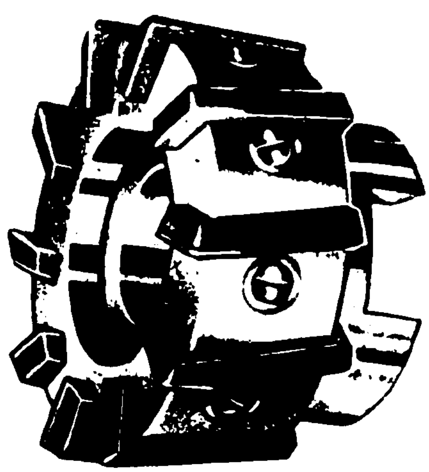
FACE MILLING CUTTER.–Inserted tooth face milling cutters (fig. 11-35) are similar to shell end mills in that they have teeth on the circumference and on the end. They are attached directly to the spindle nose and use inserted teeth made of carbide or an alloy steel. The teeth are replaceable.

T-SLOT CUTTER.–The T-slot cutter (fig. 11-36) is a small plain milling cutter with a shank. It is designed especially to mill the "head space" of T-slots. T-slots are cut in two operations. First, you cut a slot with an end mill or a plain milling cutter, and then you make the cut at the bottom of the slot with a T-slot cutter.

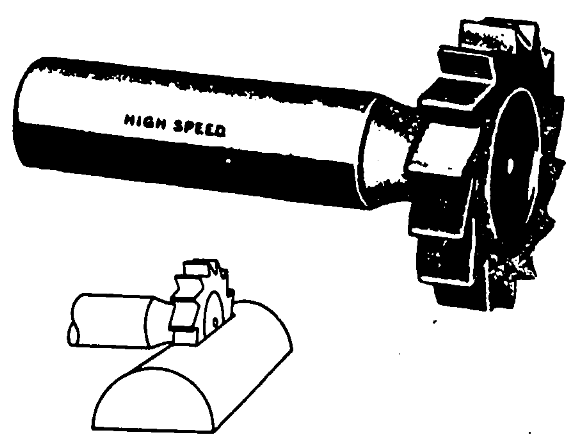


28.387
Figure 11-36.–T-slot cutter.

WOODRUFF KEYSEAT CUTTER.–A Woodruff keyseat cutter (fig. 11-37) is used to



28.386
Figure 11-35.–Inserted tooth face milling cutter.



28.388
Figure 11-37.–Woodruff keyseat cutter.

cut curved keyseats. A cutter less than 1 1/2 inches in diameter has a shank. When the diameter is greater than 1 1/2 inches, the cutter is usually mounted on an arbor. The larger cutters have staggered teeth to improve the cutting action.

GEAR CUTTERS.–There are several types of gear cutters, such as bevel, spur, involute, etc. Figure 11-38 shows an involute gear cutter. You must select the correct type of cutter to cut a particular type of gear.

CONCAVE AND CONVEX CUTTERS.–A concave cutter (fig. 11-39) is used to mill a convex surface, and a convex cutter (fig. 11-40) is used to mill a concave surface.

28.390

Figure 11-39.–Concave cutter.

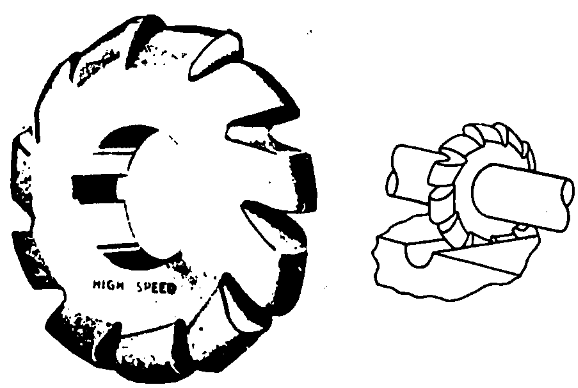
28.389

Figure 11-38.–Involute gear cutter.

CORNER ROUNDING CUTTER.–Corner rounding cutters (fig. 11-41) are formed cutters that are used to round corners up to one-quarter of a circle.

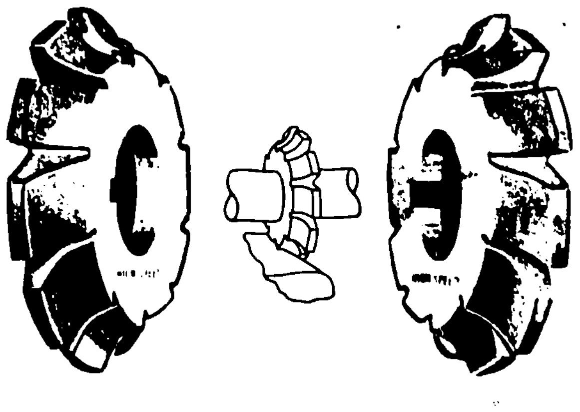
SPROCKET WHEEL CUTTER.–The sprocket wheel cutter (fig. 11-42) is a formed cutter that is used to mill teeth on sprocket wheels.

GEAR HOB.–The gear hob (fig. 11-43) is a formed milling cutter with teeth cut like threads on a screw.

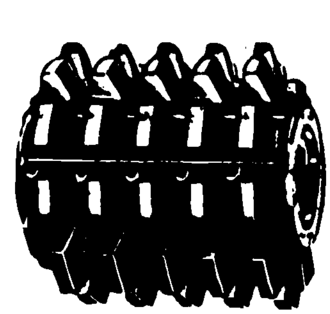


28.391

Figure 11-40.–Convex cutter.



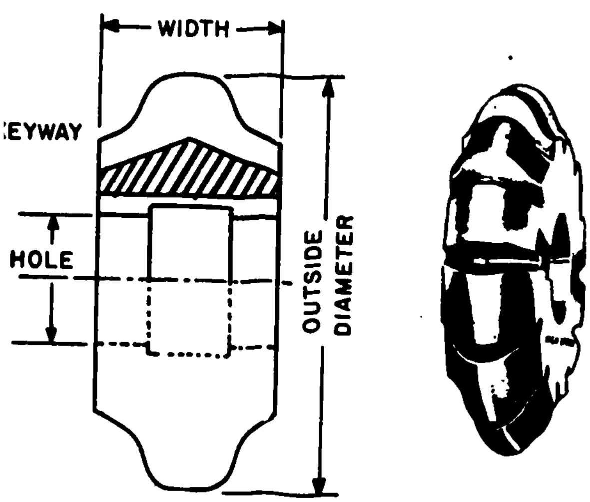
28.392
Figure 11-41.—Corner rounding cutter.



28.394
Figure 11-43.—Gear hob.

FLY CUTTER.—The fly cutter (fig. 11-44) is often manufactured locally. It is a single-point cutting tool similar in shape to a lathe or shaper tool. It is held and rotated by a fly cutter arbor. You can grind this cutter to almost any form you desire. There will be times when you need a special formed cutter for a very limited number of cutting or boring operations.

We have discussed a number of the more common types of milling machine cutters. For a more detailed discussion of these and other types of cutters and their uses, consult the *Machinery's Handbook*, machinist publications, or the applicable technical manual. We will now discuss the selection of cutters.



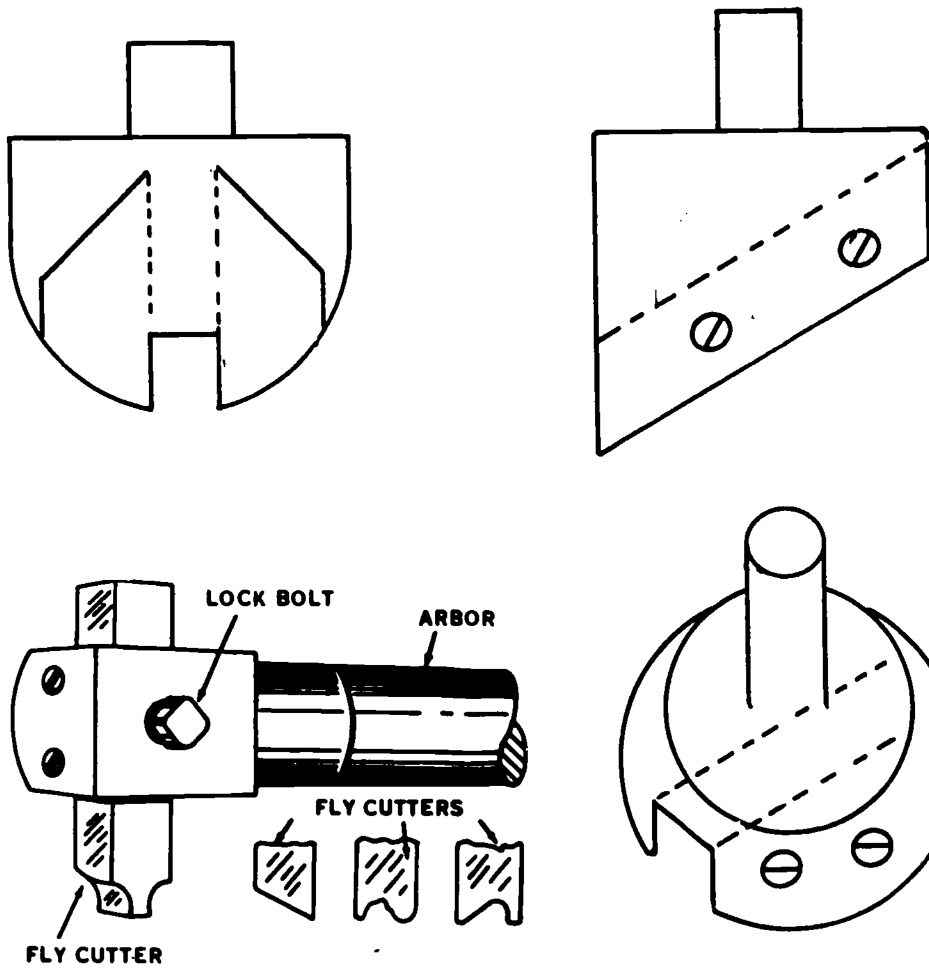
28.393
Figure 11-42.—Sprocket wheel cutter.

Selection

Each cutter can do one kind of job better than any other cutter in a given situation. A cutter may or may not be limited to a specific milling operation. In selecting the most suitable cutter for a particular milling operation, you must consider the kind of cut to be made, material to be cut, number of parts to be machined, and the type of milling machine available.

Another factor that affects a milling operation is the number of teeth in the cutter. If the number is too great, the space between the teeth is so small that it prevents the free flow of chips. The chip space should also be smooth and free of sharp corners to prevent the chips from clogging the space. A coarse-tooth cutter is more satisfactory for milling material that produces a continuous and curled chip. The coarse teeth not only permit an easier flow of chips and coolant but also help to eliminate chatter. A fine-tooth cutter is more satisfactory for milling a thin material. It reduces cutter and workpiece vibration and the tendency for the cutter teeth to "straddle" the work and dig in.

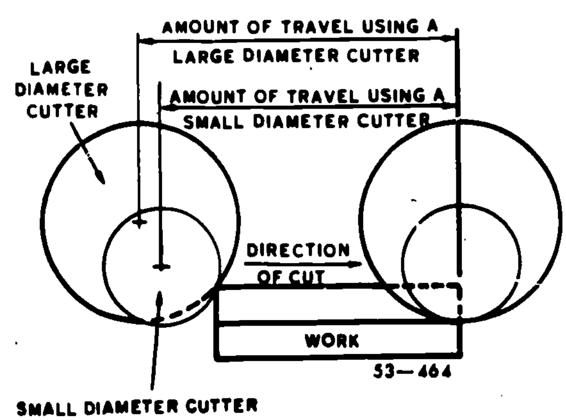
Another factor you should consider in selecting a cutter is the diameter of the cutter.



28.395

Figure 11-44.—Fly cutter arbor and fly cutters.

Select the smallest diameter of the cutter that will allow the arbor to pass over the work without interference when the cut is taken. As illustrated in figure 11-45, a small cutter requires less time than a larger cutter to take a cut.



28.396

Figure 11-45.—Cutter diameter selection.

ARBORS

You can mount milling machine cutters on several types of holding devices. You must know the devices and the purpose of each of them to make the most suitable tooling setup for the operation you are performing. We will cover the various types of arbors and the mounting and dismounting of arbors in this section.

NOTE: Technically, an arbor is a shaft on which a cutter is mounted. For convenience, since there are so few types of cutter holders that are not arbors, we will refer to all types of cutter holding devices as arbors.

STANDARD ARBOR.–There are several types of milling machine arbors. You use the common or standard types (fig. 11-46) to hold and drive cutters that have mounting holes. One end of the arbor usually has a standard milling machine spindle taper of 3 1/2 inches per foot. The largest diameter of the taper is identified by a number. For example, the large diameter of a number 40 milling machine spindle taper is 1 3/4 inches. The numbers representing common milling machine spindle tapers and their sizes are as follows:

Number	Large Diameter
10	5/8 inch
20	7/8 inch
30	1 1/4 inches
40	1 3/4 inches
50	2 3/4 inches
60	4 1/4 inches

Standard arbors are available in styles A and B, as shown in figure 11-46. Style A arbors have a pilot type bearing usually 11/32 inch in diameter. Style B arbors have a sleeve type outboard bearing. Numerals identify the outside diameter of the bearing sleeves, as follows:

Sleeve Number	Outside Diameter
3	1 7/8 inches
4	2 1/8 inches
5	2 3/4 inches

The inside diameter can be any one of several standard diameters that are used for the arbor shaft.

Style A arbors sometimes have a sleeve bearing that permits the arbor to be used as either a style A or a style B arbor. A code system, consisting of numerals and a letter, identifies the size and style of the arbor. The code number is stamped into the flange or on the tapered portion of the arbor. The first number of the code identifies the diameter of the taper. The second (and if used, the third number) indicates the diameter of the arbor shaft. The letter indicates the type of bearing. The numbers following the letter indicate the usable length of the arbor shaft. Sometimes an additional number is used to indicate the size of

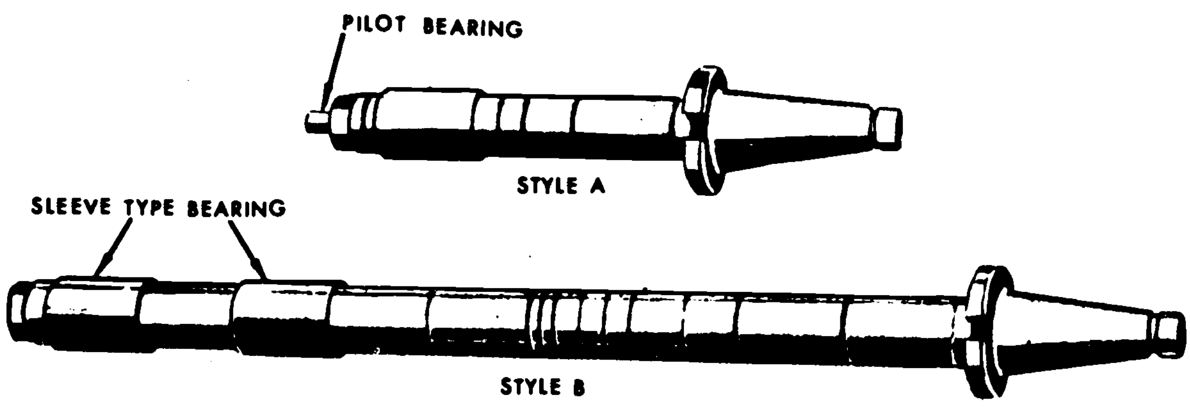


Figure 11-46.—Standard milling machine arbors.

28.397

sleeve type bearings. The meaning of a typical code number 5-1 1/4-A-18-4 is as follows:

5 = taper number–50 (the 0 is omitted in the code)

1 1/4 = shaft diameter–1 1/4 inches

A = Style A bearing–pilot type

18 = usable shaft length–18 inches

4 = bearing size–2 1/8 inches diameter

STUB ARBOR.–Arbors that have very short shafts, such as the one shown in figure 11-47, are called stub arbors. Stub arbors are used when it is impractical to use a longer arbor.

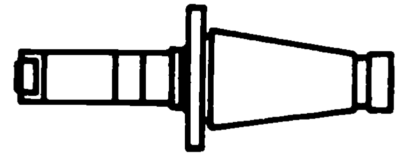


Figure 11-47.–Stub arbor.

You will use arbor spacing collars of various lengths to position and secure the cutter on the arbor. You tighten the spacers against the cutter when you tighten the nut on the arbor. Remember, never tighten or loosen the arbor nut unless the arbor support is in place.

SHELL END ARBOR.–Shell end mill arbors (fig. 11-48) are used to hold and drive shell end mills. The shell end mill is fitted over the short boss on the arbor shaft. It is driven by two keys and is held against the face of the arbor by a bolt. You use a special wrench, shown in figure 11-47, to tighten and loosen the bolt. Shell end mill arbors are identified by a code similar to the standard arbor code. The letter C indicates a shell end mill arbor. The meaning of a typical shell mill arbor code 4-1 1/2C-7/8 is as follows:

4 = taper code number–40

1 1/2 = diameter of mounting hole in end mill–1 1/2 inches

C = style C arbor–shell end mill

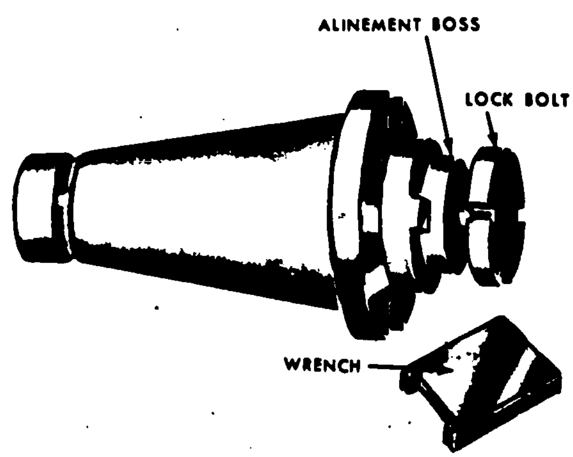
7/8 = length of shaft–7/8 inch

FLY CUTTER ARBOR.–Fly cutter arbors are used to hold single-point cutters. These cutters, which can be ground to any desired shape and held in the arbor by a locknut, are shown in figure 11-43. Fly cutter arbor shanks may have a standard milling machine spindle taper, a Brown and Sharpe taper, or a Morse taper.

SCREW SLOTTING CUTTER ARBOR.–Screw slotting cutter arbors are used with screw slotting cutters. The flanges support the cutter and prevent the cutter from flexing. The shanks on screw slotting cutter arbors may be straight or tapered, as shown in figure 11-49.

SCREW ARBOR.–Screw arbors (fig. 11-50) are used with cutters that have threaded mounting holes. The threads may be left- or right-hand.

TAPER ADAPTER.–Taper adapters are used to hold and drive taper-shanked tools, such as drills, drill chucks, reamers, and end mills, by



28.398

Figure 11-48.–Shell end mill arbor.

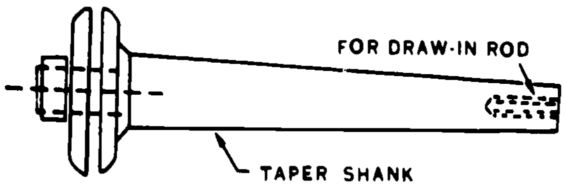


Figure 11-49.—Screw slotting cutter arbor.

inserting them into the tapered hole in the adapter. The code for a taper adapter indicates the number representing the standard milling machine spindle taper and the number and series of the internal taper. For example, the taper adapter code number 43M means:

4 = taper identification number—40

3M = internal taper—number 3 Morse

If a letter is not included in the code number, the taper is understood to be a Brown and Sharpe. For example 57 means:

5 = taper number—50

7 = internal taper—number 7 B and S

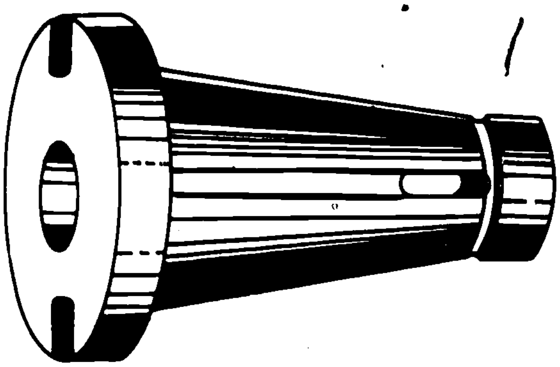
and 50-10 means:

50 = taper identification number

10 = internal taper—number 10 B and S

Figure 11-51 shows a typical taper adapter. Some cutter adapters are designed to be used with tools that have taper shanks and a cam

Figure 11-50.—Screw arbor.



28.399

Figure 11-51.—Taper adapter.

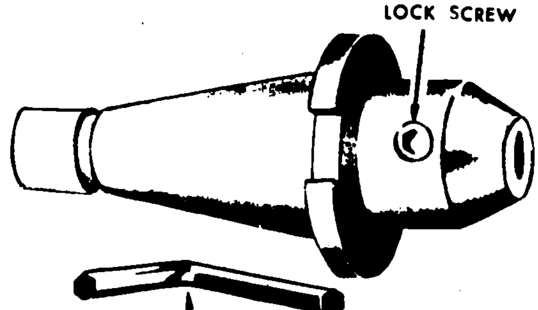
locking feature. The cam lock adapter code indicates the number of the external taper, number of the internal taper (which is usually a standard milling machine spindle taper also), and the distance that the adapter extends from the spindle of the machine. For example, 50-20-3 5/8 inches means:

50 = taper identification number (external)

20 = taper identification number (internal)

3 5/8 = distance adapter extends from spindle is 3 5/8 inches

CUTTER ADAPTER.—Cutter adapters, such as shown in figure 11-52, are similar to taper



28.400

Figure 11-52.—Cutter adapter.

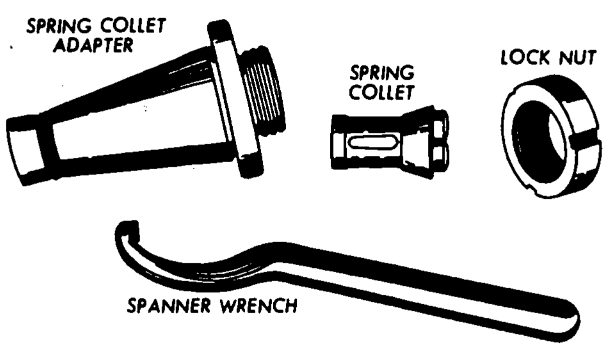
adapters except that they always have straight, rather than tapered holes. They are used to hold straight shank drills, end mills, etc. The cutting tool is secured in the adapter by a setscrew. The code number indicates the number of the taper and the diameter of the hole. For example, 50-5/8 means that the adapter has a number 50 taper and a 5/8-inch-diameter hole.

SPRING COLLET CHUCK.–Spring collet chucks (fig. 11-53) are used to hold and drive straight-shanked tools. The spring collet chuck consists of a collet adapter, spring collets, and a cup nut. Spring collets are similar to lathe collets. The cup forces the collet into the mating taper, causing the collet to close on the straight shank of the tool. The collets are available in several fractional sizes.

Mounting and Dismounting Arbors

Mounting and dismounting arbors are relatively easy tasks. Take care not to drop the arbor on the milling machine table or floor. Use figure 11-6 as a guide. To MOUNT an arbor, use the following procedure:

1. Place the spindle in the lowest speed.
2. Disengage the spindle clutch lever.
3. Turn off the motor switch.
4. Clean the spindle hole and the arbor thoroughly to ensure accurate alignment of the arbor inside the spindle.



28.401
Figure 11-53.–Spring collet chuck adapter.

5. Stand near the column at a point where you can reach both ends of the milling machine. Align the arbor keyseats with the keys in the spindle.
6. Insert the tapered shank of the arbor into the spindle.
7. Hold the arbor in place with one hand and screw the drawbolt into the arbor with your other hand. NOTE: Turn the drawbolt a sufficient number of turns to ensure that the drawbolt extends into the arbor shank a distance approximately equal to the major diameter of the threads being used. This will help to prevent stripping the threads on the drawbolt or in the arbor shank when the jamnut is tightened.
8. Hold the arbor in position by pulling back on the drawbolt and tighten the jamnut by hand.
9. Tighten the jamnut with one wrench while using a second wrench to keep the drawbolt from turning.

To DISMOUNT an arbor, use the following procedure:

1. Place the spindle in the lowest speed.
2. Turn off the motor.
3. Loosen the jamnut approximately two turns.
4. Use one wrench to turn the jamnut and another wrench to keep the drawbolt from turning.
5. Hold the arbor with one hand and gently tap the end of the drawbolt with a lead mallet until you feel the arbor break free.
6. Hold the arbor in place with one hand and unscrew the drawbolt with your other hand.
7. Remove the arbor from the spindle.

MILLING MACHINE OPERATIONS

The milling machine is one of the most versatile metalworking machines. It is capable of performing simple operations, such as milling a flat surface or drilling a hole, or more complex operations, such as milling helical gear teeth. It would be impractical to attempt to discuss all of the operations that the milling machine can do. We will limit these machining operations to plain, face, and angular milling; milling flat surfaces on cylindrical work; slotting, parting, and milling keyseats and flutes; and drilling, reaming, and boring. Even though we will discuss only the more common operations, you will find that by using a combination of operations, you will be able to produce a variety of work projects. We will conclude the chapter by discussing the milling machine attachments and gearing and gear cutting.

PLAIN MILLING

Plain milling is the process of milling a flat surface in a plane parallel to the cutter axis. You get the desired size of the work by individually milling each of the flat surfaces on the object. Plain milling cutters, such as the ones shown in figure 11-21, are used for plain milling. If possible, select a cutter that is slightly wider than the width of surface to be milled. Make the work setup before you mount the cutter. This precaution will keep you from accidentally striking the cutter and cutting your hands as you set up the work. You can mount the work in a vise or fixture, or clamp it directly to the milling machine table. You can use the same methods that you used to hold work in a shaper to hold work in a milling machine. Clamp the work as close as possible to the milling machine column so that you can mount the cutter near the column. The closer you place the cutter and work to the column, the more rigid the setup will be.

The following steps explain how to machine a rectangular work blank (for example, a spacer for an engine test stand).

1. Mount the vise on the table and position the vise jaws parallel to the table length. NOTE: The graduations on the vise are accurate enough because we are concerned only with machining a surface in a horizontal plane.

2. Place the work in the vise, as shown in figure 11-54.

3. Select the proper milling cutter and arbor.

4. Wipe off the tapered shank of the arbor and the tapered hole in the spindle with a clean cloth.

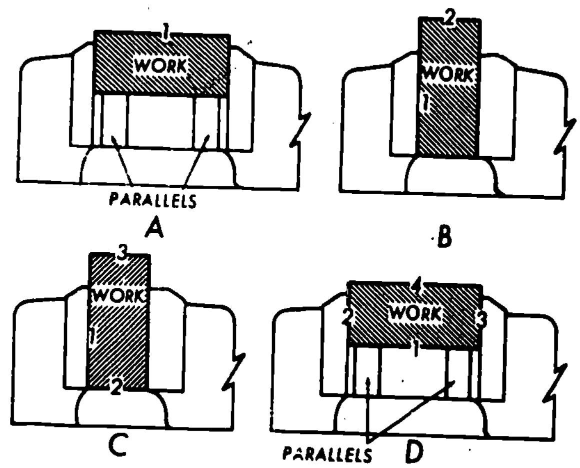


Figure 11-54.—Machining sequence to square a block.

5. Mount the arbor in the spindle.

6. Clean and position the spacing collars and place them on the arbor in such a way that the cutter is above the work.

7. Wipe off the milling cutter and any additional spacing collars that may be needed. Then place the cutter, the spacers, and the arbor bearing on the arbor, with the cutter keyseat aligned over the key. Locate the bearing as close as possible to the cutter. Make sure that the work and vise will clear all parts of the machine.

8. Install the arbor nut and tighten it fingertight only.

9. Position the overarm and mount the arbor support.

10. After supporting the arbor, tighten the arbor nut with a wrench.

11. Set the spindle directional control lever to give the required direction of cutter rotation.

12. Determine the required speed and feed, and set the spindle speed and feed controls.

13. Set the feed trip dogs for the desired length of cut and center the work under the cutter.

14. Lock the saddle.

15. Engage the spindle clutch and pick up the cut.

16. Pick up the surface of the work by holding a long strip of paper between the rotating cutter and the work; very slowly move the work toward the cutter until the paper strip is pulled between the cutter and the work. BE CAREFUL! Keep your fingers away from the cutter. A rotating milling cutter is very dangerous.

17. Move the work longitudinally away from the cutter and set the vertical feed graduated collar at ZERO.

18. Compute the depth of the roughing cut and raise the knee this distance.

19. Lock the knee, and direct the coolant flow on the work and outgoing side of the cutter.

20. Position the cutter to within 1/16 inch of the work. using hand table feed.

21. After completing the cut, stop the spindle.

22. Return the work to its starting point on the other side of the cutter.

23. Raise the table the distance required for the finish cut.

24. Set the finishing speed and feed, and take the finish cut.

25. When you have completed the operation, stop the spindle and return the work to the opposite side of the cutter.

26. Deburr the work and remove it from the vise.

To machine the second side, place the work in the vise as shown in figure 11-54B. Rough and finish machine side 2, using the same procedures that you used for side 1. When you have completed side 2, deburr the surface and remove the work from the vise.

Place the work in the vise, as shown in figure 11-54C with side 3 up. Then rough machine side 3. Finish machine side 3 for a short distance, disengage the spindle and feed, and return the work to the starting point, clear of the cutter. Now you can safely measure the distance between sides 2 and 3. If this distance is correct, you can continue the cut with the same setting. If it is not, adjust the depth of cut as necessary. If the trial finishing cut is not deep enough, raise the work slightly and take another trial cut. If the trial cut is too deep, you will have to remove the backlash from the vertical feed before taking the new depth of cut. To remove the backlash:

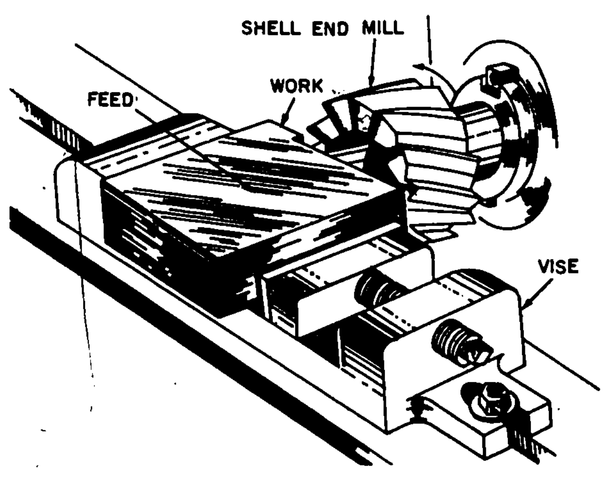
1. Lower the knee well past the original depth of the roughing cut.
2. Raise the knee the correct distance for the finishing cut.
3. Engage the feed.
4. Stop the spindle.
5. Return the wor : to the starting point on the other side of the cutter.
6. Deburr the work.
7. Remove the work from the vise.

Place side 4 in the vise, as shown in figure 11-54D and machine the side, using the same procedure as for side 3. When you have completed side 4, remove the work from the vise and check it for accuracy.

This completes the machining of the four sides of the block. If the block is not too long, you can rough and finish mill the ends to size in the same manner in which you milled the sides. This is done by placing the block on end in the vise. Another method of machining the ends is by face milling.

FACE MILLING

Face milling is the milling of surfaces that are perpendicular to the cutter axis, as shown in figure 11-55. You do face milling to produce flat



28.402

Figure 11-55.—Face milling.

surfaces and to machine work to the required length. In face milling, the feed can be either horizontal or vertical.

Cutter Setup

You can use straight shank or taper shank end mills, shell end mills, or face milling cutters for face milling. Select a cutter that is slightly larger in diameter than the thickness of the material that you are machining. If the cutter is smaller in diameter than the thickness of the material, you will be forced to make a series of slightly overlapping cuts to machine the entire surface. Mount the arbor and cutter before you make the work setup. Mount the cutter by any means suitable for the cutter you have selected.

Work Setup

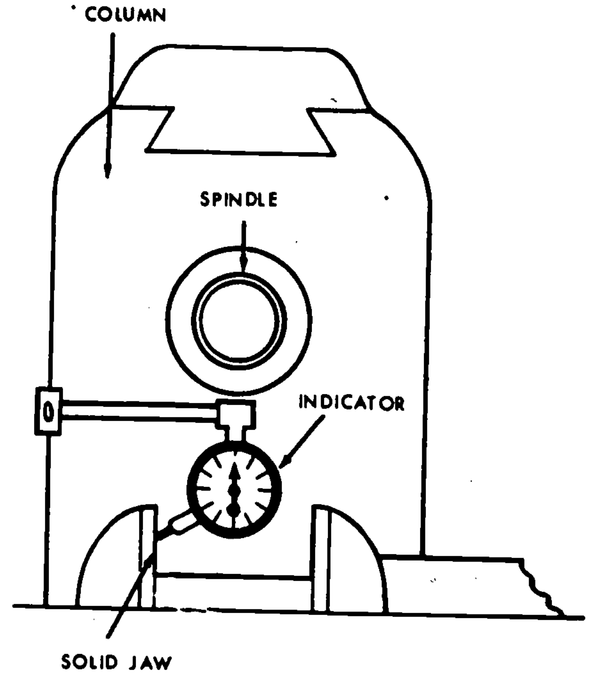
Use any suitable means to hold the work for face milling as long as the cutter clears the workholding device and the milling machine table. You can mount the work on parallels, if necessary, to provide clearance between the cutter and the table. Feed the work from the side of the cutter that will cause the cutter thrust to force the work down. If you hold the work in a vise, position the vise so that the cutter thrust is toward the solid jaw. The ends of the work are usually machined square to the sides of the work. Therefore, you will have to align the work properly. If you use a vise to hold the work, you can align the stationary vise jaw with a dial indicator, as shown in figure 11-56. You can also use a machinist's square and a feeler gage, as shown in figure 11-57.

Operation

To face mill the ends of work, such as the engine mounting block that we discussed previously:

1. Select and mount a suitable cutter.

2. Mount and position a vise on the milling machine table, as shown in figure 11-55 so that the thrust of the cutter is toward the solid vise jaw.



28.403
Figure 11-56.—Aligning vise jaws using an indicator.

3. Align the solid vise jaw square with the column of the machine, using a dial indicator for accuracy.

4. Mount the work in the vise, allowing the end of the work to extend slightly beyond the vise jaws.

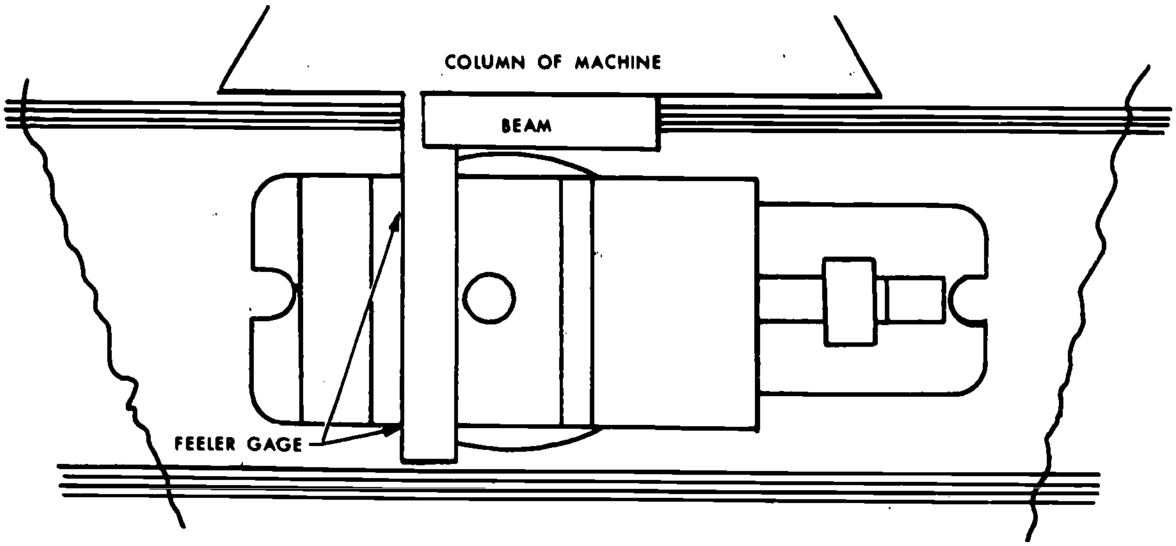
5. Raise the knee until the center of the work is approximately centered with the center of the cutter.

6. Lock the knee in position.

7. Set the machine for the proper roughing speed, feed, and table travel.

8. Start the spindle and pick up the end surface of the work by hand feeding the work toward the cutter.

9. Place a strip of paper between the cutter and the work as shown in figure 11-58 to help pick up the surface. When the cutter picks up the paper there is approximately .003-inch clearance between the cutter and the material being cut.

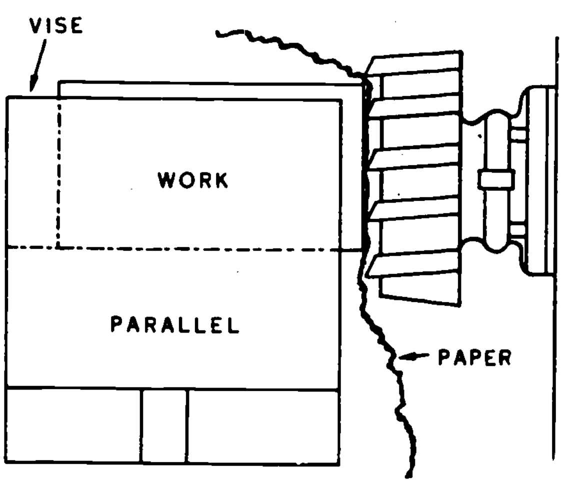


28.404
Figure 11-57.–Aligning vise jaws using a square.

10. Once the surface is picked up, set the saddle feed graduated dial at ZERO.

11. Move the work away from the cutter with the table and direct the coolant flow on the cutter.

12. Set the roughing depth of cut, using the graduated dial, and lock the saddle.



28.405
Figure 11-58.–Picking up the work surface.

13. Position the work to about 1/16 inch from the cutter, then engage the power feed.

14. After completing the cut, stop the spindle, and move the work back to the starting point before the next cut.

15. Set the speed and feed for the finishing cut, and then unlock the saddle.

16. Move it in for the final depth of cut and relock it.

17. Engage the spindle and take the finish cut.

18. Stop the machine and return the work to the starting place.

19. Shut the machine off.

20. Remove the work from the vise. Handle it very carefully to keep from cutting yourself before you can deburr the work.

21. Next, mount the work in the vise so that the other end is ready for machining. Mill this end in the same manner as the first, only now

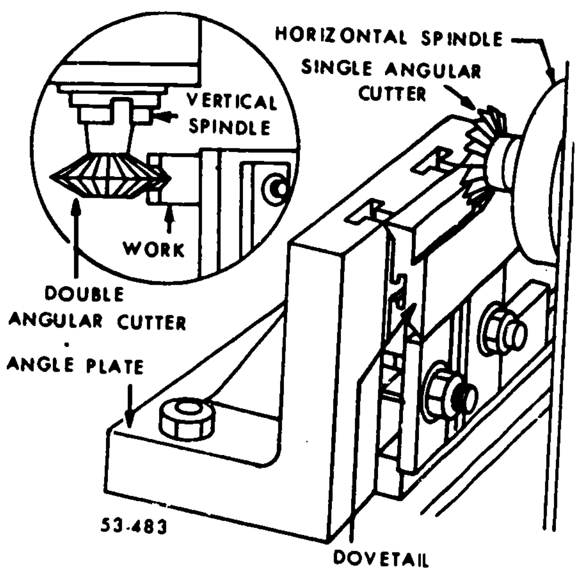
you must measure the length before taking the finishing cut. Before removing the work from the vise, check it for accuracy and remove the burrs from the newly finished end.

ANGULAR MILLING

Angular milling is the milling of a flat surface that is at an angle to the axis of the cutter. You can use an angular milling cutter, as shown in figure 11-59. However, you can perform angular milling with a plain, side, or face milling cutter by positioning the work at the required angle.

Many maintenance or repair tasks involve machining flat surfaces on cylindrical work. These tasks include milling squares and hexagons, and milling two flats in the same plane.

A square or hexagon is milled on an object to provide a positive drive, no slip area for various tools, such as wrenches and cranks. You will machine squares and hexagons frequently on the ends of bolts, taps, reamers, or other



28.406

Figure 11-59.—Angular milling.

items that are turned by a wrench and on drive shafts and other items that require a positive drive. The following information will help you to understand the machining of squares and hexagons.

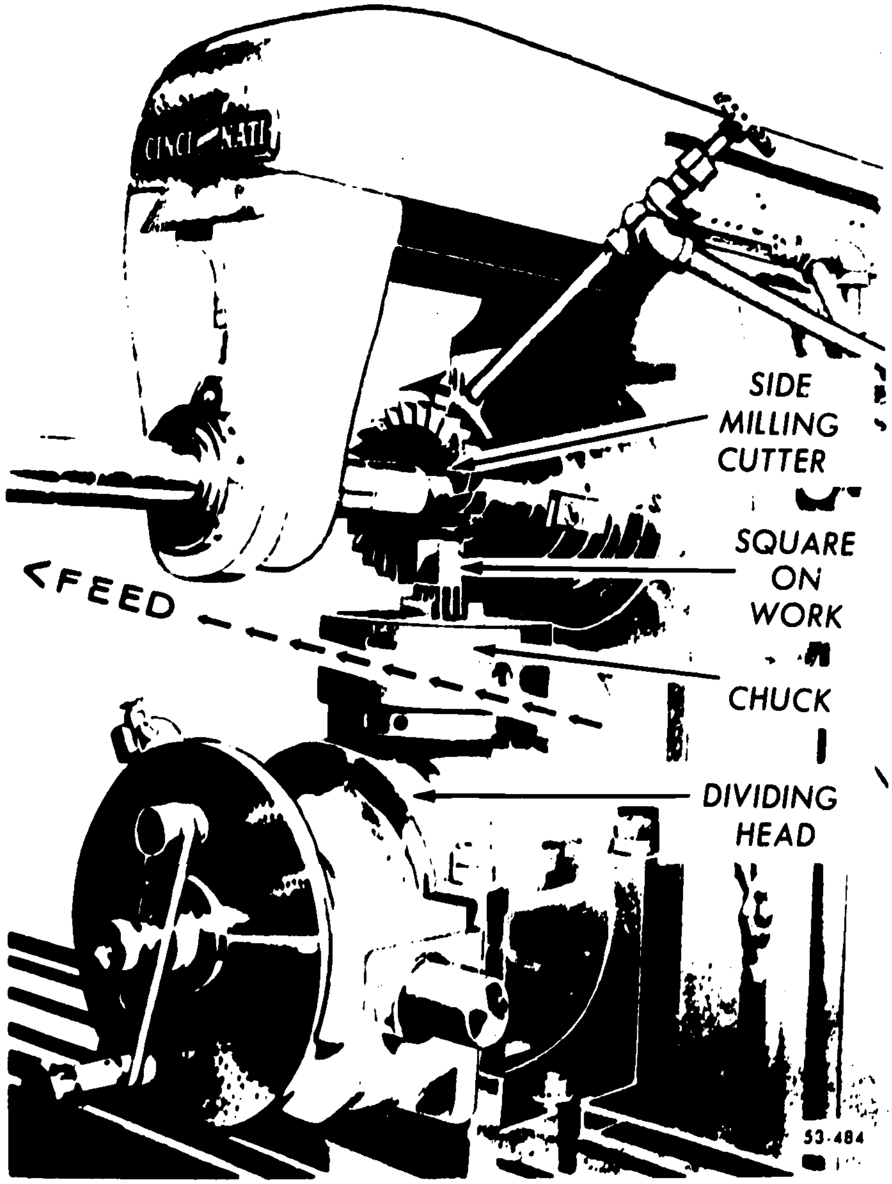
Cutter Setup

The two types of cutters you will use most often to machine squares or hexagons are side and end milling cutters. You can use side milling cutters for machining work that is held in a chuck and for heavy cutting. You can use end mills for work that is held in a chuck or between centers and for light cutting. If you use a side milling cutter, be sure that the cutter diameter is large enough so that you can machine the full length of the square or hexagon without interference from the arbor. If you use an end mill, be sure it is slightly larger in diameter than the length of the square or hexagon. The cutter thrust for both types should be up when the work is mounted vertically and down when it is mounted horizontally in order to use conventional (or up) milling.

The reason for what appears to be a contradiction in the direction of thrust is the difference in the direction of the feed. You can see this by comparing figures 11-60 and 11-61. The cutter shown in figure 11-60 rotates in a counterclockwise direction and the work is fed toward the left. The cutter shown in figure 11-61 rotates in a clockwise direction and the work is fed upward.

Work Setup

We have already discussed the methods that you will usually use to mount the work. Regardless of the workholding method that you use, you must align the index spindle in either the vertical or horizontal plane. If you are machining work between centers, you must also align the footstock center. If you use a screw-on chuck, take into consideration the cutter rotary thrust applied to the work. Always cut on the side of the work that will tend to tighten the chuck on the index head spindle. When you



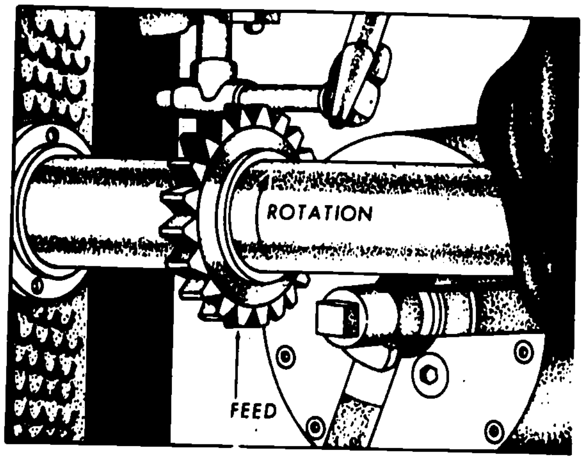
28.407

Figure 11-60.—Milling a sqaure on work held vertically.

mount work between centers, a dog rotates the work. The drive plate, shown in figure 11-62, contains two lock screws. One lock screw clamps the drive plate to the index center and ensures that the drive plate moves with the index spindle. The other lock screw clamps the tail of the dog against the side of the drive plate slot as shown in figure 11-62A. This eliminates any movement of the work during the machining operation. It may be necessary, especially if you are using a short end mill, to position the index head (fig. 11-62G) near the cutter edge of the table to ensure that cutter and work make contact.

Calculations

The following information will help you determine the amount of material you must remove to produce a square or a hexagon. The



28.408
Figure 11-61.—Milling a square on work held horizontally.

dimensions of the largest square or hexagon that you can machine from a piece of stock must be calculated.

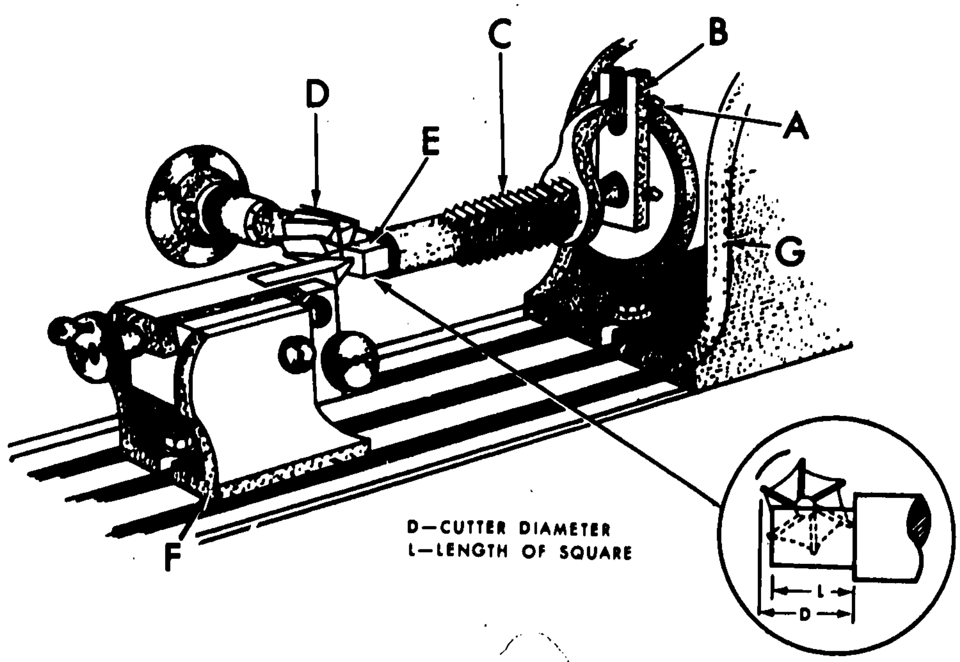
The size of a square (H in fig. 11-63) is measured across the flats. The largest square that you can cut from a given size of round stock equals the diameter of the stock in inches (which is also the diagonal of the square) times 0.707. This may be expressed as

Opposite side = Side of a square

Hypotenuse = Diagonal of square

$45^\circ = 90^\circ$ bisected

$$H = G \times 0.707 \text{ or } \frac{\text{Opposite side}}{\text{Hypotenuse}} = \text{sine } 45^\circ$$



28.409
Figure 11-62.—Milling a square using an end mill.

The diagonal of a square equals the distance across the flats times 1.414. This is expressed as

$$G = H \times 1.414 \text{ or } \frac{\text{Hypotenuse}}{\text{Opposite side}} = \text{cosec } 45^\circ$$

The amount of material that you must remove to machine each side of the square is equal to one-half the difference between the diameter of the stock and the distance across the flats.

$$I = \frac{G - H}{2}$$

You use the same formula $\left(I = \frac{G - H}{2}\right)$ to determine the amount of material to remove when you are machining a hexagon.

The size of the largest hexagon that you can machine from a given size of round stock (H in figure 11-64) is equal to the diagonal (the diameter of the stock) of the hexagon times 0.866 inch, or

Opposite side = Largest hexagon that can be machined

Hypotenuse = Diagonal or diameter of round stock

$$G = H \times 0.866 \text{ or } \frac{\text{Opposite side}}{\text{Hypotenuse}} = \text{sine } 60^\circ$$

The diagonal of a hexagon equals the distance across the flats times 1.155, or

$$H = G \times 1.155 \text{ or } \frac{\text{Hypotenuse}}{\text{Opposite side}} = \text{cosec } 60^\circ$$

The length of a flat is equal to one-half the length of the diagonal, or

$$r = \frac{G}{2}$$

We will explain two methods of machining a square or hexagon: machining work mounted in a chuck and machining work mounted between centers.

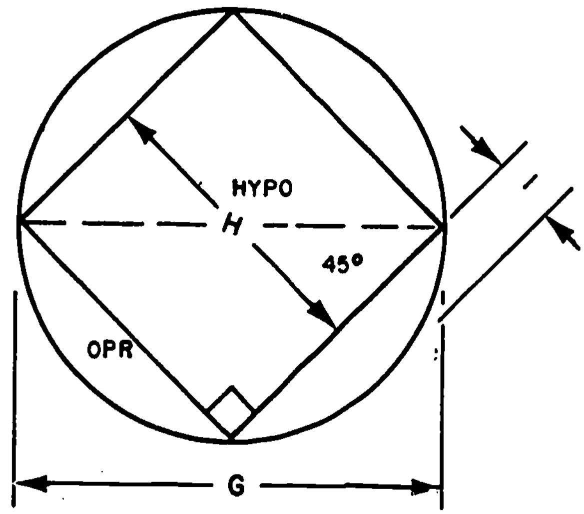


Figure 11-63.—Diagram of a square.

Square or Hexagon Work Mounted in a Chuck

You can machine a square or hexagon on work mounted in a chuck by using either a side milling cutter or an end mill. We will discuss using the side milling cutter first. Before placing

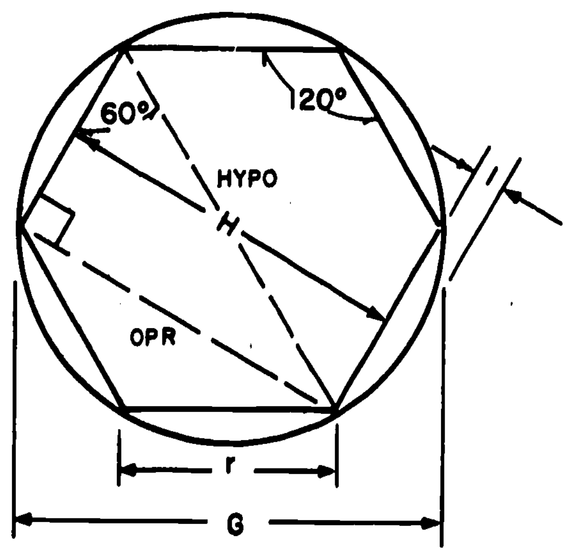


Figure 11-64.—Diagram of a hexagon.

the index head on the milling machine table, be sure that the table and the bottom of the index head have been cleaned of all chips and other foreign matter. Spread a thin film of clean machine oil over the area of the table to which the index head will be attached to prevent corrosion.

NOTE: Because most index heads are quite heavy and awkward, you should get someone to help you place the head on the milling machine table.

After the index head is mounted on the table, position the head spindle in the vertical position, as shown in figure 11-60. Use the degree graduations on the swivel block. This is accurate enough for most work requiring the use of the index head. The vertical position also means that the work will feed horizontally.

Then, tighten the work in the chuck to keep it from turning due to the cutter thrust. Install the arbor, cutter, and arbor support. The cutter should be as close as practical to the column. Remember, this is done so that the setup will be more rigid. Set the machine for the correct roughing speed and feed.

Machine the surface using the following steps:

1. With the cutter turning, pick up the cut on the end of the work.
2. Move the work sideways to clear the cutter.
3. Raise the knee a distance equal to the length of the flat surfaces to be cut.
4. Move the table toward the revolving cutter and pick up the side of the work. Use a piece of paper in the same manner as discussed earlier in this chapter.
5. Set the crossfeed graduated dial at ZERO.
6. Move the work clear of the cutter. Remember, the cutter should rotate so that the cutting action takes place as in "up milling."
7. Feed the table in the required amount for a roughing cut.
8. Engage the power feed and coolant flow.
9. When the cut is finished, stop the spindle and return the work to the starting point.
10. Loosen the index head spindle lock.
11. Rotate the work one-half revolution with the index crank.
12. Tighten the index head spindle lock.
13. Take another cut on the work.
14. When this cut is finished, stop the cutter and return the work to the starting point.
15. Measure the distance across the flats to determine whether the cutter is removing the same amount of metal from both sides of the work. If not, check your calculations and the setup for a possible mistake.
16. If the work measures as it should, loosen the index head spindle lock and rotate the work one-quarter revolution, tighten the lock, and take another cut.
17. Return the work to the starting point again.
18. Loosen the spindle lock.
19. Rotate the work one-half revolution.
20. Take the fourth cut.
21. Return the work again to the starting point and set the machine for finishing speed and feed.
22. Now, finish machine opposite sides (1 and 3), using the same procedures already mentioned.
23. Check the distance across these sides. If it is correct, finish machine the two remaining sides.
24. Deburr the work and check it for accuracy.

NOTE: You can also machine a square or hexagon with the index head spindle in the horizontal position, as shown in figures 11-61 and 11-62. If you use the horizontal setup, you must feed the work vertically.

Square or Hexagon Work Mounted Between Centers

Machining a square or hexagon on work mounted between centers is done in much the same manner as in holding the work in a chuck.

1. Mount the index head the same way, only with the spindle in a horizontal position. The feed will be in a vertical direction.

2. Insert a center in the spindle and align it rith the footstock center.

3. Select and mount the desired end mill, referably one whose diameter is slightly greater han the length of the flat you are to cut, as hown in figure 11-62.

4. Mount the work between centers. Make ure that the drive dog is holding the work ecurely.

5. Set the machine for roughing speed and ed.

6. Pick up the side of the work and set the raduated crossfeed dial at ZERO.

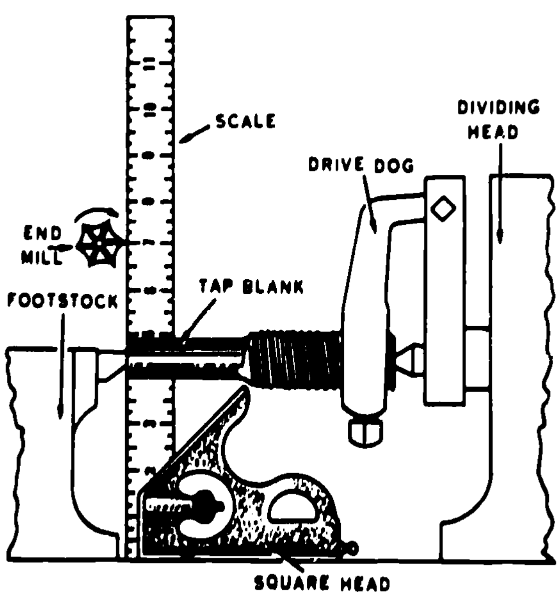
7. Lower the work until the cutter clears ne footstock.

8. Move the work until the end of the work clear of the cutter.

9. Align the cutter with the end of the ork. Use a square head and rule, as shown in gure 11-65. NOTE: Turn the machine off efore aligning the cutter by this method.

10. Move the table a distance equal to the ngth of the flat desired.

11. Move the saddle the distance required r the roughing depth of cut.



28.410

Figure 11-65.—Aligning the work and cutter.

12. While feeding the work vertically, machine side 1. Lower the work below the cutter when you have completed the cut.

13. Loosen the index head spindle lock and index the work one-half revolution to machine the flat opposite side 1.

14. Tighten the lock.

15. Engage the power feed. After completing the cut, again lower the work below the cutter and stop the cutter.

16. Measure the distance across the two flats to check the accuracy of the cuts. If it is correct, index the work one-quarter revolution to machine another side. Then lower the work, index one-half revolution, and machine the last side. Remember to lower the work below the cutter again.

17. Set the machine for finishing speed, feeds, and depth of cut, and finish machine all the sides.

18. Deburr the work and check it for accuracy.

Machining Two Flats in One Plane

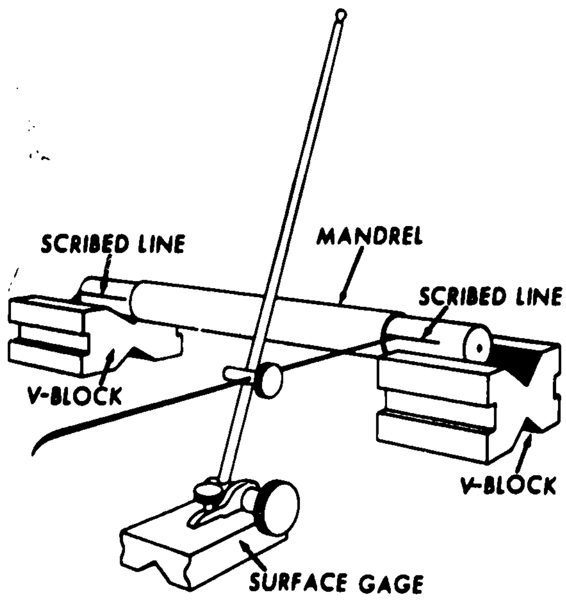
You will often machine flats on shafts to serve as seats for setscrews. One flat is simple to machine. You can machine it in any manner with a side or end mill, as long as you can mount the work properly. However, machining two flats in one plane, such as the flats on the ends of a mandrel, presents a problem since the flats must align with each other. A simple method of machining the flats is to mount the work in a vise or on V-blocks in such a manner that you can machine both ends without moving the work once it has been secured.

We will describe the method that is used when the size or shape of the work requires repositioning it in order to machine both flats.

1. Apply layout dye on both ends of the work.

2. Place the work on a pair of V-blocks, as shown in figure 11-66.

3. Set the scriber point of the surface gage to the center height of the work. Scribe horizontal lines on both ends of the work, as illustrated in figure 11-66.



28.411
Figure 11-66.—Layout of work.

4. Mount the index head on the table with its spindle in the horizontal position.

5. Again, set the surface gage scriber point, but to the centerline of the index head spindle.

6. Insert the work in the index head chuck with the end of the work extended far enough to permit all required machining operations.

7. To align the surface gage scriber point with the scribed horizontal line, rotate the index head spindle.

8. Lock the index head spindle in position.

These flats can be milled with either an end mill or a side mill or a side milling cutter.

CAUTION: Rotate the cutter in a direction that will cause the thrust to tighten the index head chuck on the spindle when you use a screw-on type chuck.

9. Raise the knee with the surface gage still set at center height until the cutter centerline is aligned with the scriber point. This puts the centerlines of the cutter and the work in alignment with each other.

10. Position the work so that a portion of the flat to be machined is located next to the cutter. Because of the shallow depth of cut, compute the speed and feed as if the cuts were finishing cuts.

11. After starting the machine, feed the work by hand so that the cutter contacts the side of the work on which the line is scribed.

12. Move the work clear of the cutter and stop the spindle.

13. Check to see if the greater portion of the cutter mark is above or below the layout line. Depending on its location, rotate the index head spindle as required to center the mark on the layout line.

14. Once the mark is centered, take light "cut and try" depth of cuts until you reach the desired width of the flat.

15. Machine the flat to the required length.

16. When one end is completed, remove the work from the chuck. Turn the work end for end and reinsert it in the chuck.

17. Machine the second flat in the same manner as you did the first.

18. Deburr the work and check it for accuracy.

19. Check the flats to see if they are in the same plane by placing a matched pair of parallels on a surface plate and one flat on each of the parallels. If the flats are in the same plane, you will not be able to wobble the work.

SLOTTING, PARTING, AND MILLING KEYSEATS AND FLUTES

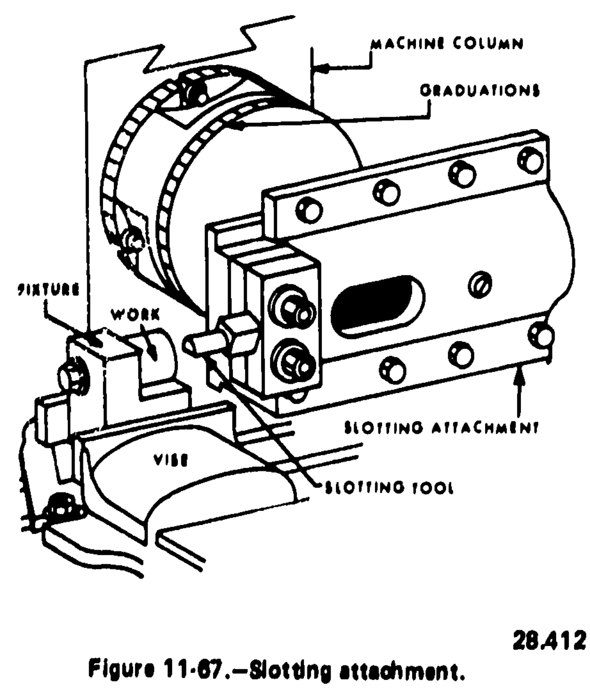
Slotting, parting, and milling keyseats and flutes are all operations that involve cutting grooves in the work. These grooves are of various shapes, lengths, and depths, depending on the requirements of the job. They range from flutes in a reamer to a keyseat in a shaft to the parting off of a piece of metal to a predetermined length.

Slotting

You can cut internal contours, such as internal gears and splines and six- or twelve-point sockets by slotting. Most slotting is lone with a milling machine attachment called a lotting attachment, as shown in figure 11-67. The slotting attachment is fastened to the nilling machine column and driven by the pindle. This attachment changes the rotary notion of the spindle to a reciprocating motion nuch like that of a shaper. You can vary the ength of the stroke within a specified range. A ointer on the slotting attachment slide ndicates the length of the stroke. You can pivot he head of the slotting attachment and position t at any desired angle. Graduations on the base of the slotting attachment indicate the angle at which the head is positioned. The number of trokes per minute is equal to the spindle rpm nd is determined by the formula

$$\text{Strokes per minute} = \frac{\text{CFS X 4}}{\text{length of stroke}}$$

The cutting tools which you use with lotting attachments are ground to any desired hape from high-speed steel tool blanks. These ools are then clamped to the front of the slide r ram. You can use any suitable means for olding the work, but the most common nethod is to hold the work in an index head chuck. If the slotted portion does not extend through the work, you will have to machine an internal recess in the work to provide clearance for the tool runout. When it is possible, position the slotting attachment and the work in the vertical position to provide the best possible view of the cutting action of the tool.



28.412
Figure 11-67.—Slotting attachment.

Parting

Use a metal slitting saw for sawing or parting operations and for milling deep slots in metals and in a variety of materials. Efficient sawing depends to a large extent upon the slitting saw you select. The work required of slitting saws varies greatly. It would not be efficient to use the same saw to cut very deep narrow slots, part thick stock, saw thin stock, or saw hard alloy steel. Soft metals, such as copper and babbitt, or nonmetallic materials, such as bakelite, fiber, or plastic, require their own styles of slitting saw.

Parting with a slitting saw leaves pieces that are reasonably square and that require the removal of a minimum of stock in finishing the surface. You can cut off a number of pieces of varying lengths and with less waste of material than you could saw by hand.

A coarse-tooth slitting saw is best for sawing brass and for cutting deep slots. A fine-tooth slitting saw is best for sawing thin metal, and a staggered-tooth slitting saw is best for making heavy deep cuts in steel. You should use slower feeds and speeds to saw steels to prevent cutter breakage. Use conventional milling in sawing thick material. In sawing thin material, however, clamp the stock directly to the table and use down milling. Then the slitting saw will tend to force the stock down on the table. Position the work so that the slitting saw extends through the stock and into a table T-slot.

External Keyseat

Machining an external keyseat on a milling machine is less complicated than machining it on a shaper. In milling, starting an external keyseat is no problem. You simply bring the work in contact with a rotating cutter and start cutting. It should not be difficult for you to picture in your mind how you would mill a straight external keyseat with a plain milling cutter or an end mill. If the specified length of the keyseat exceeds the length you can obtain by milling to the desired depth, you can move the work in the direction of the slot to obtain the desired length. Picturing in your mind how you would mill a Woodruff keyseat should be easier. The secret is to select a cutter that has the same diameter and thickness as the key.

Straight External Keyseats

Normally, you would use a plain milling cutter to mill a straight external keyseat. You could use a Woodruff cutter or a two-lipped end mill.

Before you can begin milling the keyseat, you must align the axis of the work with the midpoint of the width of the cutter. Figure 11-68 shows one method of alignment.

Suppose that you are going to cut a keyseat with a plain milling cutter. Move the work until the side of the cutter is tangent to the circumference of the work. With the cutter turning very slowly and before contact is made, insert a piece of paper between the work and the side of the cutter. Continue moving the work toward the cutter until the paper begins to tear. When it does, lock the graduated dial at ZERO on the saddle feed screw. Then lower the milling machine knee. Use the saddle feed dial as a guide, and move the work a distance equal to the radius of the work plus one-half the width of the cutter.

You use a similar method to align work with an end mill. When you use an end mill, move the work toward the cutter while you hold a piece of paper between the rotating cutter and the work, as shown in figure 11-69. After the paper tears, lower the work to just below the bottom of the end mill. Then move the work and saddle

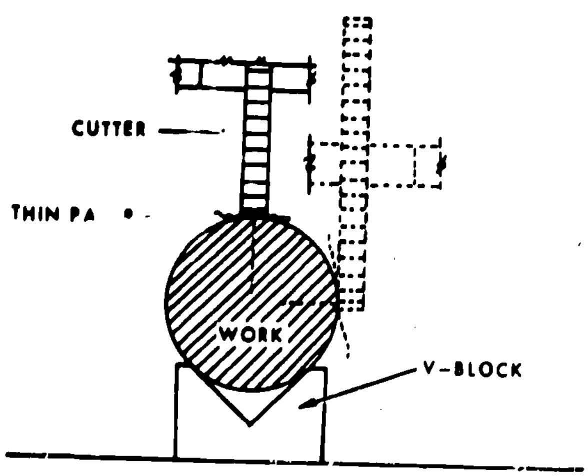
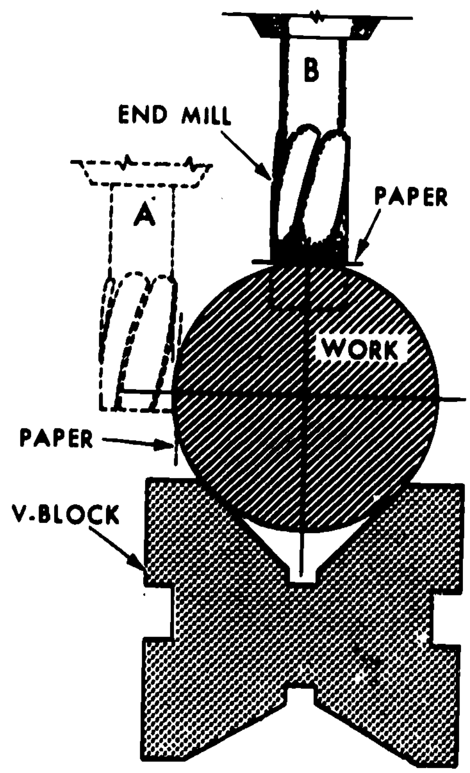


Figure 11-68.—Aligning cutter using a paper strip.



28.413
Figure 11-69.—Aligning an end mill with the work.

a distance equal to the radius of the work plus the radius of the end mill. Move the work up, using hand feed, until a piece of paper held between the work and the bottom of the end mill begins to tear, as shown in figure 11-69B. Then move the table and work away from the bottom of the end mill. Set and lock the graduated dial at ZERO on the vertical feed, and then feed up for the roughing cut. The cutter rpm and the longitudinal feed are computed in the same manner as for conventional milling cutters. Because of the higher speeds and feeds involved, more heat is generated, so flood the work and cutter with coolant.

When extreme accuracy is not required, you can align the work with the cutter visually, as shown in figure 11-70. Position by eye the work as near as possible to the midpoint of the cutter. Make the final alignment by moving the work in or out a slight amount, as needed. The cutter should be at the exact center of the work diameter measurement of the steel rule. You can use this method with both plain milling cutters and end mills.

Before you begin to machine the keyseat, you should measure the width of the cut. You cannot be certain that the width will be the same as the thickness of the cutter. The cutter may not run exactly true on the arbor or the arbor may not run exactly true on the spindle. The recommended practice is to nick the end of the work with the cutter and then measure the width of the cut.

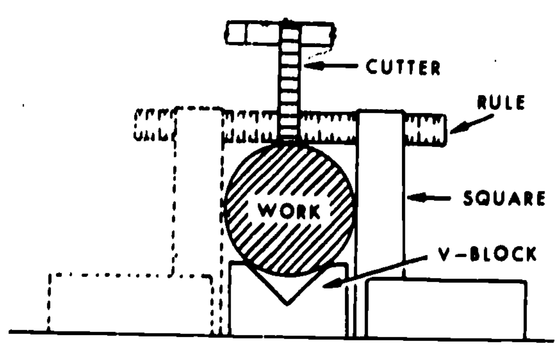


Figure 11-70.—Visual alignment of a cutter.

Specifications for the depth of cut are usually furnished. When specifications are not available, the total depth of cut for a square keyseat may be determined by the following formula and dimensions in figure 11-71.

$$\text{Total depth of cut } (T) = d + f$$

where

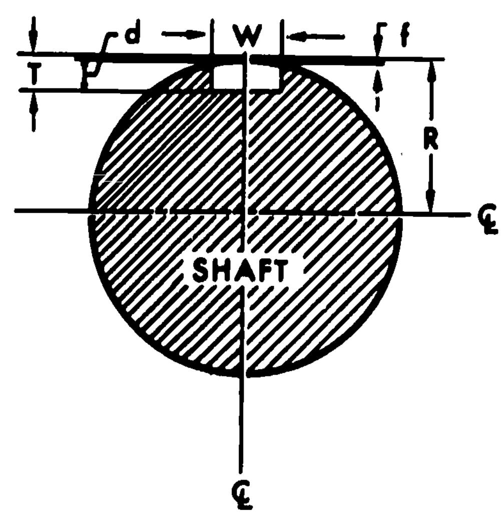
$$d = \frac{W}{2} = \text{depth of keyseat}$$

$$f = R - \sqrt{R^2 - \left(\frac{W}{2}\right)^2} = \text{height of arc}$$

W = width of key

R = radius of shaft

The height of arc (f) for various sizes of shafts and keys is shown in table 11-1. Keyseat dimensions for rounded end and rectangular keys are contained in the *Machinery's Handbook*. Check keyseats for accuracy with rules, outside and depth micrometers, vernier calipers, and go-no-go gages. Use table 11-1 for



28.414

Figure 11-71.—Keyseat dimensions for straight square key.

Table 11-1.—Values for Factor (f) for Various Sizes of Shafts

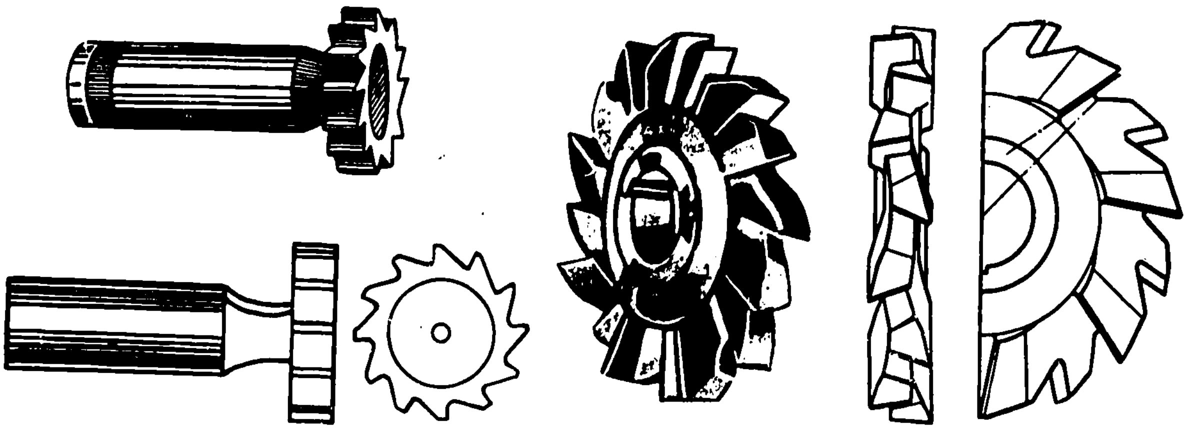
	WIDTH OF KEY IN INCHES							
DIAMETER OF SHAFT (INCHES)	1/16	3/32	1/8	5/32	3/16	7/32	1/4	5/16
SHAFT SIZE	FACTOR (f)							
1/2	.002	.004	.008	.013	.018	.025	.033	---
5/8	.001	.003	.006	.010	.014	.019	.025	.042
3/4	.001	.003	.005	.008	.012	.016	.022	.034
7/8	.001	.002	.004	.007	.010	.014	.018	.028
1	.001	.002	.004	.006	.009	.012	.015	.024
1 1/8	----	.002	.003	.005	.008	.011	.014	.022
1 1/4	----	.002	.003	.005	.007	.010	.013	.019
1 1/2	----	.001	.002	004	.006	.008	.011	.016
1 3/4	----	.001	.002	.003	.005	.007	.009	.014

both square and Woodruff keyseats, which will be explained next.

Woodruff Keyseat

A Woodruff key is a small half-disk of metal. The rounded portion of the key fits in the slot in the shaft. The upper portion fits into a slot in a mating part, such as a pulley or gear. You align the work with the cutter and measure the width of the cut in exactly the same manner as for milling straight external keyseats.

A Woodruff keyseat cutter (fig. 11-72) has deep flutes cut across the cylindrical surface of the teeth. The cutter is slightly thicker at the crest of the teeth than it is at the center. This feature provides clearance between the sides of the slot and the cutter. Cutters with a 2-inch



28.415

Figure 11-72.—Woodruff keyseat cutter.

diameter and larger have a hole in the center for arbor mounting. On smaller cutters the cutter and the shank are one piece. Note that the shank is "necked in" back of the cutting head to give additional clearance. Also, note that large cutters usually have staggered teeth to improve the cutting action.

As discussed earlier, to mill a Woodruff keyseat in a shaft, you use a cutter that has the same diameter than thickness as the key. Cutting a Woodruff keyseat is relatively simple. You simply move the work up into the cutter until you obtain the desired keyseat depth. The work may be held in a vise, chuck, between centers, or clamped to the milling machine table. The cutter is held on an arbor, or in a spring collet or drill chuck that has been mounted in the spindle of the milling machine, as in figure 11-73.

In milling the keyseat, centrally locate the cutter over the position in which the keyseat is to be cut and parallel with the axis of the work. Raise the work by using the hand vertical feed until the revolving cutter tears a piece of paper held between the teeth of the cutter and the work. At this point, set the graduated dial on the vertical feed at ZERO and set the clamp on the table. With the graduated dial as a guide, raise the work by hand until the full depth of the keyseat is cut. If specifications for the total depth of cut are not available, use the following formula to determine the correct value:

$$\text{Total depth } (T) = d + f$$

where

$$d \text{ (depth of the keyseat)} = H - \frac{W}{2}$$

H = total height of the key

W = width of the key

The most accurate way to check the depth of a Woodruff keyseat is to insert a Woodruff key of the correct size in the keyseat. Measure over the key and work with an outside micrometer to obtain the distance M in figure 11-74. Measure the correct micrometer reading over the shaft and key by using the formula

$$M = D + \frac{(W)}{(2)} - f$$

where

M = micrometer reading

D = diameter of shaft

W = width of key

f = height of arc

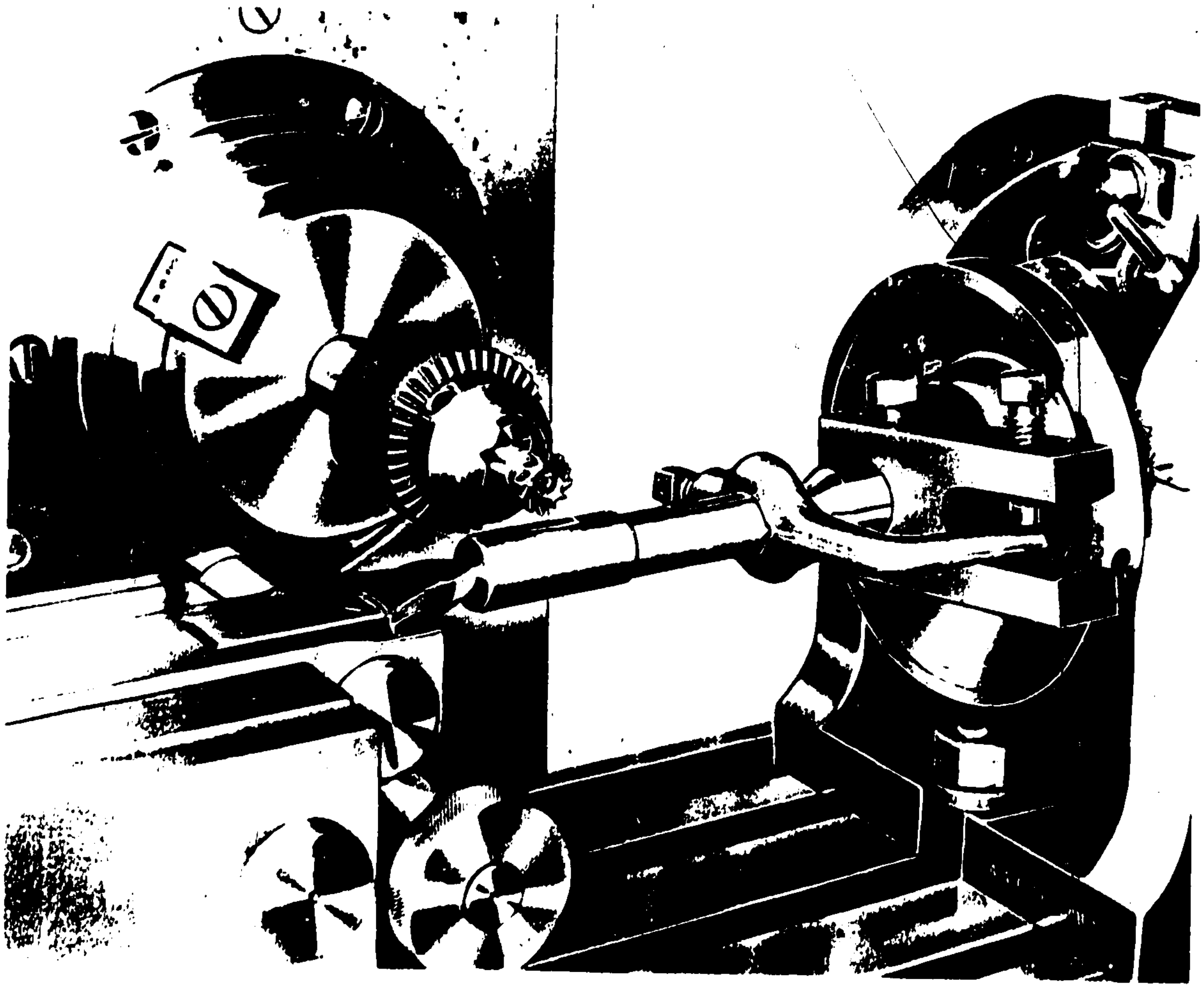


Figure 11-73.—Milling a Woodruff keyseat.

28.416

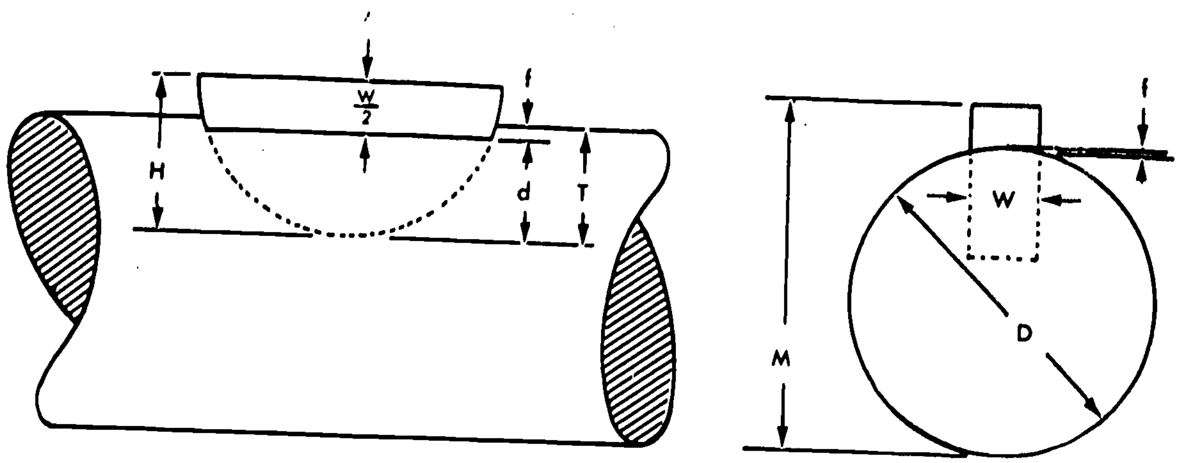


Figure 11-74.—Dimensions for Woodruff keyseat.

28.417

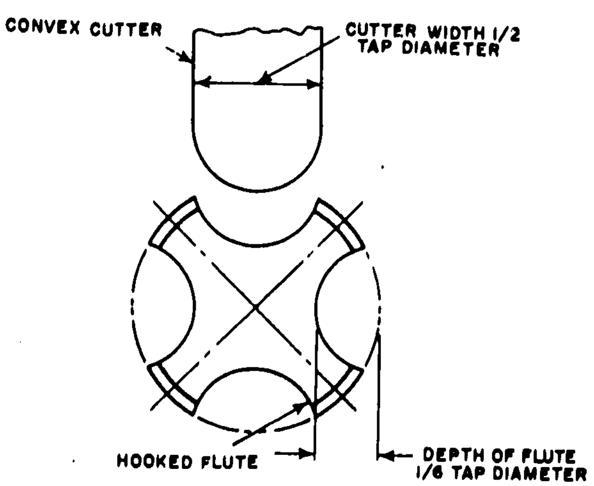
NOTE: Tables in some references may differ slightly from the above calculation for the value M, due to greater allowance for clearance at the top of the key.

Straight Flutes

The flutes on cutting tools serve three purposes. They form the cutting edge for the tool, provide channels for receiving and discharging chips, and let coolant reach the cutting edges. The shape of the flute and tooth depends upon the cutter you use to machine the flute. The following information pertains specifically to taps and reamers. Since flutes are actually special purpose grooves, you can apply much of the information to grooves in general.

Tap Flutes

You usually use a convex cutter to machine tap flutes. This type of cutter produces a "hooked" flute as shown in figure 11-75. The number of flutes is determined by the diameter of the tap. Taps 1/4 inch to 1 3/4 inches in diameter usually have four flutes, and taps 1 7/8 inches (and larger) in diameter usually have six flutes. The width of the convex cutter should be equal to one-half the tap diameter. The depth of



28.418
Figure 11-75.—Hooked tap flutes.

the flute is normally one-fourth the tap diameter. The minimum length of the full depth of the flute should be equal to the length of the threaded portion of the tap. Table 11-2 lists the width of the cutter and the depth of the flutes for taps of various diameters. You usually mount the tap blank between centers and feed it longitudinally past the cutter. For appearance sake, the flutes are usually cut in the same plane as the sides of the square on the tap blank.

You can mill the flutes on a tap blank in the following manner:

1. Mount and align the index centers.

2. Set the surface gage to center height.

3. Place the tap blank between the centers with one flat of the square on the tap shank in a vertical position.

4. Align the flat with a square head and blade.

5. Scribe a line on the tap shank.

6. Remove the tap blank, place a dog on the shank, and remount the blank between centers.

7. Align the scribed line with the point of the surface gage scriber.

8. Make sure that the surface gage is still at center height.

9. Mount the convex cutter.

10. Make sure that the direction of the cutter rotation is correct for conventional (or up) milling and that the thrust is toward the index head.

11. Align the center of the cutter with the axis of the tap blank.

12. Pick up the surface of the tap.

13. Set the table trip dogs for the correct length of cut.

14. Set the machine for roughing speed and feed.

15. Rough mill all flutes to within 0.015 to 0.020 inch of the correct depth.

Table 11-2.—Tap Flute Dimensions

Diameter of tap (inches)	Width of cutter (inches)	Depth of flute (inches)
1/8	1/16	1/32
1/4	1/8	1/16
1/2	1/4	1/8
3/4	3/8	3/16
1	1/2	1/4
1 1/4	5/8	5/16
1 1/2	3/4	3/8
1 3/4	7/8	7/16
2	1	1/2
2 1/4	1 1/8	9/16
2 1/2	1 1/4	5/8
2 3/4	1 3/4	11/16
3	1 1/2	3/4

16. Set the machine for finishing speed and feed and finish machine all flutes to the correct size.

17. Remove the work, deburr, and check it for accuracy.

Reamer Flutes

Flutes may be milled on reamers with angular fluting cutters, but you normally use special formed fluting cutters. The advantages of the formed flute compared to the flute milled with an angular cutter are that the chips are more readily removed and the cutting tooth is stronger. Also, the tooth is less likely to crack or warp during heat treatment. Formed reamer fluting cutters have a 6° angle on one side and a radius on the other side. The size of the radius depends upon the size of the cutter. Reamer fluting cutters are manufactured in eight sizes. The size of the cutter is identified by a number (1 through 8). Reamers from 1/8 inch to 3 inches in diameter are fluted by the eight sizes of cutters. The correct cutters for fluting reamers of various diameters are given in table 11-3. You machine reamer teeth with a slight negative rake to help prevent chatter. To obtain the negative rake, position the work and cutter slightly ahead of the reamer center, as shown in figure 11-76.

Table 11-4 lists the recommended offset for reamers of various sizes. Straight reamer flutes are usually unequally spaced to help prevent chatter. To obtain the unequal spacing, index the required amount as each flute is cut. The recommended variation is approximately 2°. Machinists' publications, such as the *Machinery's Handbook*, contain charts that list the number of holes to advance or retard the index crank to machine a given number of flutes when you use a given hole circle. You normally mill the flutes in pairs. After you have machined one flute, index the work one-half revolution and mill the opposite flute.

The depth of the flute is determined by trial and error. The approximate depth of flute to obtain the recommended width of land is one-eighth the diameter for an eight-fluted reamer, one-sixth the diameter for a six-fluted reamer, etc.

You can machine the flutes on a hand reamer in the following manner:

1. Mount the reamer blank between centers and the reamer fluting cutter on the arbor.

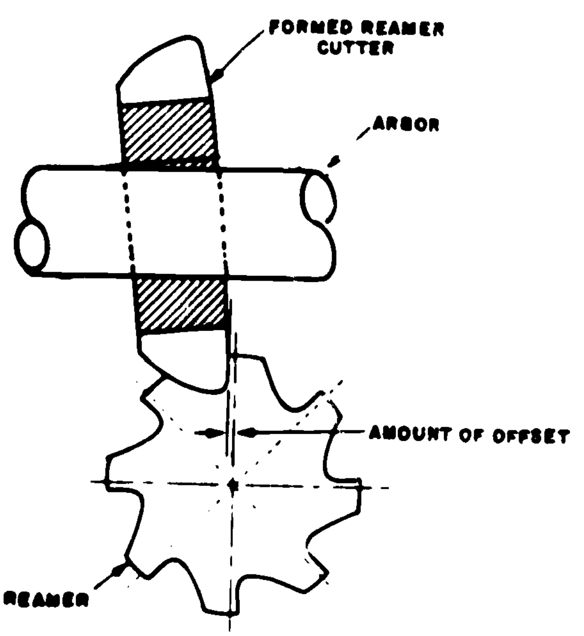
Table 11-3.–Reamer Fluting Cutter Numbers

Cutter number	Reamer diameter (inches)	Number of reamer flutes
1	1/8 to 3/16	6
2	1/4 to 5/16	6
3	3/8 to 7/16	6
4	1/2 to 11/16	6 to 8
5	3/4 to 1	8
6	1 1/16 to 1 1/2	10
7	1 9/16 to 2 1/8	12
8	2 1/4 to 3	14

2. Align the point of the cutter with the reamer blank axis and just touch the surface of the reamer with the rotating cutter.

3. Remove the work blank.

4. Then raise the table a distance equal to the depth of the flute plus one-half the grinding allowance.



28.419A

Figure 11-76.–Negative rake tooth.

5. Rotate the cutter until a tooth is in the vertical position.

6. Shut off the machine.

7. Move the table until the point of the footstock center is aligned with the tooth that is in the vertical position.

8. Place an edge of a 3-inch rule against the 6° surface of the reamer tooth. Move the saddle until the edge of the 3-inch rule that is contacting the cutter tooth is aligned with the point of the footstock center.

Table 11-4.–Required Offset

Size of reamer (inches)	Offset of cutter (inches)
1/4	0.011
3/8	0.016
1/2	0.022
5/8	0.027
3/4	0.033
7/8	0.038
1	0.044
1 1/4	0.055
1 1/2	0.066
1 3/4	0.076
2	0.087
2 1/4	0.098
2 1/2	0.109
2 3/4	0.120
3	0.131

9. To eliminate backlash, move the saddle in the same direction it will be moved when you offset the cutter. Continue feeding the saddle until you get the desired amount of offset; then lock it in position.

10. Move the table until the cutter clears the end of the reamer blank.

11. Remount the blank between the centers.

12. Calculate the indexing required to space the flutes unequally.

13. Set the table feed trip dogs so that the minimum length of the full depth of flute is equal to the length of the reamer teeth.

14. Rough machine all flutes.

NOTE: Write down the exact indexing which you used for each of the flutes to avoid confusion when you index for the finish cut.

Fly Cutting

You will use a fly cutter when a formed cutter is required but is not available. Fly cutters are high-speed steel tool blanks that have been ground to the required shape. Any shape can be

28.419

Figure 11-77.—Offset boring head and boring tools.

ground on the tool if the cutting edges are given a sufficient amount of clearance. Fly cutters are mounted in fly cutter arbors, such as the one shown in figure 11-44. Use a slow feed and a shallow depth of cut to prevent breaking the tool. It is a good idea to rough out as much excess material as possible with ordinary cutters and to use the fly cutter to finish shaping the surface.

DRILLING, REAMING, AND BORING

Drilling, reaming, and boring are operations that you can do very efficiently on a milling machine. The graduated feed screws make it possible to accurately locate the work in relation to the cutting tool. In each operation the cutting tool is held and rotated by the spindle, and the work is fed into the cutting tool.

Drilling and Reaming

You use the same drills and reamers that you use for drilling and reaming in the lathe and drill press. Drills and reamers are held in the spindle by the same methods that you use to hold straight and taper-shanked end mills. The work may be held in a vise, clamped to the table, held in fixtures or between centers, and in index head chucks, as is done for milling. The speeds used for drilling and reaming are determined in the same manner as for drilling and reaming in the lathe or drill press. The work is fed into the drill or reamer either by hand or power feed. If the cutting tool is held in a horizontal position, the transverse or saddle feed is used. When the drill or reamer is held in a vertical position, as in a vertical type machine, you use the vertical feed.

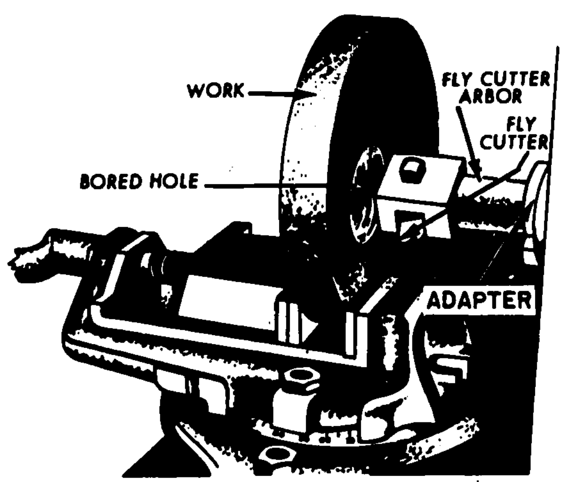
Boring

Of the three operations, the only one that warrants special treatment is boring. On a milling machine you usually bore holes with an offset boring head. Figure 11-77 shows several views of an offset boring head and several boring tools. Note that the chuck jaws, which grip the boring bar, can be adjusted at a right angle to the spindle axis. This feature lets you accurately position the boring cutter to bore holes of varying diameters. This adjustment is more convenient than adjusting the cutter in the boring bar holder or by changing boring bars.

Although the boring bars are the same on a milling machine as on a lathe or drill press, the manner in which they are held is different. Note in figure 11-78 that a boring bar holder is not used. The boring bar is inserted into an adapter, and the adapter is fastened in the hole in the adjustable slide. Power for driving the boring bar is transmitted directly through the shank. The elimination of the boring bar holder results in a more rigid boring operation, but the size of the hole that can be bored is more limited than in boring on a lathe or drill press.

Fly cutters, which we discussed previously, can also be used for boring, as shown in figure 11-78. A fly cutter is especially useful for boring relatively shallow holes. The cutting tool must be adjusted for each depth of cut.

The speeds and feeds you should use in boring on a milling machine are comparable to those you would use in boring on a lathe or drill press and depend upon the same factors: hardness of the metal, kind of metal in the cutting tool, and depth of cut. Because the boring bar is a single-point cutting tool, the diameter of the arc through which the tool



28.420

Figure 11-78.—Boring with a fly cutter.

moves is also a factor. For all of these reasons, you must guard against operating at too great a speed or vibration will occur.

MILLING MACHINE ATTACHMENTS

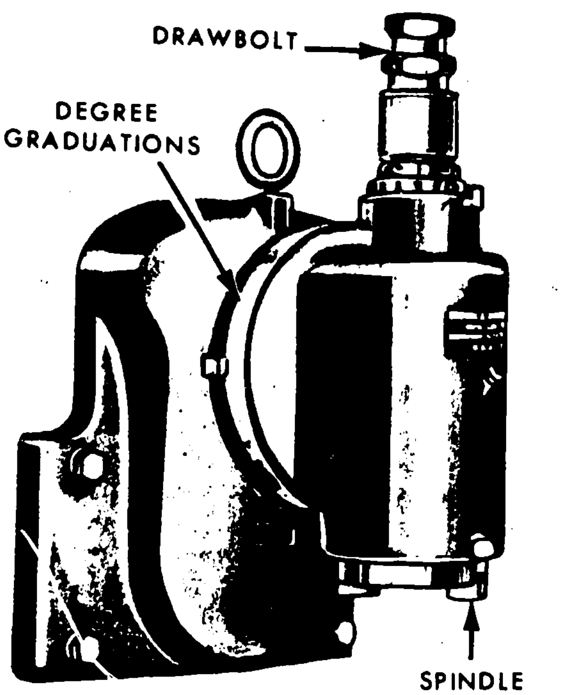
Many attachments have been developed which increase the number of jobs a milling machine can do or which make such jobs easier to do.

VERTICAL MILLING ATTACHMENT

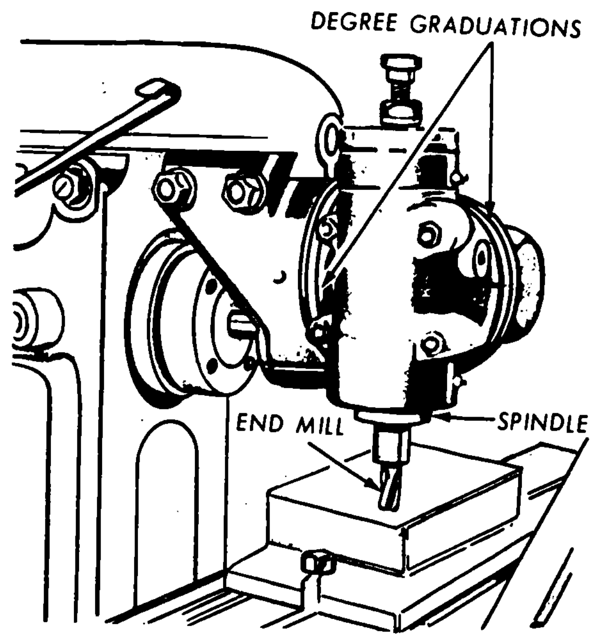
For instance, by using a vertical milling attachment (fig. 11-79) you can convert the horizontal spindle to a vertical spindle machine and swivel the cutter to any position in the vertical plane. By using a universal milling attachment, you can swivel the cutter to any position in both the vertical and horizontal planes. These attachments will enable you to do more easily jobs which would otherwise be very complex.

HIGH-SPEED UNIVERSAL ATTACHMENT

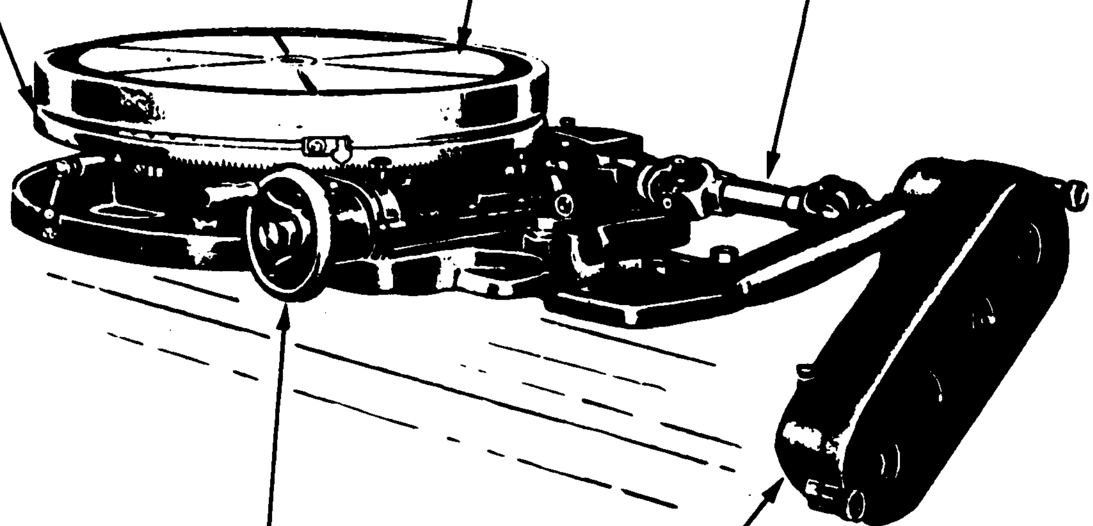
By using a high-speed universal attachment, you can perform milling operations at higher speeds than those for which the machine was designed. This attachment is clamped to the machine and is driven by the milling machine spindle, as you can see in figure 11-80. You can swivel the attachment spindle head and cutter 360° in both planes. The attachment spindle is driven at a higher speed than the machine spindle. You must consider the ratio between the rpm of the two spindles when you calculate cutter speed. Small cutters, end mills, and drills should be driven at a high rate of speed to maintain an efficient cutting action.



28.421
Figure 11-79.—Vertical milling attachment.



28.422
Figure 11-80.—High-speed universal milling attachment.



28.423

Figure 11-81.–Circular milling attachment with power feed.

CIRCULAR MILLING ATTACHMENT

This attachment (fig. 11-81) is a circular table that is mounted on the milling machine table. The circumference of the table is graduated in degrees. Smaller attachments are usually equipped for hand feed only, and larger ones are equipped for both hand and power feed. This attachment may be used for milling circles, arcs, segments, circular T-slots, and internal and external gears. It may also be used for irregular form milling.

RACK MILLING ATTACHMENT

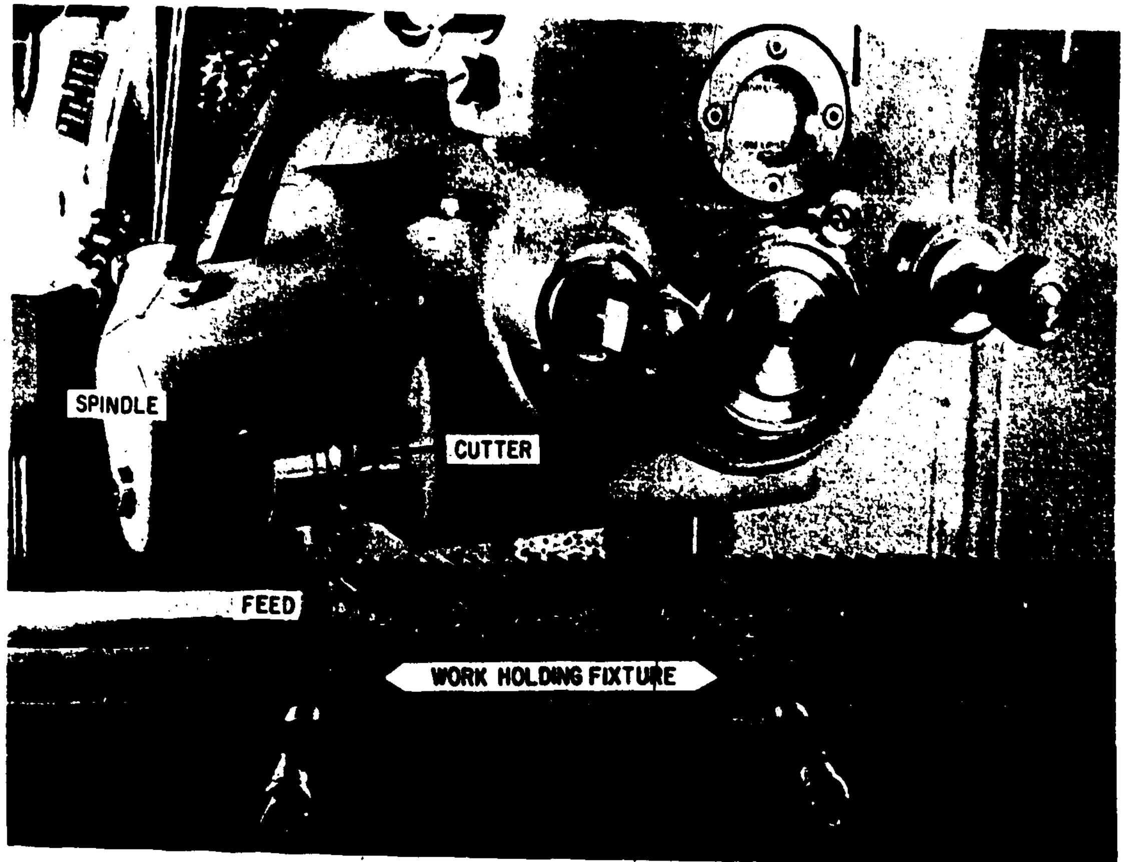
The rack milling attachment, shown in figure 11-82, is used primarily for cutting teeth on racks, although it can be used for other operations. The cutter is mounted on a spindle that extends through the attachment parallel to the table T-slots. An indexing arrangement is used to space the rack teeth quickly and accurately.

RIGHT-ANGLE PLATE

The right-angle plate (fig. 11-83) is attached to the table. The right-angle slot permits mounting the index head so that the axis of the head is parallel to the milling machine spindle. With this attachment you can make work setups which are off center or at a right angle to the table T-slots. The standard size plate T-slots make it convenient to change from one setting to another for milling a surface at a right angle.

RAISING BLOCK

Raising blocks (fig. 11-84) are heavy-duty parallels which usually come in matched pairs. They are mounted on the table, and the index head is mounted on the blocks. This



28.424

Figure 11-82.—Rack milling attachment.

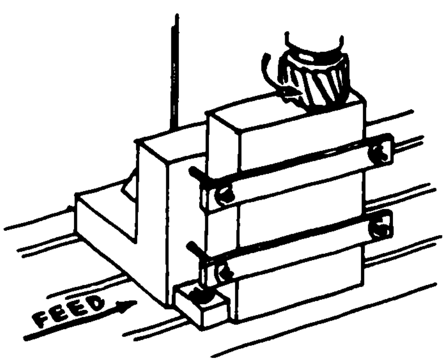


Figure 11-83.—Right-angle plate.

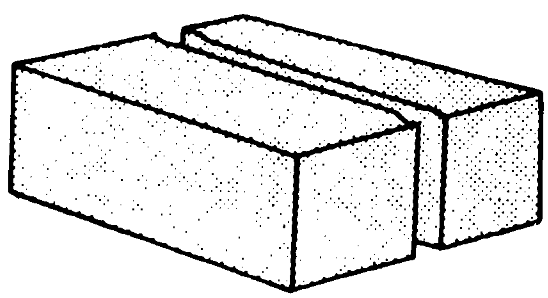


Figure 11-84.—Raising blocks.

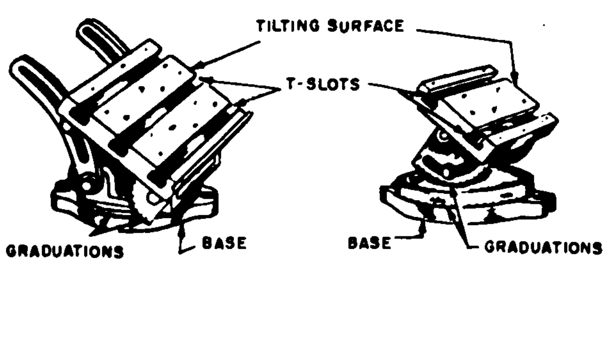
arrangement raises the index head and makes it possible to swing the head through a greater range to mill larger work.

TOOLMAKER'S KNEE

The toolmaker's knee (fig. 11-85) is a simple but useful attachment for setting up angular work, not only for milling but also for shaper, drill press, and grinder operations. You mount a toolmaker's knee, which may have either a stationary or rotatable base, to the table of the milling machine. The base of the rotatable type is graduated in degrees. This feature enables you to machine compound angles. The toolmaker's knee has a tilting surface with either a built-in protractor head graduated in degrees for setting the table or a vernier scale for more accurate settings.

FEEDS, SPEEDS, AND COOLANTS

Milling machines usually have a spindle speed range from 25 to 2,000 rpm and a feed range from 1/4 inch to 30 inches per minute (ipm). The feed is independent of the spindle speed; thus, a workpiece can be fed at any rate available in the feed range regardless of the spindle speed being used. Some of the factors concerning the selection of appropriate feeds and speeds for milling are discussed in the following paragraphs.



28.425

Figure 11-85.—Toolmaker's knees.

SPEEDS

Heat generated by friction between the cutter and the work may be regulated by the use of proper speed, feed, and cutting coolant. Regulation of this heat is very important because the cutter will be dulled or even made useless by overheating. It is almost impossible to set down any fixed rules that will govern cutting speeds because of varying conditions from job to job. Generally speaking, you should select a cutting speed that will give the best compromise between maximum production and longest life of the cutter. In any particular operation, consider the following factors in determining the proper cutting speed.

- Hardness of the Material Being Cut: The harder and tougher the metal being cut, the slower should be the cutting speed.

- Depth of Cut and Desired Finish: The amount of friction heat produced is directly proportional to the amount of material being removed. Finishing cuts, therefore, often may be made at a speed 40% to 80% higher than that used in roughing.

- Cutter Material: High-speed steel cutters may be operated from 50% to 100% faster than carbon steel cutters because high-speed steel cutters have better heat resistant properties than carbon steel cutters.

- Type of Cutter Teeth: Cutters that have undercut teeth cut more freely than those that have a radial face; therfore, cutters with undercut teeth may run at higher speeds.

- Sharpness of the Cutter: A sharp cutter may be run at much higher speed than a dull cutter.

- Use of Coolant: Sufficient coolant will usually cool the cutter so that it will not overheat even at relatively high speeds.

The approximate values given in table 11-5 may be used as a guide when you are selecting the proper cutting speed. If you find that the machine, the cutter, or the work cannot be suitably operated at the suggested speed, make immediate readjustment.

By referring to table 11-6, you can determine the cutter revolutions per minute for cutters varying in diameter from 1/4 inch to 5 inches. For example: You are cutting with a 7/16-inch cutter. If a surface speed of 160 feet per minute is required, the cutter revolutions per minute will be 1,398.

If the cutter diameter you are using is now shown in table 11-6, determine the proper revolutions per minute of the cutter by the formula:

(a) $$\text{rpm} = \frac{\text{Cutting speed X 12}}{\text{3.1416 X Diameter}}$$

or $$\text{rpm} = \frac{\text{fpm}}{\text{0.2618 X D}}$$

where

rpm = revolutions per minute of the cutter

fpm = required surface speed in feed per minute

D = diameter of cutter in inches

$$0.2618 = \text{constant} = \frac{\pi}{12}$$

EXAMPLE: What is the spindle speed for a 1/2-inch cutter running at 45 fpm?

$$\text{rpm} = \frac{45}{0.2618 \text{ X } 0.5}$$

$$\text{rpm} = 343.7$$

To determine cutting speed when spindle speed and cutter diameter are known, use the following formula:

$$\text{fpm X 12} = \text{rpm X 3.1416 X D}$$

$$\text{fpm} = \frac{\text{3.1416 X Diameter X rpm}}{12}$$

$$\text{fpm} = \text{0.2618 X D X rpm}$$

Table 11-5.—Surface Cutting Speeds

	Carbon steel cutters (ft. per min.)		High Speed steel cutters (ft. per min.)	
	Rough	Finish	Rough	Finish
Cast iron:				
Malleable	60	75	90	100
Hard castings	10	12	15	20
Annealed tool steel	25	35	40	50
Low carbon steel	40	50	60	70
Brass	75	95	110	150
Aluminum	460	550	700	900

28.280A

EXAMPLE: What is the cutting speed of a 2 1/4-inch end mill running at 204 rpm?

$$\text{fpm} = \text{0.2618 X D X rpm}$$

$$\text{rpm} = \text{0.2618 X 2.25 X 204}$$

$$\text{fpm} = 120.1$$

FEEDS

The rate of feed is the rate of speed at which the workpiece travels past the cut. When selecting the feed, you should consider the following factors.

- Forces are exerted against the work, the cutter, and their holding devices during the cutting process. The force exerted, varying directly with the amount of metal removed, can be regulated by the feed and depth of cut. The feed and depth of cut, therefore, seem to be interrelated, and in turn, are dependent upon

Table 11-6.—Cutter Speeds in Revolutions Per Minute

Diameter of cutter (in.)	Surface speed (ft. per min.)																
	25	30	35	40	50	55	60	70	75	80	90	100	120	140	160	180	200
	Cutter revolutions per minute																
1/4	382	458	535	611	764	851	917	1,070	1,147	1,222	1,376	1,528	1,834	2,139	2,445	2,750	3,056
5/16	306	367	428	489	611	672	733	856	917	978	1,100	1,222	1,466	1,711	1,955	2,200	2,444
3/8	255	306	357	408	509	560	611	713	764	815	916	1,018	1,222	1,425	1,629	1,832	2,036
7/16	218	262	306	349	437	481	524	611	656	699	786	874	1,049	1,224	1,398	1,573	1,748
1/2	191	229	268	306	382	420	459	535	573	611	688	764	917	1,070	1,222	1,375	1,528
5/8	153	184	214	245	306	337	367	428	459	489	552	612	736	857	979	1,102	1,224
3/4	127	153	178	203	254	279	306	357	381	408	458	508	610	711	813	914	1,016
7/8	109	131	153	175	219	241	262	306	329	349	392	438	526	613	701	788	876
1	95.5	115	134	153	191	210	229	267	287	306	344	382	458	535	611	688	764
1 1/4	76.3	91.8	107	123	153	168	183	214	230	245	274	306	367	428	490	551	612
1 1/2	63.7	76.3	89.2	102	127	140	153	178	191	204	230	254	305	356	406	457	508
1 3/4	54.5	65.5	76.4	87.3	109	120	131	153	164	175	196	218	262	305	349	392	436
2	47.8	57.3	66.9	76.4	95.5	105	115	134	143	153	172	191	229	267	306	344	382
2 1/2	38.2	45.8	53.5	61.2	76.3	84.2	91.7	107	114	122	138	153	184	213	245	275	306
3	31.8	38.2	44.6	51	63.7	69.9	76.4	89.1	95.3	102	114	127	152	178	208	228	254
3 1/2	27.3	32.7	38.2	44.6	54.5	60	65.5	76.4	81.8	87.4	98.1	109	131	153	174	196	213
4	23.9	28.7	33.4	38.2	47.8	52.6	57.3	66.9	71.7	76.4	86	95.6	115	134	153	172	191
5	19.1	22.9	26.7	30.6	38.2	42	45.9	53.5	57.3	61.1	68.8	76.4	91.7	107	122	138	153

28.281

the rigidity and power of the machine. Machines are limited by the power they can develop to turn the cutter and by the amount of vibration they can withstand when coarse feeds and deep cuts are being used.

- The feed and depth of cut also depend upon the type of cutter being used. For example, deep cuts or coarse feeds should not be attempted with a small diameter end mill; such an attempt would spring or break the cutter. Coarse cutters with strong cutting teeth can be fed at a relatively high rate of feed because the chips will be washed out easily by the cutting lubricant.

- Coarse feeds and deep cuts should not be used on a frail piece of work or on work mounted in such a way that the holding device will spring or bend.

- The desired degree of finish affects the amount of feed. When a fast feed is used, metal is removed rapidly and the finish will not be very smooth. However, a slow feed rate and a high cutter speed will produce a finer finish. For roughing, it is advisable to use a comparatively low speed and a coarse feed. More mistakes are made by overspeeding the cutter than by overfeeding the work. Overspeeding may be detected by a squeaking, scraping sound. If chattering occurs in the milling machine during the cutting process, reduce the speed and increase the feed. Excessive cutter clearance, poorly supported work, or a badly worn machine gear are also common causes of chattering.

One procedure for selecting appropriate feed for a milling operation is to consider the chip load of each cutter tooth. The chip load is the thickness of the chip that a single tooth removes from the work as it passes over the surface. For example, with a cutter turning at 60 rpm, having 12 cutting teeth, and a feed rate of 1 ipm, the chip load of a single tooth of the cutter will be 0.0014 inch. A cutter speed increase to 120 rpm reduces the chip load to 0.0007 inch; a feed increase to 2 ipm increases chip load to 0.0028 inch. The formula for calculating chip load is:

$$\text{Chip load} = \frac{\text{feed rate (ipm)}}{\text{cutter speed (rpm)} \times \text{number of teeth in cutter}}$$

Table 11-7 provides recommended chip loads for milling various materials with various types of cutters.

COOLANTS

The purpose of a cutting coolant is to reduce frictional heat and thereby extend the life of the cutter's edge. Coolant also lubricates the cutter face and flushes away the chips, reducing the possibility of damage to the finish.

If a commercial cutting coolant is not available, you can make a good substitute by thoroughly mixing 1 ounce of sal soda and 1 quart of lard oil in 1 gallon of water. Since the coolant tank holds 4 or 5 gallons, increase the ingredients proportionally. This emulsion is suitable for machining most metals.

In machining aluminum, you should use kerosene as a cutting coolant. Machine cast iron dry, although you can use a blast of compressed air to cool the work and the cutter. If you use compressed air, be extremely careful to prevent possible injury to personnel and machinery.

When using a periphery milling cutter, apply the coolant to the point at which the tooth leaves the work. This will allow the tooth to cool before you begin the next cut. Allow the coolant to follow freely on the work and cutter.

HORIZONTAL BORING MILL

The horizontal boring mill is used for many kinds of shopwork, such as facing, boring, drilling, and milling. In horizontal boring mill work, the setup of the work, as well as the setting of the tools, is similar to that found in lathe and milling machine work; therefore, a detailed discussion of these operations will not be necessary in this section.

Table 11-7.–Recommended Chip Loads

Material	Face Mills	Helical Mills	Slotting & Side Mills	End Mills	Form Relieved Cutters	Circular Saws
Plastic	.013	.010	.008	.007	.004	.003
Magnesium and alloys	.022	.018	.013	.011	.007	.005
Aluminum and alloys	.022	.018	.013	.011	.007	.005
Free cutting brasses & bronzes	.022	.018	.013	.011	.007	.005
Medium brasses & bronzes	.014	.011	.008	.007	.004	.003
Hard brasses & bronzes	.009	.007	.006	.005	.003	.002
Copper	.013	.010	.007	.006	.004	.003
Cast iron, soft (150-180 BH)*	.016	.013	.009	.008	.005	.004
Cast iron, med. (180-220 BH)	.013	.010	.007	.007	.004	.003
Cast iron, hard (220-300 BH)	.011	.008	.006	.006	.003	.003
Malleable iron	.012	.010	.007	.006	.004	.003
Cast steel	.012	.010	.007	.006	.004	.003
Low carbon steel, free mach.	.012	.010	.007	.006	.004	.003
Low carbon steel	.010	.008	.006	.005	.003	.003
Medium carbon steel	.010	.008	.006	.005	.003	.003
Alloy steel, annealed (180-220 BH)	.008	.007	.005	.004	.003	.002
Alloy steel, tough (220-300 BH)	.006	.005	.004	.003	.002	.002
Alloy steel, hard (300-400 BH)	.004	.003	.003	.002	.002	.001
Stainless steel, free mach.	.010	[illegible]	.006	.005	.003	.002
Stainless steels	.006	.005	.004	.003	.002	.002
Monel metals	.008	.007	.005	.004	.003	.002

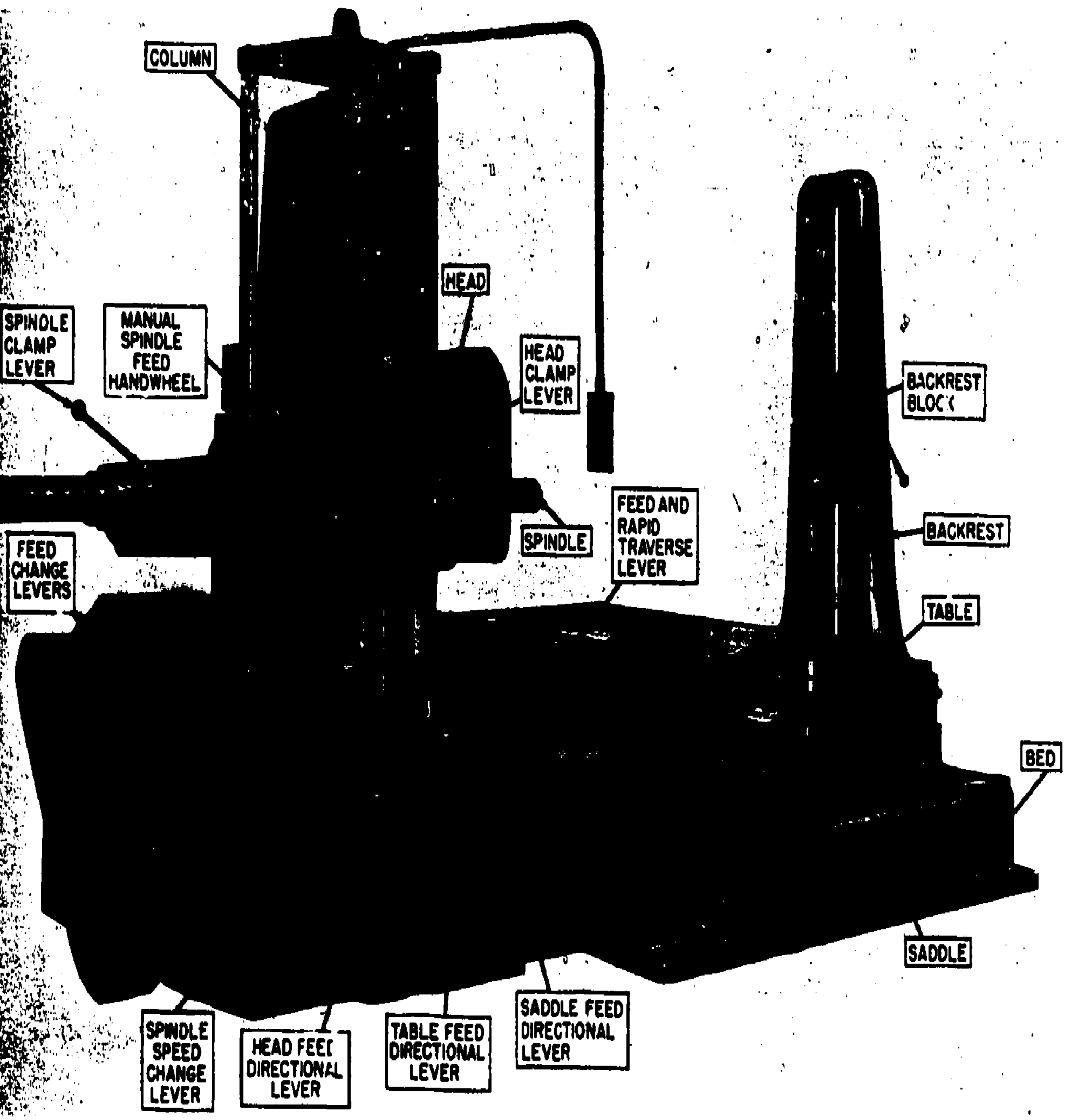
*(BH: Brinell Hardness Number)

28.282

The horizontal boring mill (fig. 11-86) consists of four major elements:

BASE AND COLUMN: The base contains all the drive mechanisms for the machine and provides a platform which has precision ways machined lengthwise for the saddle. The column provides support for the head and has two rails machined the height of the column for full vertical travel of the head.

HEAD: The head contains the horizontal spindle, auxiliary spindle, and the mechanism for controlling it. The head also provides a station for mounting various attachments. The spindle feed and spindle hand feed controls are



28.426

Figure 11-86.—Horizontal boring mill.

contained in the head, along with the quick traverse turnstile and spindle feed engagement lever.

SADDLE AND TABLE: A large rectangular slotted table is mounted on a saddle which can be traversed the length of the ways. The T-slots are machined the entire length of the table for holding down work and various attachments, such as rotary table angle plates, etc.

BACKREST OR END SUPPORT: The backrest is mounted on the back end of the ways. It is used to support arbors and boring bars as they rotate and travel lengthwise through the work, such as in-line boring of a pump casing or large bearing. The back rest blocks have an antifriction bearing which the boring bar passes through and rotates within. The back rest blocks travel vertically in unison with the head.

The two types of horizontal boring mill usually found in Navy machine shops and shore repair activities are the table type, used for small work, and the floor type, used for large work. The floor type is the most common of the two types found in shops. You will find this machine well-suited for repair work where machining of large, irregular jobs is commonplace.

The reference to size of horizontal boring mills differs with the manufacturer. Some use spindle size. For example, Giddings and Lewis model 300T has a 3-inch spindle. Other manufacturers refer to the largest size boring bar the machine will accept. In planning a job both of these factors should be considered along with the table size and the height that the spindle can be raised. Always refer to the technical manual for your machine.

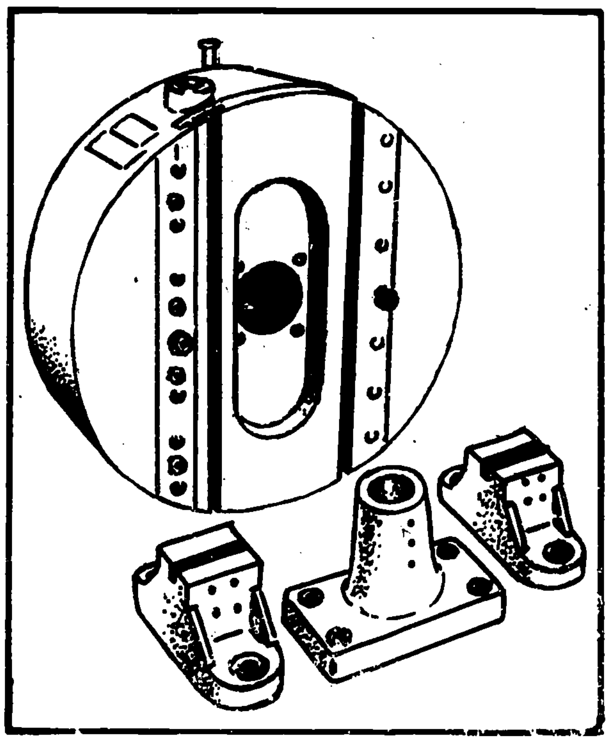
Setting up the work correctly is most important. Failure to set the work up properly can prove costly in man-hours and material. Oftentimes you will find that it is not advisable to set up a casting to a rough surface and that it will be preferable to set it up to the layout lines, since these lines will always be used as reference.

It is important that holding clamps used to secure a piece of work be tight. If braces are used, they should be placed in such a manner that they cannot come loose. Blocks, stops, and shims should all be fastened securely. If a workpiece is not properly secured, there is always the possibility of ruining the material or the machine and the risk of causing injury to the machine shop personnel.

Different jobs to be done on the boring mill may require different types of attachments. Such attachments include angular milling heads, combination boring and facing heads, thread lead arrangements, etc. Boring heads are available in a variety of diameters. These boring heads prove particularly useful in boring large diameter holes and facing large castings. Locally made collars may be used also. Stub arbors are used to increase desired diameters.

COMBINATION BORING AND FACING HEAD

The boring and facing head (fig. 11-87) is used for facing and boring large diameters. This



126.32X

Figure 11-87.—Combination boring and facing head.

attachment is mounted and bolted directly to the spindle sleeve and has a slide with automatic feed which holds the boring or facing tools. (This attachment can be fed automatically or positioned manually.) Although there are various sizes, each is made and used similarly. The heads are balanced to permit high-speed operation with the tool slide centered. When using tools off center, you must be careful to counterbalance the head, or use it at lower speeds.

Generally, the boring and facing head will come equipped with several toolholders for single point tools, right angle arm, boring bar, and boring bar holder which mounts on the slide.

To set up and operate the boring and facing head

1. Retract the spindle of the machine into the sleeve. Engage the spindle ram clamp lever.
2. Disengage the overrunning spindle feed clutch to prevent inadvertent engagement of the spindle power feed while the combination head is mounted on the machine. (If the slide is centered and locked, the spindle may be run through it for use in other operations without removing the attachment, but the spindle overrunning clutch should be disengaged again before use of the slide is resumed).
3. Set the spindle for the speed to be used.
4. Before shifting the spindle back-gear to neutral or making any spindle back-gear change when the combination head is mounted on the sleeve, rotate the sleeve by jogging it until the heavy end of the head is down. This is a safety precaution to prevent injury to the operator or damage to the work. Any spindle back-gear change requires a momentary shift to neutral, allowing free turning of the sleeve. The sleeve may then unexpectedly rotate until the heavy end of the facing head is down contacting the operator or the work.
5. Lift the head into position on the machine at the sleeve by inserting an eyebolt into the tapped hole in the periphery of the head.
6. To line up the bolt holes in the sleeve with those in the head, jog the spindle into position.
7. After you have tightened the mounting bolts, rotate the feed adjusting arm on the backing plate until the arm points directly toward the front.
8. Mount the restraining block on the head.
9. Set the slide manually by inserting the tee-handled wrench into the slot in the slide adjusting dial and turning the wrench until the slide is positioned. The dial is graduated in thousandths of an inch with one complete turn equaling a 0.125-inch movement of the slide.

After the slide is clamped in place, a spring-loaded safety clutch prevents movement of the slide or damage to the feed mechanism if the feed is inadvertently engaged. You must remember that this is not provided to allow continuous operation of the head when the slide is clamped and the feed is engaged. It is a jamming protection only. A distinct and continuous ratcheting of the safety clutch warns you to unlock the slide or disengage the feed. Do not confuse this warning with the intermittent ratcheting of the feed driving clutches as the head rotates. The same safety clutch stops the feed at the end of travel of the slide, thus preventing jamming of the slide or mechanism through overtravel.

The slide directional lever is located on the backing plate beneath the feed adjusting arm. The arrows on the face of the selector indicate which way it should be turned for feeding the slide in either direction. There are also two positions of the selector for disengaging the slide feed. The direction of the spindle rotation has no effect on the direction of the slide feed.

The slide feed rate adjusting arm scale is graduated in 0.010-inch increments from 0.000 to 0.050 inch, except that the first two increments are each 0.005 inch. Set the feed rate by turning the knurled adjusting arm to the desired feed in thousandths per revolution.

When mounting the single point toolholders, be sure the tool point is on center or slightly below center so the cutting edge has proper clearance at the small diameters. The feed mechanism may be damaged if the head is operated with the tool above center.

After you mount the facing head, perform the machining operation using the instructions found in the operator's manual for your boring machine.

RIGHT ANGLE MILLING ATTACHMENT

The right angle milling attachment is mounted over the spindle sleeve and is bolted directly to the face of the head. It is driven by a drive dog inserted between the attachment and the spindle sleeve. This attachment lets you perform milling operations at any angle setting through a full 360°. You can perform boring operations at right angles to the spindle axis using either the head or the table feed depending on the position of the hole to be bored. Standard milling machine tooling may be used in the spindle, being held in by a drawbolt extending through the spindle. A right angle milling attachment is shown in figure 11-88.

126.33X
Figure 11-88.—Angular milling head.

BORING MILL OPERATIONS

You should be able to perform drilling, reaming, and boring operations. In addition, you may be required to use a boring mill to face valve flanges, bore split bearings, and bore pump cylindrical liners.

Drilling, Reaming, and Boring

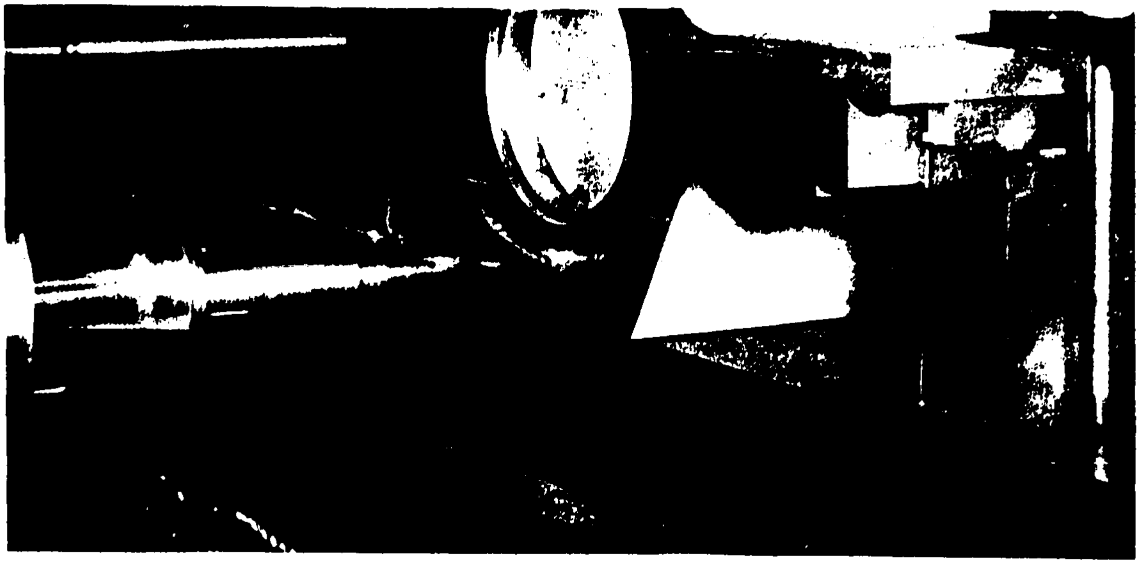
Drilling and reaming operations are performed in the horizontal boring mill in the same manner in which they are performed in a radial drill. The only major difference in the two setups is in the way the tools are held in the machines. In the horizontal boring mill the tool is held in the horizontal position (fig. 11-89) and in the radial drill the tool is held in the vertical position.

In Line Boring

To set the horizontal boring machine for a line boring operation, insert a boring bar into the spindle and pass it through the work. The boring bar is supported on the foot end by the back rest assembly. Depending on the size of the bore required, you can use either standard or locally manufactured tooling. The head provides the rotary motion for the tools mounted in the boring bar. Align the work with the axis of the boring bar, and bolt and/or clamp it to the table. The cutting operation is usually performed by having the spindle move relative to the work, which is held stationary. The machine is also designed to hold the bar in a fixed position and move the table lengthwise to complete the operation. (See fig. 11-90.)

The table can be power driven to provide travel perpendicular to the spindle, making it possible to bore, elongated and slotted when used in conjunction with vertical movement of the head.

Some boring mills have a single spindle in the head while others have a secondary or auxiliary spindle which can be fitted with a precision head and used in some boring operations. This secondary spindle may also be used on light work such as drilling accurately spaced small holes.

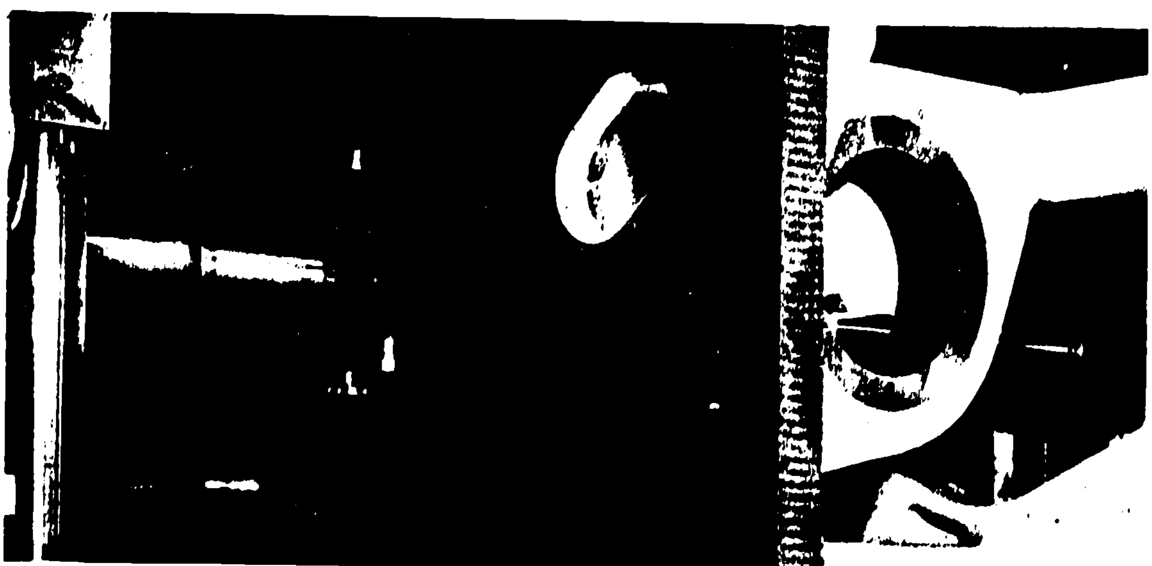


126.30X

Figure 11-89.—Drilling in the horizontal boring mill.

Reconditioning Split-Sleeve Bearings

Practically all of the high-speed bearings the Navy uses on turbines are the babbitt-lined split-sleeve type. Once a bearing of this type has wiped, it must be reconditioned at the first opportunity. If it has wiped only slightly, it can probably be scraped to a good bearing surface and restored to service. If badly wiped, it will have to be rebabbitted and rebored, or possibly replaced.



28.280

Figure 11-90.—Boring bar driven by the spindle and supported in the backrest block.

When a wiped bearing is sent to the machine shop for repair the following procedure should be followed as closely as possible.

1. Check for extent of damage and wear marks.
2. Take photos of bearing to indicate actual condition of bearing and for future reference in machining steps and assembly.
3. Check shell halves for markings. A letter or number should be on each half for proper identification and assembly. (If not marked, you should do so before disassembly.)
4. Inspect outer shell for burrs, worn ends and condition of alignment pins and holes.
5. Check blueprint and job order to ensure information required is available to you.
6. Ensure that actual shaft size has not been modified from blueprint.

When you must reline and rebore babbitt-lined bearings, remove all of the old lining by machining down to the base metal of the shell. Then clean the bearing shell with special cleaning solutions and rebabbitt them after plugging all oil holes with suitable material.

After relining the shell, remove the excess babbitt extending above the horizontal flanges by rough machining on a shaper. Take extreme care to see that the base metal of the horizontal flanges is not damaged during this machining operation. After rough machining, blue the remaining excess babbitt and scrape it until no more excess babbitt extends above the horizontal flanges.

Next, assemble the two half-shells and set them up on the horizontal boring mill. Check the spherical diameter of the bearing to ensure that it is not distorted beyond blueprint specifications in accordance with NAVSHIPS 9411.8.13.2. Generally, the words "BORE TRUE TO THIS SURFACE" are inscribed on the front face of the bearing shell. When dialing in the bearing, be sure to dial in on this surface.

Once the bearing is properly aligned in the boring mill, you can complete practically all the other operations without changing the setup. Bore the bearing to the finished diameter and machine the oil grooves as per blueprint specifications.

Distribution of oil is brought about by the oil grooves. These grooves may be of several forms; the two simplest are the axial and the circumferential grooves. Sometimes circumferential grooves are placed at the ends of the bearings as a controlling device for preventing side leakage, but this type of grooving does not affect the distribution of lubricant.

When grooves are machined into a bearing, you must be careful in beveling the groove out into the bearing leads to prevent the oil from constricting. The type of grooves used in a bearing should not be changed from the original design, unless warranted by continuous trouble traceable to improper lubricant distribution within the bearing.

Upon completion of all machining operations, it is the responsibility of both the repair activity and the ship's force to determine that the bearing is in accordance with the blueprint specifications and that a good bond exists between the shell and the babbitt metal.

Threading

Threads may be cut using the horizontal boring mill on machines that are equipped with a thread lead arrangement. On some machines, a thread lead arrangement with as many as 23 different threads, both standard and metric, is available.

T cut threads with these machines, use a system of change gear combinations to obtain the different leads. Secure a single point tool in a suitable toolholder and mount the toolholder in the spindle of the machine. While you cut threads, keep the spindle locked in place. The saddle, carrying the workpiece, advances at a rate determined by the change gear combination. Feeding, in conjunction with the spindle rotation in the low back gear range, produces the threads.

Cut the thread a little at a time in successive passes. The thread profile depends on how the cutting tool is ground. When you have completed the first pass, back the cutting tool off a few thousandths of an inch to avoid touching the workpiece on the return movement. Then reverse the spindle driving motor. This causes the saddle direction to reverse while the direction selection lever position remains unchanged. Allow the machine to run in this direciton until the cutting tool has returned to its starting point. Advance the cutter to take out a little more stock, and after setting the spindle motor to run in forward, make another cutting pass. Follow this procedure until the thread is completed. A boring bar with a microadjustable tool bit or a small precision head is ideal for this operation. It allows fast, easy adjustment of the tool depth, plus accuracy and control of the depth setting.

To set up for cutting threads, remove the thread lead access covers and set up the correct gear train combination as prescribed by the manufacturer's technical manual. After the gear train has been set up, lock the sliding arm by tightening the nuts on the arm clamp. Be sure to replace the retaining washers on all the studs and lock them with the screws provided with the machine. Refer to the manufacturer's technical manual for the machine you are using for correct gear arrangement.

Some of the gear combinations use only one gear on the B stud. When this occurs, take up the additional space on the stud by adding spacers to the stud. The following check-off list will be of assistance to you in threading in a horizontal boring mill:

1. Be sure the correct change gears are on the proper centers.

2. Position the head back-gear in the low range.

3. Place the feed change lever in the correct position to release the standard feed.

4. Engage the thread lead engaging lever.

5. Shift the driving gear lever to the thread lead position.

6. Start the spindle rotation forward.

7. Place the saddle directional lever in the left position. It will remain in this position until the thread is completed.

8. Place the feed/rapid traverse selector lever in the feed position. This will lock in the feed clutch until the complete threading operation is completed.

9. To disengage the feed, place the thread lead driving gear lever in standard position. The feed clutch will disengage. Do NOT do this during the threading operation or the thread lead timing will be lost.

MILLING MACHINE SAFETY PRECAUTIONS

Your first consideration as a Machinery Repairman should be your own safety. CARELESSNESS and IGNORANCE are the two great menaces to personal safety. Milling machines are not playthings and must be accorded the full respect that is due any machine tool. For your own safety observe the following precautions:

- NEVER attempt to operate a machine unless you are sure that you understand it thoroughly.

- Do NOT throw an operating lever without knowing in advance what is going to take place.

- Do NOT play with the control levers or idly turn the handles of a milling machine, even if it is stopped.

- NEVER lean against or rest your hands upon a moving table. If it is necessary to touch a moving part, know in advance the direction in which it is moving.

- Do NOT take a cut without making sure that the work is held securely in the vise or fixture and that the holding member is rigidly fastened to the machine table.

- Always remove chips with a brush or other suitable tool; NEVER use fingers or hands.

- Before attempting to operate any milling machine, study it thoroughly. Then if an emergency arises, you can stop the machine immediately. Knowing how to stop a machine is just as important, if not more important, as knowing how to start it.

- You must above all KEEP CLEAR OF THE CUTTERS. Do NOT touch a cutter, even when it is stationary, unless there is good reason to do so, and then be very careful.

The milling machine is not dangerous to operate, but if you do not follow certain safety practices you are likely to find it dangerous. There is always the danger of getting caught in the cutter. Never attempt to remove chips with the fingers at the point of contact of cutter and work. There is danger to the eyes from flying chips, and you should always protect your eyes with goggles and keep your eyes out of line of the cutting action.

CHAPTER 12

SHAPERS, PLANERS, AND ENGRAVERS

In this chapter we shall discuss the major types of shapers, planers, and pantographs (engraving), their individual components, cutters, and operating principles and procedures. A shaper has a reciprocating single-edged cutting tool which removes metal from the work as it is fed into the tool. A planer operates on a similar principle except that the work reciprocates, and the tool is fed into the work. A pantograph is used primarily for machine engraving letters and designs on any type of material. A pantograph can be used to engrave concave, convex, and spherical surfaces as well as flat surfaces.

SHAPERS

A shaper has a reciprocating ram that carries a cutting tool. The tool cuts on the forward stroke of the ram, which is slower than the return stroke. The work is held in a vise or on the worktable, which moves at right angles to the line of motion of the ram, permitting the cuts to progress across the surface being machined. Shapers are identified by the maximum size of a cube it can machine; thus, a 24-inch shaper will machine a 24-inch cube.

TYPES OF SHAPERS

There are three distinct types of shapers—crank, geared, and hydraulic. The type depends on how the ram receives motion to produce its own reciprocating motion. In a crank shaper the ram is moved by a rocker arm, which is driven by an adjustable crankpin secured to the main driving gear. Quick return of the ram is a feature of a crank shaper. In a geared shaper the ram is moved by a spur gear which meshes with a rack secured to the bottom of the ram. In a hydraulic shaper, the ram is moved by a hydraulic cylinder whose piston rod is attached to the bottom of the ram. Uniform tool pressure, smooth drive, and smooth work are features of the hydraulic type shaper.

There are many different makes of shapers, but the essential parts and controls are the same on all. When you learn how to operate one make of shaper, you should not have too much trouble in learning to operate another make. Figure 12-1 is an illustration of a crank shaper found in shops in some Navy ships.

SHAPER ASSEMBLIES

To perform the variety of jobs you will be required to do using the shaper, you must know the construction and operation of the main components which are the main frame assembly, drive assembly, crossrail assembly, toolhead assembly, and table feed mechanism. (See fig. 12-[illegible].)

Main Frame Assembly

The main frame assembly consists of the base and the column. The base houses the lubricating pump and sump which provide forced lubrication to the machine. The column contains the drive and feed actuating mechanisms. A dovetail slide is machined on top of the column to receive the ram. Vertical flat ways are machined on the front of the column to received the crossrail.

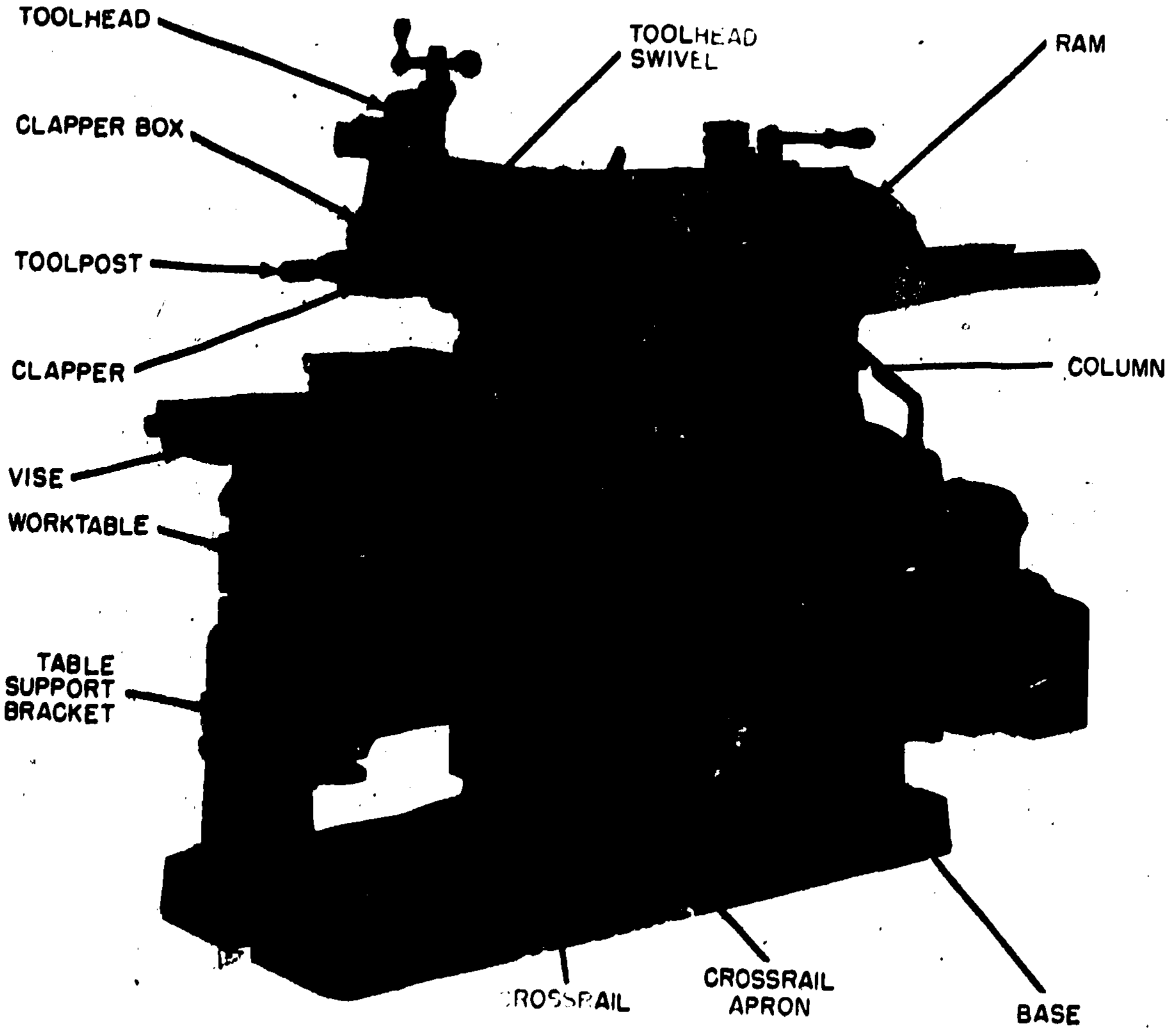
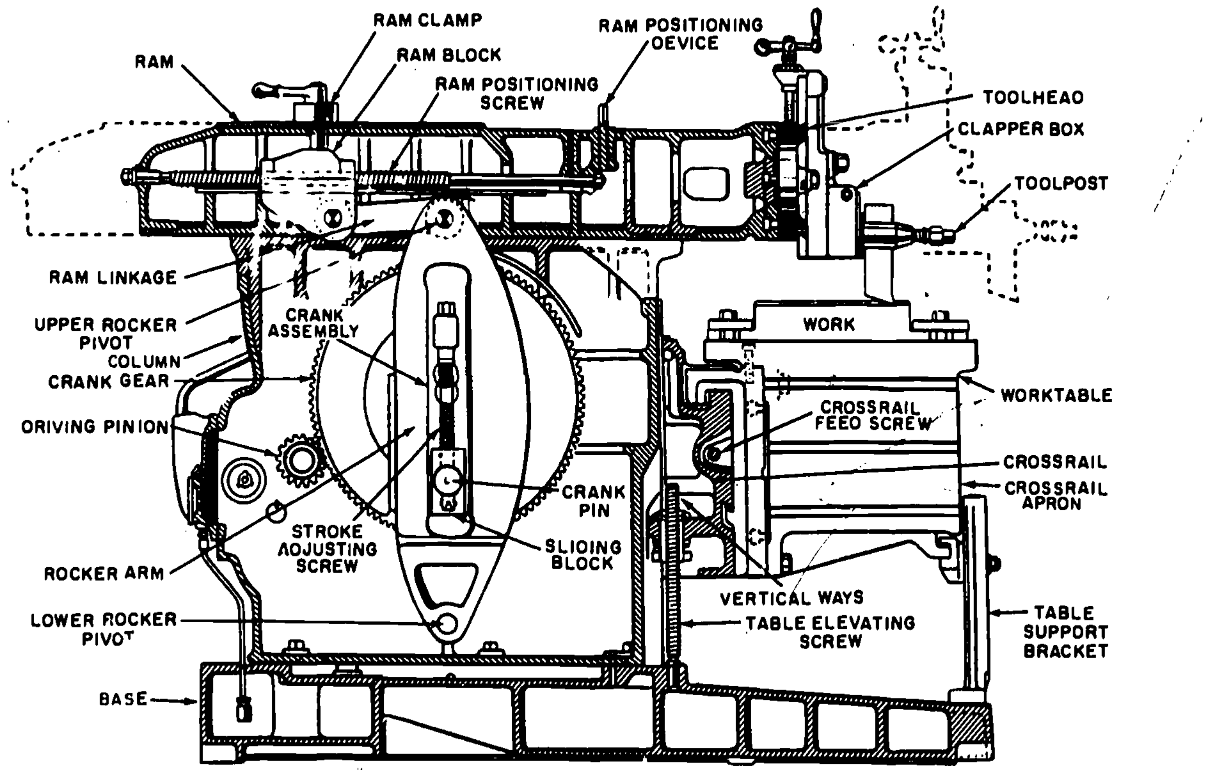


Figure 12-1.—Standard shaper. 28.219X

Drive Assembly

The drive assembly consists of the ram and the crank assembly. These parts convert the rotary motion of the drive pinion to the reciprocating motion of the ram. By using the adjustments provided, you can increase or decrease the length of stroke of the ram, and also position the ram so that the stroke is in the proper area in relation to the work.

You can adjust the CRANKPIN, which is mounted on the crank gear, from the center of the crank gear outward. The sliding block fits over the crankpin and is a free sliding fit in the rocker arm. If you center the crankpin (and therefore the sliding block) with the axis of the crank gear, the rocker arm will not move when the crank gear turns. But if you set the crankpin off center (by turning the stroke adjusting screw), any motion of the crank gear will cause rocking motion of the rocker arm. This motion is transferred to the ram through the ram linkage and starts the reciprocating motion of the ram. The distance that the crankpin is set off center determines the length of stroke.



28.220X

Figure 12-2.—Cross-sectional view of a crank type shaper.

To position the ram, turn the ram positioning screw until the ram is placed properly with respect to the work. Specific procedures for positioning the ram and setting the stroke are in the manufacturer's technical manual for the specific machine(s).

Crossrail Assembly

The crossrail assembly includes the crossrail, the crossfeed screw, the table, and the table support bracket (foot). (See fig. 12-1.) The crossrail slides on the vertical ways on the front of the shaper column. The crossrail apron (to which the worktable is secured) slides on horizontal ways on the crossrail. The crossfeed screw engages in a mating nut which is secured to the back of the apron. The screw can be turned either manually or by power to move the table horizontally

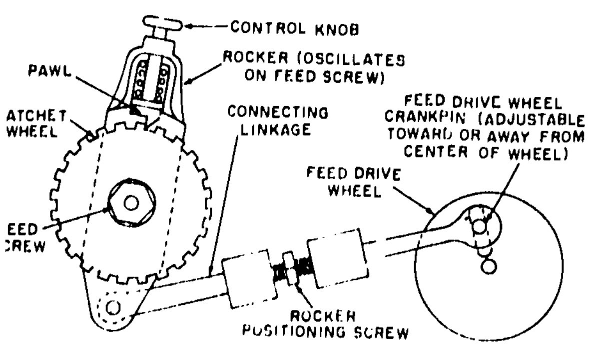
The worktable may be plain or universal as shown in figure 12-3. Some universal tables can be swiveled only right or left, away from the perpendicular; others may be tilted fore or aft at small angles to the ram. T-slots on the worktables are for mounting the work or work-holding devices. A table support bracket (foot) holds the worktable and can be adjusted to the height required. The bracket slides along a flat surface on the base as the table moves horizontally. The table can be adjusted vertically by the table elevating screw (fig. 12-2).

Table Feed Mechanism

The table feed mechanism (fig. 12-4) consists of a ratchet wheel and pawl, a rocker, and a feed drive wheel. The feed drive wheel (driven by the main crank), which operates similarly to the ram drive mechanism, converts rotary motion to reciprocating motion. As the

28.221X
Figure 12-3.—Swiveled and tilted table.

feed drive wheel rotates, the crankpin (which can be adjusted offcenter) causes the rocker to oscillate. The straight face fo the pawl engages a slot in the ratchet wheel, thus turning the ratchet wheel and the feed screw. The back face of the pawl is angular and therefore rides over



28.222
Figure 12-4.—Mechanical table feed mechanism.

the tooth as it is rocked in an opposite direction. To change the direction of feed, lift the pawl and rotate it one-half turn. To increase the rate of feed, increase the distance between the feed drive wheel crankpin and the center of the feed drive wheel.

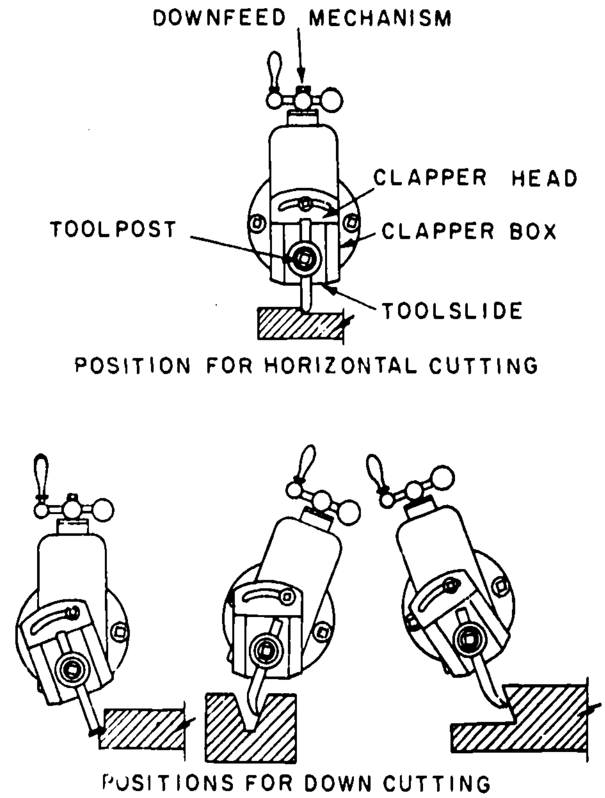
The ratchet wheel and pawl method of feeding crank type shapers has been used for many years. Relatively late model machines still use similar principles. As specific procedures for operating feed mechanisms may vary, you should consult manufacturers' technical manuals for explicit instructions.

Toolhead Assembly

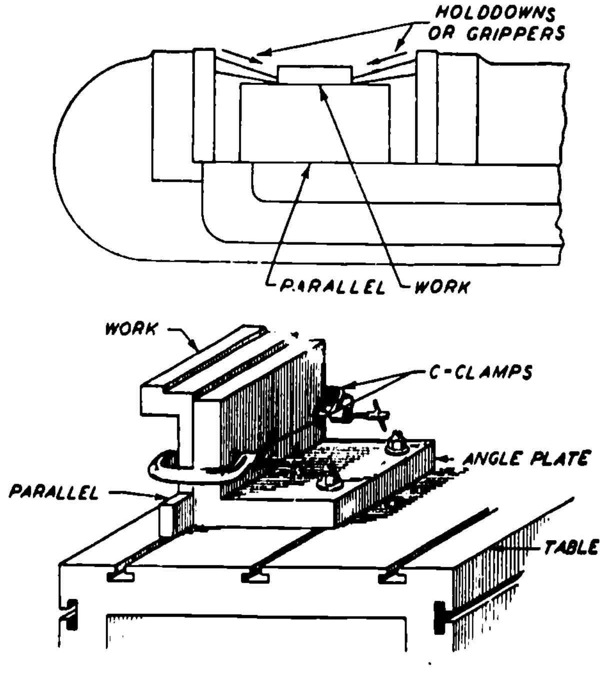
The toolhead assembly consists of the toolslide, the downfeed mechanism, the clapper box, the clapper head, and the toolpost at the forward end of the ram. The entire assembly can be swiveled and set at any angle not exceeding 50° on either side of the vertical. The toolhead is raised or lowered by hand feed to make vertical cuts on the work. In making vertical or angular cuts, the clapper box must be swiveled away from the surface to be machined (fig. 12-5); otherwise, the tool will dig into the work on the return stroke.

SHAPER VISE

The shaper vise is a sturdy mechanism secured to the table by T-bolts. The vise has two jaws, one stationary, the other movable, that can be drawn together by a screw. The vise differs from other vises in that the jaws are longer and deeper and will open to accommodate large work. Most such vises have hardened steel jaws ground in place. The universal vise may be swiveled in a horizontal plane from 0° to 180°. The usual positions are either to have the jaws set parallel with the stroke of the ram or to have them at right angles to the stroke. See that the vise is free from any obstruction which might keep the work from seating properly. Burrs and rough edges on the vise and chips left from previous machinings should be removed before starting to work.



28.223
Figure 12-5.—Toolhead assembly in various positions.



28.224X
Figure 12-6.—Methods of holding and clamping.

Work can be set on parallels so that the surface to be cut is above the top of the vise. Shaper holddowns can be used in holding the work between the jaws of the vise (fig. 12-6). Work larger than the vise will hold, can be clamped directly to the top or side of the machine table. When work too large or awkward for a swivel vise must be rotated, the work can be clamped to a rotary table. V-blocks, angle plates, and C-clamps are also used in mounting work on shaper tables.

TOOLHOLDERS

Various types of toolholders, made to hold interchangeable tool bits, are used to a great extent in planer and shaper work. Tool bits are available in different sizes and are hardened and cut to standard lengths to fit the toolholders.

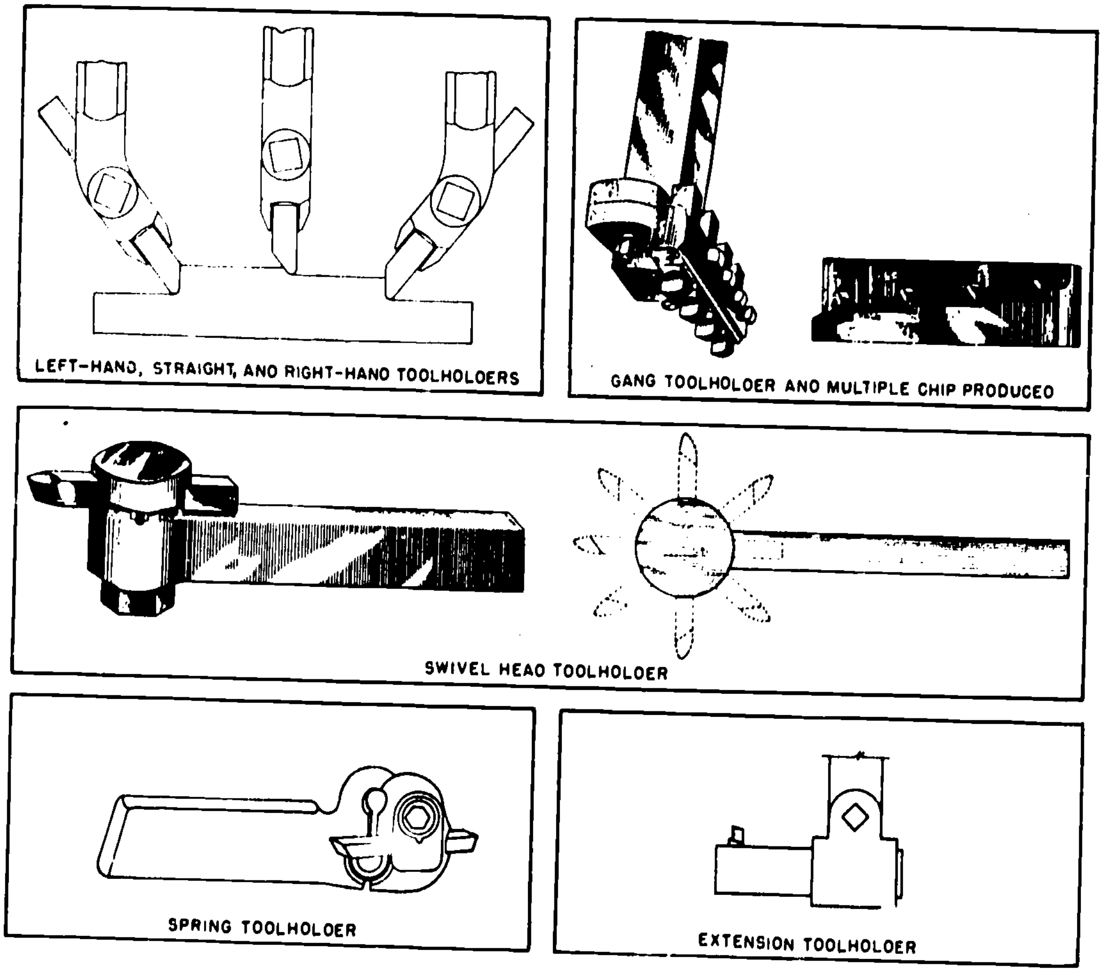
The toolholders that you will most commonly use are:

1. Right-hand, straight, and left-hand toolholders (fig. 12-7) which may be used for the majority of common shaper and planer operations.

2. Gang toolholders (fig. 12-7) are especially adapted for surfacing large castings. With this toolholder you make multiple cuts with each forward stroke of the shaper. In this manner each tool takes a light cut and there is less tendency to "break out" at the end of a cut.

3. Swivel head toolholders (fig. 12-7) are universal, patented holders that may be adjusted to place the tool in various radial positions. This feature allows the swivel head toolholder to be converted into a straight, right-hand, or left-hand holder at will.

4. Spring toolholders (fig. 12-7) have a rigid U-shaped spring which lets the holder cap absorb a considerable amount of vibration. This holder is particularly good for use with formed cutters



28.225

Figure 12-7.—Toolholders.

which have a tendency to chatter and dig into the work.

5. Extension toolholders (fig. 12-7) are adapted for cutting internal keyways, splines, and grooves on the shaper. The extension arm of the holder can be adjusted for length and radial position of the tool.

Procedures for grinding shaper and planer tool bits for various operations are discussed in Chapter 6 of this training manual.

SHAPER SAFETY PRECAUTIONS

The shaper, like all machines in the machine shop, is not a dangerous piece of equipment if good safety practices are observed. You should read and understand the safety precautions and operating instructions posted on or near the machine prior to operating the shaper. Some good safety practices are listed here but are intended only to supplement those posted on the machine.

- Always wear goggles or face shield.

- Ensure that the workpiece, vise, and setup fixture is properly secured.

- Ensure that the work area is clear of tools.

- Inform other personnel in the area to prevent possible injury to them from flying chips.

- Ensure that the travel of the ram is clear to both front and rear of the machine.

- Never stand in front of the shaper while it is in operation.

- Avoid touching the tool, clapper box, or workpiece while the machine is in operation.

- Never remove chips with your bare hand; always use a brush or piece of wood.

- Keep area around the machine clear of chips to help prevent anyone from slipping and falling into the machine.

- Remember: SAFETY FIRST, ACCURACY SECOND, and SPEED LAST.

SHAPER OPERATIONS

Before beginning any job on the shaper, you should thoroughly study and understand the blueprint or drawing from which you are to work. In addition, the following precautions should be taken:

- Make certain that the shaper is well oiled.

- Clean away ALL chips from previous work.

- Be sure that the cutting tool is set properly; otherwise the tool bit will chatter. Set the toolholder so that the tool bit does not extend more than about 2 inches below the clapper box.

- The piece of work should be held rigidly in the vise to prevent chatter. You can seat the work by tapping it with a babbitt hammer.

- Test the table to see if it is level and square. Make these tests with a dial indicator and machinist's square as shown in figure 12-8. If either the table or the vise is off parallel, check for dirt under the vise or improper adjustment of the table support bracket.

- Adjust the ram for length of stroke and position. The cutting tool should travel 1/8 to 1/4 inch past the edge of the work on the forward stroke and 3/4 to 7/8 inch behind the rear edge of the work on the return stroke.

Feeds and Speeds

The speed of the shaper is regulated by the gear shift lever. A speed indicator plate shows relative positions of the shift lever for a variety of speeds. The change gear box, located on the operator's side of the shaper, lets you change the speed of the ram and cutting tool according to the length of work and hardness of the metal. When the driving gear is at a constant speed, the ram will make the same number of strokes per minute regardless of whether the stroke is 4 inches or 12 inches. Therefore, to maintain the same cutting speed, the cutting tool must make three times as many strokes for the 4-inch cut as it does for the 12-inch cut.

Determining cutting speeds for the shaper is more difficult than determining speeds for the lathe, drill press, or milling machine. Because the motion is reciprocating, you must consider the number of strokes a minute, taking into account the time ratio between the cutting stroke and the return stroke. In most shapers it takes approximately 1 1/2 times as long to make the cutting stroke as it does to make the return stroke. This means that in any one cycle of ram action the cutting stroke consumes 3/5 of the time and the return stroke 2/5 of the time. On this basis, we can conclude that ram speed in feet per minute divided by 3/5 will give the speed of the cutting stroke in feet per minute.

The formula given below, based on information in the preceding paragraph, can be used for determining strokes per minute necessary to produce a particular cutting speed

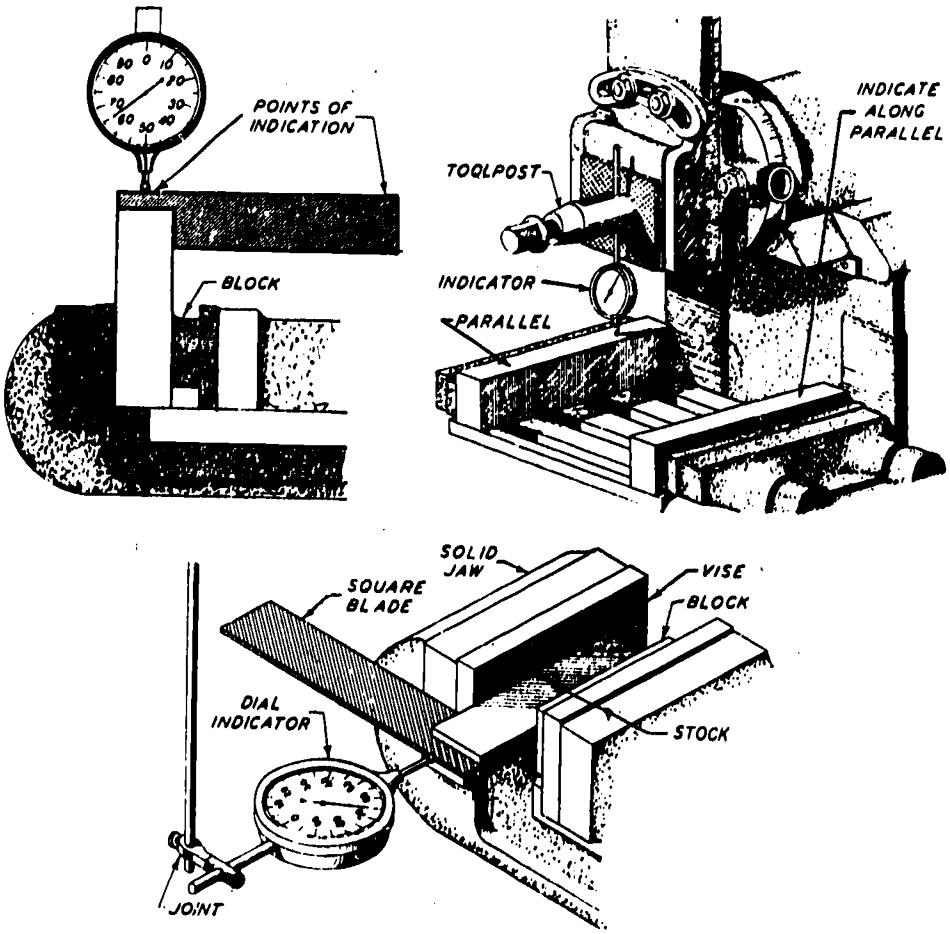


Figure 12-8.—Squaring the table and the vise. 28.226X

or for determining the cutting speed from strokes per minute:

CS = SPM x LOS x 0.14

Where: CS = cutting speed in feet per minute

SPM = strokes per minute
LOS = length of stroke in inches
0.14 = constant

In the formula for cutting speed, the constant converts the cutting time factor from a full time basis to the correct fractional basis and also converts inches to feet.

EXAMPLE: If the number of strokes per minute is 60 and the length of the stroke is 5 inches, what is the cutting speed?

SOLUTION:

SPM x LOS x 0.14 = CS

60 x 5 x 0.14 = 42 fpm

To find the number of strokes per minute when the desired cutting speed is known, simply transpose the formula:

$$SPM = \frac{CS}{0.14 \times LOS}$$

Table 12-1 gives recommended cutting speeds for various metals.

Varying rates of horizontal feed up to approximately 0.170 inch per stroke are available on most shapers. There are no hard and fast rules for selecting a specific feed rate in shaping. Consequently, when selecting feeds, you must rely on past experience and common sense. Generally, for making roughing cuts on rigidly held work, the feed should be as heavy as the machine will allow. For less rigid setups and for finishing, use light feeds and small depths of cut. The best procedure is to start with a relatively light feed and increase the feed until a desirable feed rate is reached.

Shaping a Rectangular Block

An accurately machined rectangular block has opposite surfaces that are parallel to each other, and the edges adjacent to any surface are square with that surface. In this discussion, face surfaces are the surfaces of the block having the largest surface area, the end edges are the surfaces which limit the length of the block; and the side edges are the surfaces limiting the width of the block.

The rectangular block can be machined in four setups when a shaper vise is used. One face and an end edge are machined in the first setup. The opposite face and opposite end edge are machined in the second setup. The side edges are machined in two similar but separate setups. The vise jaws are aligned at right angles to the ram. The ram stroke and position are set as required in relation to the width of the block; however, the stroke and position must be readjusted for machining the side edges.

To machine a rectangular block from a rough casting, proceed as follows:

1. Clamp the casting in the vise so that a face surface is horizontally level and slightly above the top of the vise jaws. Allow one end edge to extend out of the side of the vise jaws enough so that a cut can be taken on the end edge without unclamping the casting. Now feed the cutting tool down to the required depth and take a horizontal cut across the face surface. After the face is machined, readjust the cutting tool for down cutting and, using the downfeed mechanism, take a vertical cut on the end edge extending out of the vise jaws. Check the end edge for squareness with the face surface and adjust the toolhead swivel to offset inaccuracies.

2. To machine the second face surface and second end edge, turn the block over and set the previously machined face surface on parallels (similar to the method used in step 1). Insert

Table 12-1.–Recommended Cutting Speeds for Various Metals

Type of metal	Cutting speed (feet per minute)			
	Carbon steel tools		High-speed steel tools	
	Roughing	Finishing	Roughing	Finishing
Cast iron	30	20	60	40
Mild steel	25	40	50	80
Tool steel	20	30	40	60
Brass / Bronze	75	100	150	200
Aluminum	75	100	150	200

28.228.0

small strips of paper between each corner of the block and the parallels. This provides a means of ensuring that the block is firmly seated on the parallels. Clamp the block in the vise and use a soft-face mallet to tap the block down solidly on the parallels. When the block is held securely in the vise and the strips of paper indicate that the block is resting solidly on the parallels, machine the second face surface and second end edge to the correct thickness and length dimensions of the block.

3. To machine the side edge, open the vise jaws so that the jaws can be clamped on the end edges of the block. Now set the block on parallels in the vise with the side edge extending out of the jaws enough to permit a cut using the downfeed mechanism. Adjust the ram for length of stroke and for position to machine the side edge and make the cut.

4. The other side edge is set up and machined as described in step 3.

Shaping Angular Surfaces

Two methods are used for machining angular surfaces. For steep angles, such as on V-blocks, the work is mounted horizontally level and the toolhead is swiveled to the desired angle. For small angles of taper, such as on wedges, the work is mounted on the table at the desired angle from the horizontal, or the table may be tilted if the shaper is equipped with a universal table.

To machine a steep angle using the toolhead swiveled to the proper angle:

1. Set up the work as for machining a flat surface parallel with the table.

2. Swivel the toolhead (see fig. 12-5) to the required angle. (Swivel the clapper box in the opposite direction.)

3. Start the machine and, using the manual feed wheel on the toolhead, feed the tool down across the workpiece. The depth of each cut is regulated by the horizontal feed controls. (Because the tool is fed manually, be careful to feed the tool toward the work only during the return stroke.)

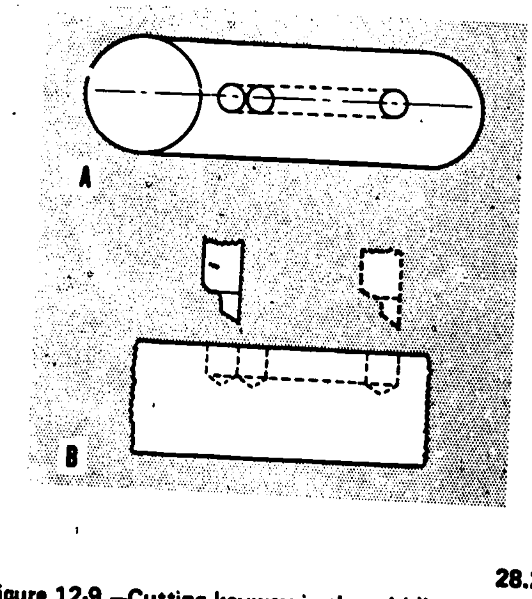
Shaping Keyways in Shafts

Occasionally, you may have to cut a keyway in a shaft by using the shaper. Normally, you will lay out the length and width of the keyway on the circumference of the shaft. A centerline laid out along the length of the shaft and across the end of the shaft will make the setup easier. As the shaper cutting tool cannot be used for making a cut all the way to the end of the keyway, holes (of the same diameter as the keyway width and slightly deeper than the keyway) must be drilled to provide tool clearance at the end of the cutting stroke. Figure 12-9 shows how the holes should be located for cutting a blind keyway (not ending at the end of a shaft). If the keyway extends to the end of the shaft, only one hole is necessary.

To cut a keyway in a shaft, proceed as follows:

1. Lay out the centerline, the keyway width, and the clearance hole centers as illustrated in part A of figure 12-9. Drill the clearance holes.

2. Position the shaft in the shaper vise or on the worktable so that it is parallel to the ram. Use a machinist's square to check the centerline



28.227
Figure 12-9.—Cutting keyway in the middle of a shaft.

on the end of the shaft to ensure that it is perpendicular to the surface of the worktable. This ensures that the keyway layout is exactly centered at the uppermost height of the shaft, thus preventing angular misalignment of the sides of the keyway.

3. Adjust the stroke and position of the ram, so that the forward stroke of the cutting tool ends at the center of the clearance hole. (If a blind keyway is being cut, ensure that the cutting tool has enough clearance at the end of the return stroke so that the tool will remain in the keyway slot.) (See B of fig. 12-9.)

4. Position the work under the cutting tool so that its center is aligned with the centerline of the keyway. (If the keyway is over 1/2 inch wide, cut a slot down the center and shave each side of the slot until the proper width is obtained.)

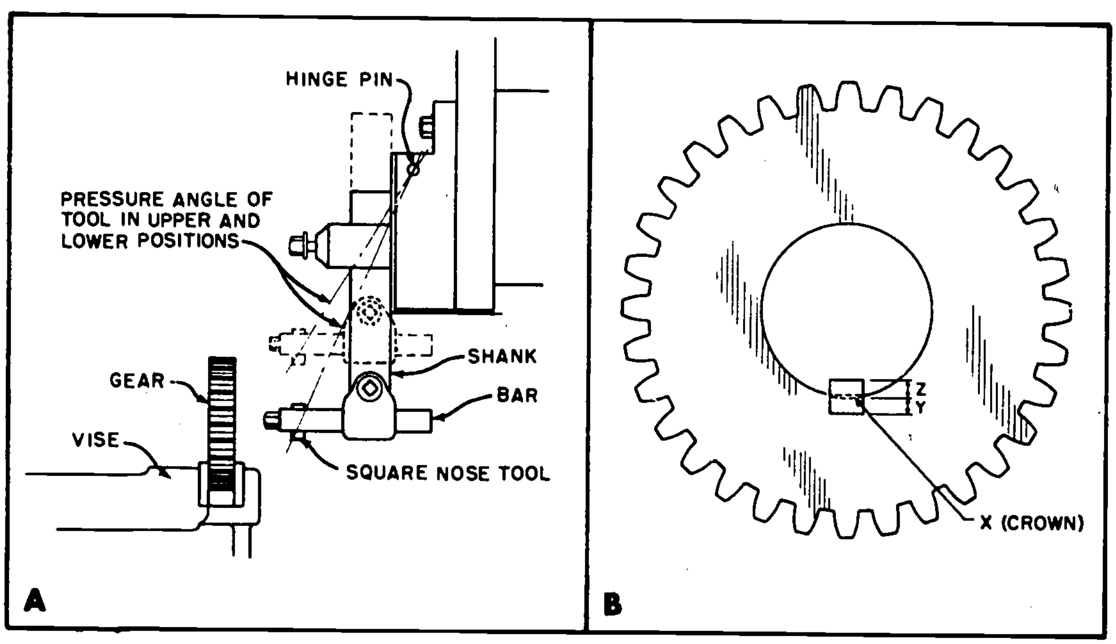
5. Start the shaper and, using the toolhead slide, feed the tool down to the depth required, as indicated by the graduated collar.

Shaping on Internal Keyway

When an internal keyway is to be cut in a gear, you will have to use extension tools. They lack the rigidity of external tools, and the cutting point will tend to spring away from the work unless you take steps to compensate for this condition. The keyway MUST be in line with the axis of the gear. Test the alignment with a dial indicator by taking a reading across the face of the gear; swivel the vise slightly, if necessary, to correct the alignment.

The bar of the square-nose toolholder should not extend any farther than necessary from the shank; otherwise the bar will have too much "spring" and will allow the tool to be forced out of the cut.

The extension toolholder should extend as far as practicable below the clapper block, rather than in the position shown by the dotted lines in part A of figure 12-10. The pressure angle resulting with the toolholder in the upper position may cause the pressure of the cut to open the clapper block slightly and so allow the tool to leave the cut. In the lower position, the



28.227A

Figure 12-10.—Internal keyway. A. Shaping internal keyway in a gear. B. Depth of keyways.

pressure angle is more nearly vertical and prevents the clapper block from opening. Another method for preventing the clapper block from opening is to mount the tool in an inverted position.

With the cutting tool set up as in part A of figure 12-10, center the tool with the layout lines in the usual manner, and make the cut to the proper depth while feeding the toolhead down by hand. With the setup in an inverted position, center the tool with the layout lines at the top of the hole, and make the cut by feeding the toolhead upward.

The relative depths to which external and internal keyways are cut to produce the greatest strength, are illustrated by part B of figure 12-10. In cutting a keyway in a shaft, the crown "X" is removed first, then the downfeed micrometer collar is set to zero and the depth of the keyway is gaged from this zero setting. This depth will be equal to the distance "Z." However, in cutting the mating keyway in the gear, the micrometer collar is set to zero at the point where the cutting tool first touches the edge of the hole. The distance "Y," to which the keyway is cut in the gear is then made equal to the depth "Z." The total distance of "X" plus "Y" plus "Z" is equal to the height of the key which is to lock the two parts together. (See fig. 12-10.)

Shaping Irregular Surfaces

Irregular surfaces can be machined by using form ground tools and by hand feeding the cutting tool vertically while power feed moves the work horizontally. An example of work that might be shaped using form tools is a gear rack. Irregularly shaped work such as concave and convex surfaces can be shaped using the toolhead feed. When irregular surfaces are being machined, close attention is required because the cutting tool is controlled manually. Also in this work the job should be laid out before machining to provide reference lines. Roughing cuts should be taken to remove excess material to within 1/16 inch of the layout lines.

You can cut RACK TEETH on a planer as well as on a shaper or milling machine. During the machining operation the work may be held in the vise or clamped directly to the worktable. After the work has been mounted and positioned, the tooth space is generally roughed out in the form of a plain rectangular groove with a roughing tool, then finished with a tool ground to tooth contour and size.

The operations required for machining a rack should be performed in the following sequence.

1. Clamp the work in the vise or to the table.

2. Position a squaring tool, which is narrower than the required tooth space, so that the tool is centered with the first tooth space to be cut.

3. Set the graduated dial on the crossfeed screw to zero, and use it as a guide for spacing of the teeth.

4. Move the toolslide down until the tool just touches the work and lock the graduated collar on the toolslide feed screw.

5. Start the machine and feed the toolslide down slightly less than the whole depth of the tooth, using the graduated collar as a guide, and rough out the first tooth space.

6. Raise the tool to clear the work and move the crossfeed a distance equal to the linear pitch of the rack tooth by turning the crossfeed lever. Rough out the second tooth space and repeat this operation until all spaces are roughed out.

7. Replace the roughing tool with a tool ground to size for the tooth form desired, and align the tool.

8. Adjust the work so that the tool is in proper alignment with the first tooth space that has been rough cut.

9. Set the graduated dial on the crossfeed screw at zero and use it as a guide for spacing the teeth.

10. Move the toolslide down until the tool just touches the work and lock the graduated collar on the toolslide feed screw.

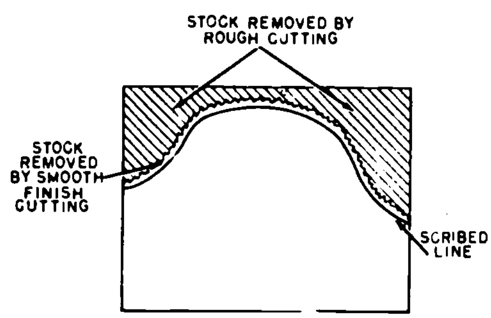
11. Feed the toolslide down the whole depth of the tooth, using the graduated collar as a guide, and finish the first tooth space.

12. Raise the tool to clear the work and move the crossfeed a distance equal to the linear pitch of the rack tooth by turning the crossfeed lever.

13. Finish the second tooth space, then measure the thickness of the tooth with the gear tooth vernier caliper. Adjust the toolslide to compensate for any variation indicated by this measurement.

14. Repeat the process of indexing and cutting until all teeth have been finished.

Irregular surfaces commonly machined on the shaper are CONVEX and CONCAVE radii. On one end of the work, lay out the contour of the finished job. When shaping to a scribed line, as illustrated in figure 12-11, it is good practice to rough cut to within 1/16 inch of the line. You can do this by making a series of horizontal cuts using automatic feed and removing excess stock. Use a left-hand cutting tool to remove stock on the right side of the work and a right-hand cutting tool to remove stock on the left side of the work. When 1/16 inch of metal remains above the scribed line, take a file and bevel the edge to the line. This will eliminate tearing the line by the breaking of the chip. Starting at the right-hand side of the work, set the automatic feed so that the horizontal travel is rather slow and, feeding the tool vertically by hand, take the finishing cuts to produce a smooth contoured surface.

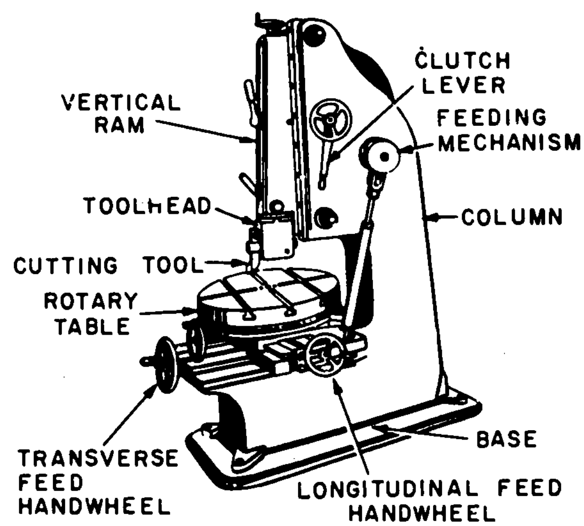


28.229

Figure 12-11.—Shaping irregular surfaces.

VERTICAL SHAPERS

The vertical shaper (slotter) shown in figure 12-12 is especially adapted for slotting internal holes or keyways with angles up to 10°. Angular slotting is accomplished by tilting the vertical ram (fig. 12-12B), which reciprocates up and down, to the required angle. The toolhead (fig. 12-12C) can be rotated to any angle. Although different models of machines will vary in the location of the control levers, all of them will have the same basic functions and capabilities. The speed of the ram is adjustable to allow for the various materials and machining requirements and is expressed in either strokes per minute or feet per minute, depending on the particular model. The length and the position of the ram stroke may also be adjusted. Automatic feed for the cross and longitudinal movements, and on some models the rotary movement, is provided by a ratchet mechanism, gear box, or variable speed hydraulic system, again, depending on the model. Work may be held in a vise mounted on the rotary table, clamped directly to the rotary or held by special fixtures. A valve handwheel is an example of work that can be done on a machine of this type. The



28.331

Figure 12-12.—Vertical shaper.

square hole in the center of the handwheel is cut on a slight angle to match the angle of the square on the valve stem. If this hole were cut by using a broach or an angular (square) hole drill, the square would wear prematurely due to the reduced area of contact resulting from a straight and an angular surface being mated together.

PLANERS

Planers are rigidly constructed machines, particularly suitable for machining large and heavy work where long cuts are required. In general, planers and shapers can be used for similar operations. However, reciprocating motion of planers is provided by the worktable (platen), while the cutting tool is caused to feed at right angles to this motion, thus allowing the cut to advance. Like the shaper, the planer cuts only on the forward stroke, after which the table is caused to make a quick return to bring the work into position for the next cut. Planer size is designated by the size of the largest work that can be clamped and machined on its table, thus a 30 inch by 30 inch by 6 foot planer is one that can accommodate work up to these dimensions.

TYPES OF PLANERS

Planers are divided into two general classes, the OPEN side type and the DOUBLE HOUSING type.

Planers of the open side type (fig. 12-13) have a single vertical housing to which the crossrail is attached. The advantage of this design lies in the fact that work may be planed that is too wide to pass between the uprights of a double housing machine.

In the double housing planer, the worktable moves between two vertical housings to which a crossrail and toolhead are attached. The larger machines are usually equipped with two cutting heads mounted to the crossrail as well as a side head mounted on each housing. With this setup, it is possible to simultaneously machine both the side and the top surfaces of work mounted on the table.

CONSTRUCTION AND MAINTENANCE

All planers consist of five principal parts: the bed, table, columns, crossrail, and the toolhead.

The bed is a heavy, rigid casting which supports the entire piece of machinery. On the upper surface of the bed are the ways upon which the planer table [illegible]

The table is a cast iron surface to which the work is mounted. The planer table has T-slots and reamed holes for fastening work to the table. On the underside of the table there is [illegible] a gear train or [illegible] mechanism which gives the table its reciprocating motion.

The columns of a double housing planer are attached to either side of the bed and at one end of the planer. On the open side planer there is only one column or housing attached on one side of the bed. The columns serve to support and carry the crossrail.

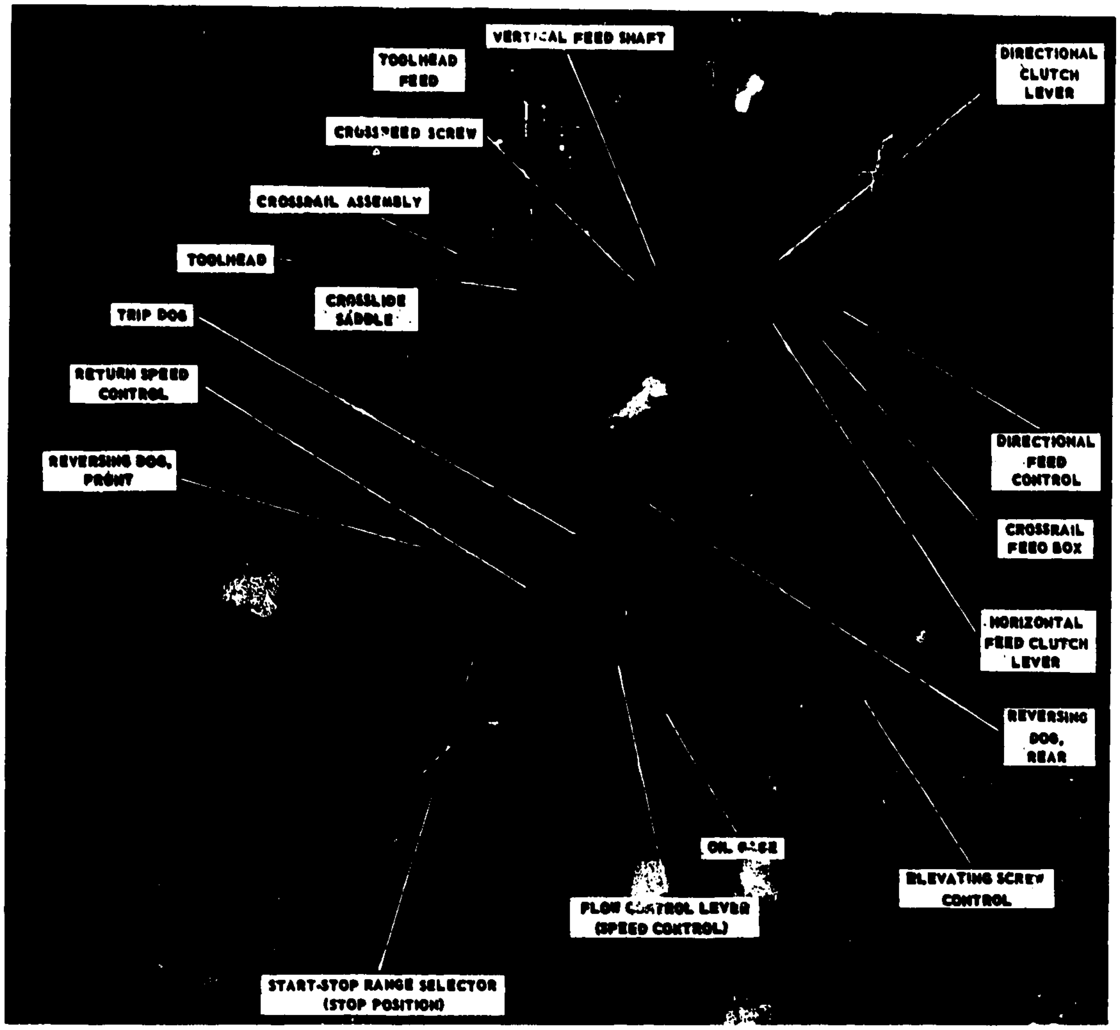
The crossrail serves as the rigid support for the toolheads. Two adjustments on the crossrail, vertical and horizontal feed screws, enable an operator to adjust for various size pieces of work.

The toolhead is similar to that of the shaper in operation and construction.

All sliding surfaces subject to wear are provided with adjustments. Keep the gibs adjusted to take up any looseness due to wear.

OPERATING THE PLANER

Before operating a planer, be sure that you know where the various controls are and what function each controls. Once you have mastered the operation of one model or type of planer, you will have little difficulty in operating others. Specific details of operation, however, should be obtained from the manufacturer's technical manual for the machine you are using. General



28.230X

Figure 12-13.—Open side planer.

information on one type of planer is provided in the following sections.

Table Speeds

The table speeds are controlled by the start-stop lever and the flow control lever (fig. 12-13). There are two ranges of speeds provided and a variation of speeds within each range. The speed range (LOW-MAXIMUM CUT or HIGH-MINIMUM CUT) is selected by the start-stop lever, and the variation of speeds within each range is accomplished by means of the flow control lever. By moving the flow control lever toward the right, the table speed will be gradually increased until it reaches the highest possible speed.

The LOW speed range is for shaping hard materials which require high cutting force at low speeds. The HIGH range is for softer materials which require less cutting force but higher cutting speeds.

The RETURN speed control provides two return speed ranges (NORMAL and FAST).

When NORMAL is selected, the return speed varies in ratio with the cutting speed selected. In FAST, the return speed remains constant (full speed), independent of the cutting speed setting.

Feeds

Feed adjustment is made by turning the handwheel which controls the amount of toolhead feed. Turning the handwheel clockwise decreases the amount of feed, while turning the handwheel counterclockwise increases the feed. The amount of feed can be read on the graduated dials at the operator's end of the crossrail feed box. Each graduation indicates a movement of 0.001 inch.

The toolhead on the crossrail is controlled as to direction of feed (right or left, up or down) by the lever on the rear of the feed box. The vertical feed is engaged or disengaged by the upper of the two levers on the front of the feed box. Shifting the rear, or directional, lever to the down position and engaging the clutch lever by pressing it downward gives a downward feed to the toolhead. Shifting the directional lever to the up position gives an upward feed.

The bottom clutch lever on the front of the feed box engages the horizontal feed of the toolhead. The directional lever on the rear of the box in the down position feeds the head toward the left, or away from the operator. In the up position it feeds toward the right.

The ball crank on top of the vertical slide (toolhead feed) is for the hand movement, up or down, of the toolslide. A graduated dial directly below the crank indicates the amount of travel.

For traversing the toolhead up or down, a handcrank is provided for use on the upper of the two squared shafts at the end of the crossrail. Horizontal traverse is accomplished by using the crank on the lower squared shaft. When using either of these squared shafts, be sure that the directional lever on the rear of the feed box is in the central or neutral position.

Lock screws are provided on both the cross-slide saddle and the vertical slide so that these slides may be locked in position after the desired tool setting has been made.

Planers having a side head are provided with power vertical feed and hand horizontal feed for the side head. The vertical feed is engaged or disengaged and the direction, up or down, is selected by means of the lever on the rear of the side head feed box. Vertical traverse of the side head is accomplished through the squared shaft projecting from the end of the feed box. Horizontal movement, both feed and traverse, is by means of the bellcrank on the end of the toolhead slide.

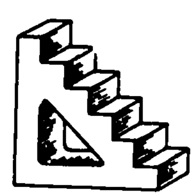
Rail Elevation

The crossrail is raised or lowered by a handcrank on the squared shaft projecting from the rear of the rail brace. When moving the rail, first loosen the two clamp nuts at the rear of the column and the two clamp nuts at the front; then with the handcrank move the rail to the desired height. Be sure to tighten the clamp nuts before doing any machining.

On machines having power rail elevation, a motor is mounted within the rail brace and connected to the elevating mechanism. Operation of the motor, forward or reverse, is controlled by pushbuttons suitably marked. The clamp nuts have the same use on all machines whether manual or power elevation is used.

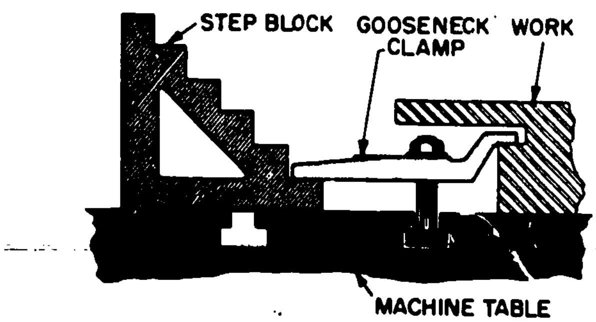
Holding the Work

The various accessories generally used in planer or shaper work may make the difference between a job well done and a poor job. There are no set rules on utilization of planer accessories for clamping down a piece of



11.15

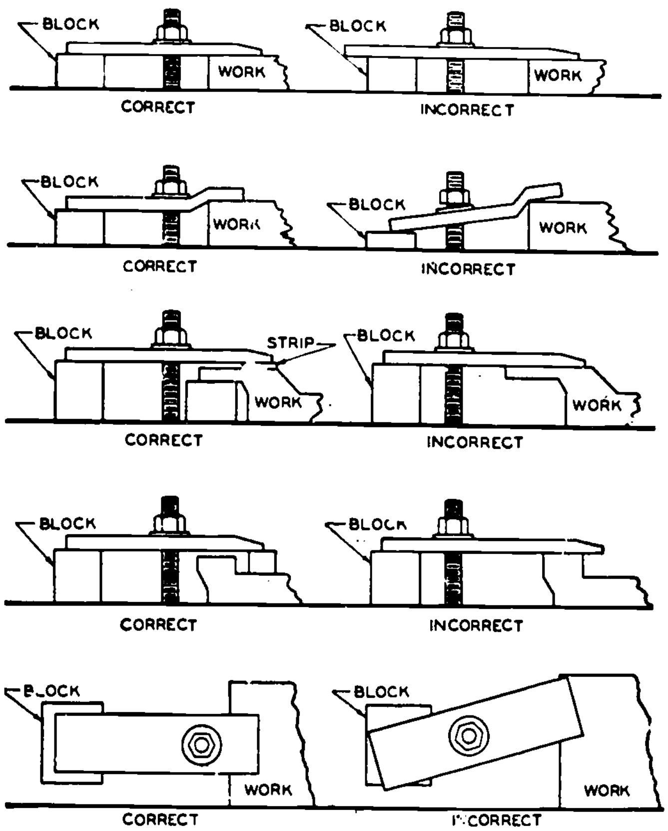
Figure 12-14.—Step block.



28.231
Figure 12-15.–Application of step block and clamp.

work–results will depend upon your ingenuity and experience.

One means of holding down work on the worktable is by using clamps. The clamps are bolted down to the worktable by way of the



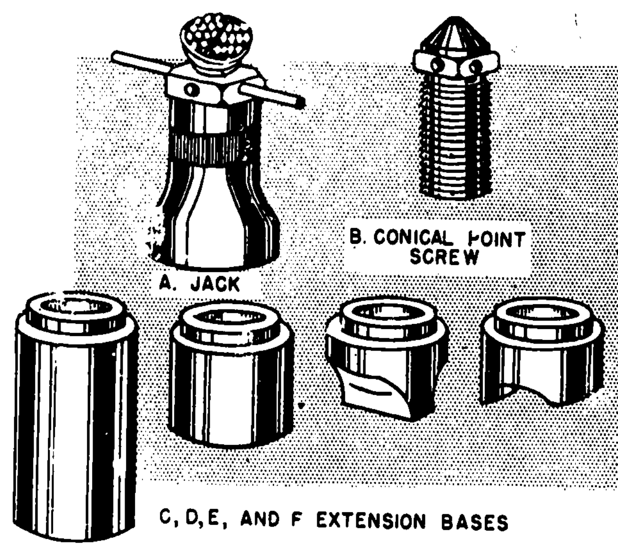
28.232
Figure 12-16.–Correct and incorrect clamp applications.

T-slots. Figure 12-14 illustrates a step block used with the clamps shown in figure 5-32. At some time it may be necessary to clamp an irregular piece of work to the planer table. A typical application for holding irregularly shaped work is illustrated in figure 12-15; here an accurately machined step block is used with a gooseneck clamp. Figure 12-16 illustrates the correct and incorrect methods of clamp application.

For leveling and supporting work on the planer table, jacks of different sizes are used. The conical point screw (fig. 12-17) replaces the swivel type screw for use in a corner. Extension bases (fig. 12-17, C, D, E, and F) are used for increasing the effective height of the jack.

SURFACE GRINDING ON THE PLANER

While it is not a recommended practice, it is possible, with the use of a toolpost grinder, to use the planer as a surface grinder. Most of the large tender and repair type ships of the Navy have surface grinders on board, but due to space limitations this machine may not always have a large enough capacity to accommodate large work pieces. It sometimes may become necessary to utilize the planer as a surface grinder. Basically speaking, it is a matter of



28.233
Figure 12-17.–Planer jack and extension bases.

replacing the toolbit with the toolpost grinder and computing feeds and speeds for grinding instead of planing. Prior to attempting surface grinding on the planer, a thorough understanding of chapter 13 of this manual will be necessary.

Once the grinding job is completed, the planer must be extensively cleaned inside and out. The oil in the hydraulic system must be filtered or changed prior to further operation. The planer, unlike the surface grinder, has no built-in protection against the grinding particles left by the grinding operation.

The same safety precautions observed for the shaper apply to the planer. Standard machine shop practices should always be observed.

PANTOGRAPHS

The pantograph (engraving machine) is essentially a reproduction machine. It is used in the Navy for work such as engraving letters and numbers on label plates, engraving and graduating dials and collars, and in other work which requires the exact reproduction of a flat pattern on the workpiece. The pantograph may be used for engraving flat and uniformly curved surfaces.

There are several different models of engraving machines which you may have to operate. Figure 12-18 shows one model which mounts on a bench or a table top and is used primarily for engraving small items. It is capable of reproducing work at ratios ranging from 1:1 to 7:1. A 1 to 1 ratio will result in the work being the same size as the pattern being copied, while a 7 to 1 ratio will result in the work being 1/7 the size of the pattern. This particular machine is manufactured by the New Hermes Engraving Machine Corporation.

The Gorton 3-U pantograph (figure 12-19) is commonly used by the Navy. The principles of operation and setup procedures for the 3-U machine are similar to those for other models of pantograph type engraving machines. Because of the similarity in operating principles and setup procedures, you should have no difficulty in applying the information contained in this section to the operation of any model of pantograph engraver.

PANTOGRAPH ENGRAVER UNITS

The pantograph engraving machine, shown in figure 12-19, consists of five principal parts: the supporting base, pantograph assembly, cutterhead assembly, worktable, and copyholder.

Supporting Base

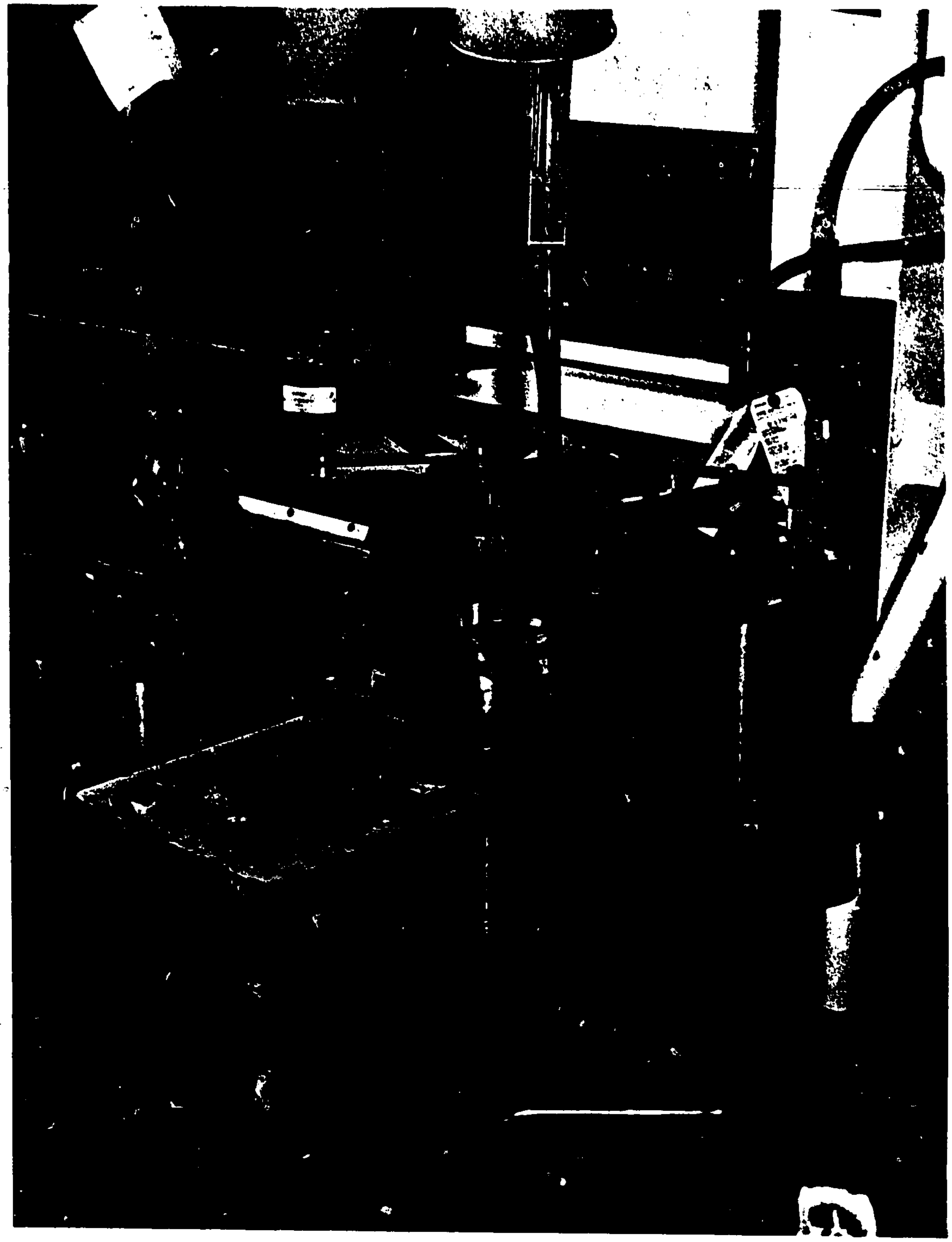
The supporting base is a heavy, rigid casting which supports the entire piece of machinery. If the machine table is not level, the base should be shimmed. If the deck vibrates too much because of surrounding machinery, the base should be set on rubber or cork pads.

Pantograph Assembly

The pantograph assembly has four. connecting arms: a tracer arm, an upper bar, a lower bar, and a connecting link between the tracer arm and the lower bar, and also the cutterhead link which supports the cutterhead. The relationship between movement of the stylus point and movement of the cutter is governed by the relative positions of the sliding blocks on the upper bar and the lower bar. The pantograph assembly can be set for a given reduction by loosening the sliding block bolts and setting the blocks at a desired distance from the datum lines. This will give the desired reduction ratio. The upper and lower bar are inscribed with marks (for whole number and standard reductions from 2:1 to 16:1) to indicate the position for setting the slider blocks for commonly used reductions.

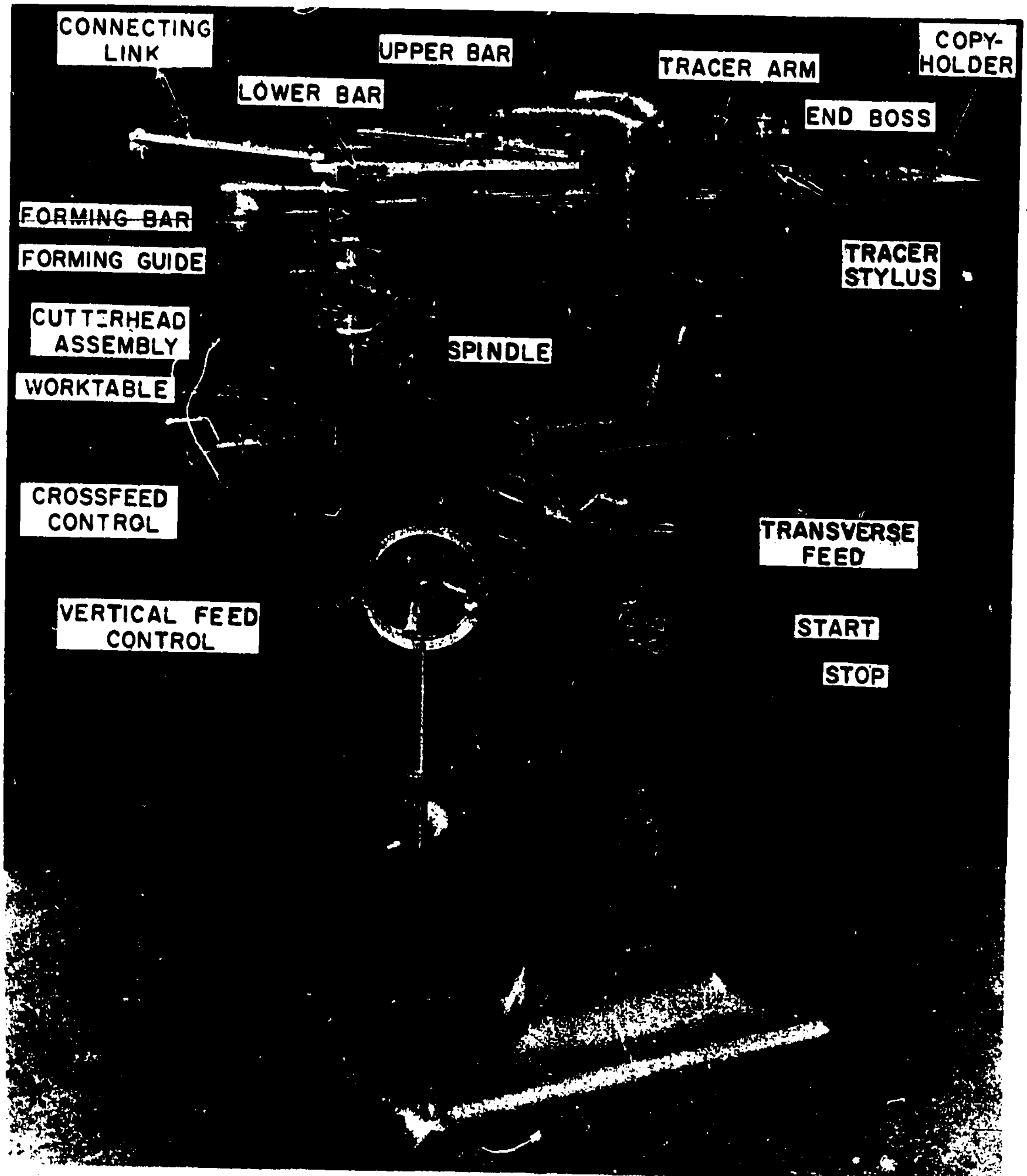
Cutterhead Assembly

The cutterhead assembly houses the precision cutter spindle. Pulley drives between the motor and spindle enable you to adjust the



:8.332

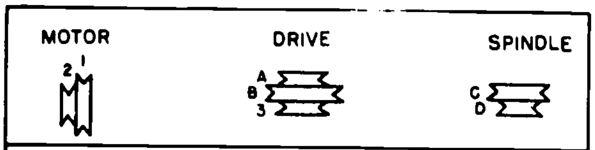
Figure 12-18.–Engraving machine.



28.234X

Figure 12-19.—Pantograph, Model 3-U (Gorton).

spindle speeds. Figure 12-20 gives the spindle speeds and the arrangement of the drive belts for varying spindle speeds. At the head of the cutter there is a vertical feed lever which provides a range of vertical movement from 1/16 inch to 1/4 inch to prevent the cutter from breaking when feeding into work. These are both automatically obtained in one lever movement. A plunger locks the spindle for flat surface engraving or releases it for floating vertical



2-3-A-C	3800 rpm	2-3-B-C	8100 rpm
2-3-A-D	5300 rpm	2-3-B-D	11,000 rpm
1-3-A-C	5300 rpm	1-3-B-C	11,000 rpm
1-3-A-D	7400 rpm	1-3-B-D	15,000 rpm

28.235X

Figure 12-20.–Spindle speeds.

movement of 1/2 inch with the forming guide on curved work. The cutterhead assembly is hinged to permit spindle removal from the side, making it unnecessary to disturb any work by lowering the table.

Worktable

The worktable of the 3-U pantograph engraver is 8 inches by 12 inches, flat, and of highly polished cast iron. The worktable surface has four 3/8-inch T-slots for mounting a vise or table dogs to hold down a piece of work. T-slots in the worktable are parallel with the front edge of the table. Longitudinal feed can move the worktable 10 inches, while the crossfeed can move the table 11 inches. Vertical feed of the worktable is 9 3/4 inches.

Copyholder

The copyholder is made of steel castings with beveled grooves or T-slots machined from the solid plate holder. Standard copyholders for the 3-U pantograph engravers have four or six grooves. Two stops are supplied for each groove in the copyholder.

SETTING COPY

Lettering, used in conjunction with an engraver, is known by various terms–however, the Navy uses a term "copy" to designate the characters used as sample guides. Copy applies specifically to the standard brass letters or type which are set in the copyholder of the machine and which guide the pantograph in reproducing. Shapes, as distinguished from characters, are called templates or masters.

Strictly speaking, copy is not self-spacing; therefore, the spaces between the characters should be adjusted by inserting suitable blank spacers which are furnished with each set of copy. Each line when set in the copyholder should be held firmly between the clamps.

After setting up the copy in the holder, and before engraving, be sure that the holder is firmly set against the stop screws in the copyholder base. This ensures that the holder is square with the table. Do not disturb these stops; they have been properly adjusted by the factory, and any change will throw the copyholder out of square with the table. The worktable T-slots are parallel with the table's front edge, making it easy to set up the work and the copy in accurate parallel relation to each other.

In addition to copy, circular copy plates are sometimes used for engraving work. A copy plate is a flat disk with letters, numbers, and other characters inscribed on the face of the disk near the rim. The rim of the plate is notched beside each character so that a spring-loaded indexing pawl can be used to hold the disk in the proper position during the engraving procedure. The plate is set on a pivot on the copyholder and may be rotated 360° so that any character on the plate may be placed in the required position.

SETTING THE PANTOGRAPH

The correct setting of the pantograph is determined from a ratio (1) of the size of the work to the size of the copy layout, or (2) of the desired size of engraved characters to the size of the copy characters. This ratio is called a reduction. A 1:1 reduction results in an engraved layout equal in size to the copy layout;

a 16:1 reduction results in an engraved layout 1/16 the size of the copy layout.

If a length of copy is 10 inches and the length of the finished job is to be 2 inches, divide the length of the job into the length of the copy:

$$10 \div 2 = 5 \text{ inches}$$

Therefore, set the slider blocks at reduction.

If the length of the copy is 11 inches and the length of the finished job is to be 4 inches, the reduction is:

$$11 \div 4 = 2.75 \text{ inches}$$

You will note that reduction 2.75 is not marked on the pantograph bars. To find the correct slider blocks settings, use the reduction formula in figure 12-21.

All settings are measured from the first reduction marking on the upper and lower arms. On the model 3-U pantograph, reductions are measured from the line marked 2 on the upper arm, and NOT the line marked 1. For accurately setting special reductions use a hundredth-inch scale and a magnifying glass.

After a special reduction has been set, check the pantograph. First, place a point into the spindle, then raise the table until the point barely clears the table. Next, trace along an edge of a copy slot in the copyholder with the tracing stylus. If the cutter point follows parallel to the T-slots, the reduction is proper. If the point forms an arc or angle, the setting should be recalculated and reset. If the point still runs off, loosen either of the slider blocks and tap one way or the other, until the path of the point is parallel to the T-slots.

For 1:1 reduction, transfer the stylus collet from the end boss of the tracer arm to the second boss on the arm. Set the lower slider block on the graduation marked "1 and 2," and the upper bar slider block on graduation 1.

Table 12-2 provides dimensions for setting slider blocks on the upper and lower bars for reductions 2 through 16. After setting the reduction, lock the upper and lower bars in the slider blocks by tightening the capscrews in each block. (Note: For special reductions between 1 and 2, follow the sample solution in fig. 12-22.)

CUTTER SPEEDS

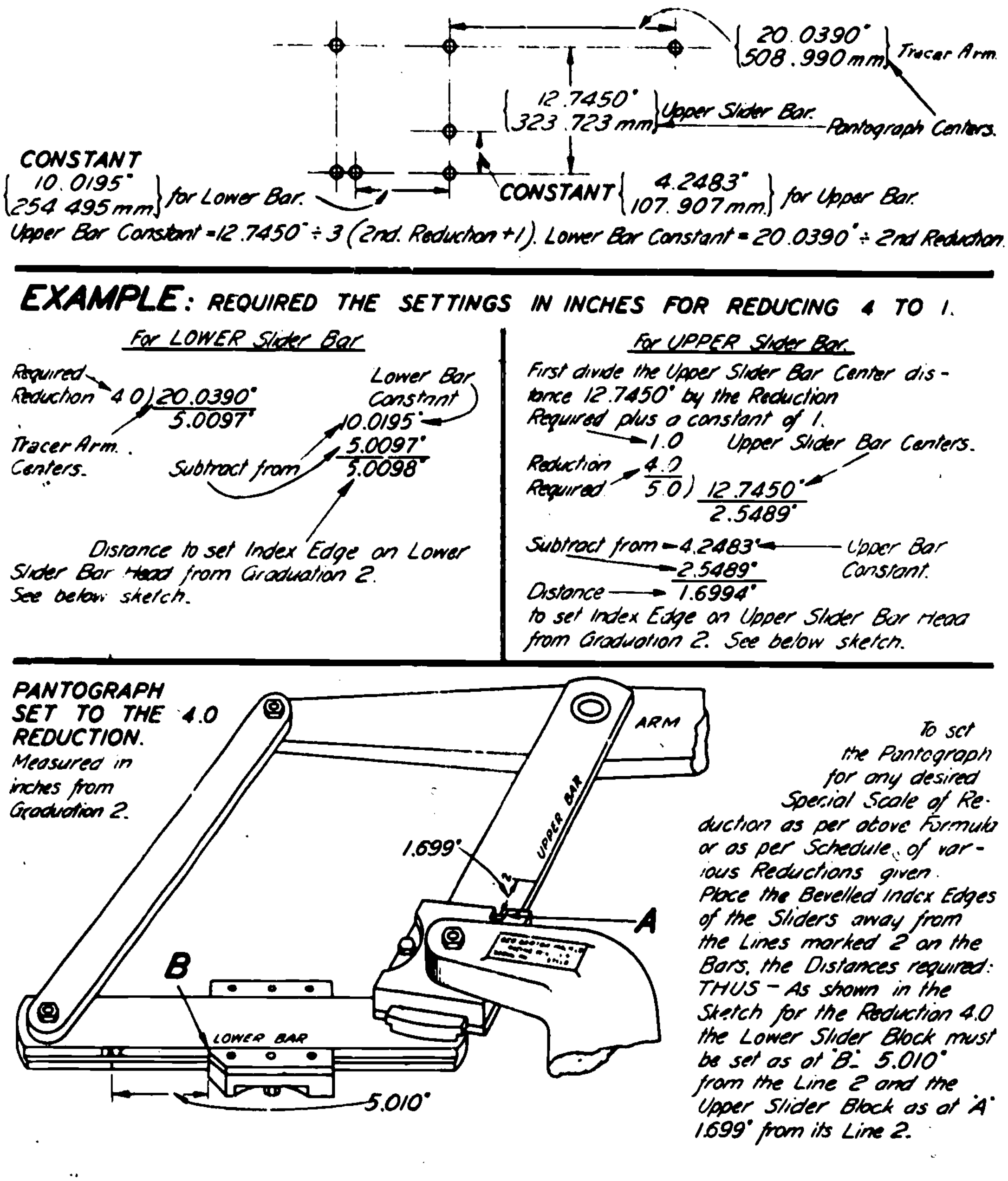
The speeds listed in table 12-3 represent typical speeds for given materials. In using the table, keep in mind that the speeds recommended will vary greatly, depending on the depth of cut, and particularly the rate at which the cutter is fed through the work. Since the 3-U engravers are fed manually, the rate of feed is subject to a wide variation by individual operators; this will affect the spindle speeds used.

Run the cutters at highest speeds possible, and remove stock with several light, fast cuts rather than one heavy cut at slower spindle speeds. Always use the highest speed possible without burning the cutter. When cutting steel and all hard materials, start with a slow speed and work up to the fastest speed a cutter will stand without losing its cutting edge. Sometimes it may be advisable to sacrifice cutter life to obtain the smoother finish possible at higher speeds. With experience you will know when the cutter is running at maximum efficiency.

GRINDING CUTTERS

Most of the difficulties experienced in using very small cutters on small lettering are caused by improper grinding. The cutter point must be accurately sharpened. When trouble is experienced, usually the point is burned, or the flat is either too high or too low. Perhaps the clearance does not run clear out to the point. Stoning off the flat with a small fine oilstone will make the cutting edge keener.

A cutter can be made to run almost perfectly by sharpening the cutter in the spindle



28.236X

Figure 12-21.–Formula for obtaining special reductions for model 3-U pantograph.

in which it runs. Most pantograph machines have provision for removing the cutter spindle from the machine and placing it in a V-block toolhead on the cutter grinder. The cutter is then ground to the desired shape without removing it from the cutter spindle.

Grinding Single-Flute Cutters

Before grinding cutters, true up the grinding wheel with the diamond tool supplied with the grinder. After inserting the diamond, set the toolhead at approximately the same relation to

Table 12-2.—Reduction Schedules in Inches and Millimeters

Schedule of Reductions for Engraving Machine No. 3U		
Reduction	Lower Bar Inches	Upper Bar Inches
2.0	0.000	0.000
2.1	0.477	0.137
2.2	0.911	0.265
2.3	1.307	0.386
2.4	1.670	0.500
2.5	2.004	0.607
2.6	2.312	0.708
2.7	2.598	0.804
2.8	2.863	0.894
2.9	3.109	0.980
3.0	3.340	1.062
3.1	3.555	1.140
3.2	3.757	1.214
3.3	3.947	1.284
3.4	4.126	1.352
3.5	4.294	1.416
3.6	4.453	1.478
3.7	4.604	1.537
3.8	4.746	1.593
3.9	4.881	1.647
4.0	5.010	1.699
4.1	5.132	1.749
4.2	5.248	1.797
4.3	5.359	1.844
4.4	5.465	1.888
4.5	5.566	1.931
4.6	5.663	1.972
4.7	5.756	2.012
4.8	5.845	2.051
4.9	5.930	2.088
5.0	6.012	2.124
5.1	6.090	2.159
5.2	6.166	2.193
5.3	6.239	2.225
5.4	6.309	2.257

Schedule of Reductions for Engraving Machine No. 3U		
Reduction	Lower Bar Millimeters	Upper Bar Millimeters
2.0	00.00	0.00
2.1	12.12	3.48
2.2	23.14	6.74
2.3	33.19	9.81
2.4	42.42	12.69
2.5	50.90	15.41
2.6	58.73	17.98
2.7	65.98	20.41
2.8	72.71	22.72
2.9	78.98	24.90
3.0	84.83	26.98
3.1	90.30	28.95
3.2	95.44	30.83
3.3	100.26	32.62
3.4	104.79	34.33
3.5	109.07	35.97
3.6	113.11	37.53
3.7	116.93	39.03
3.8	120.55	40.46
3.9	123.98	41.84
4.0	127.25	43.16
4.1	130.35	44.43
4.2	133.31	45.65
4.3	136.13	46.83
4.4	138.82	47.96
4.5	141.39	49.05
4.6	143.84	50.10
4.7	146.20	51.11
4.8	148.46	52.09
4.9	150.62	53.04
5.0	152.70	53.95
5.1	154.69	54.84
5.2	156.61	55.69
5.3	158.46	56.52
5.4	160.24	57.33

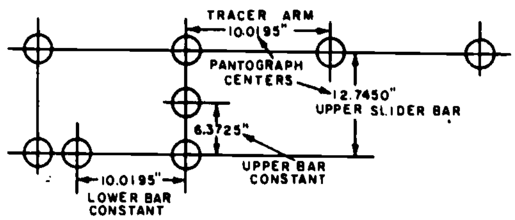
28.236.01X

Table 12-2.–Reduction Schedules in Inches and Millimeters–Continued

Schedule of Reductions for Engraving Machine No. 3U		
Reduction	Lower Bar Inches	Upper Bar Inches
5.5	6.376	2.288
5.6	6.441	2.317
5.7	6.504	2.346
5.8	6.564	2.374
5.9	6.623	2.401
6.0	6.680	2.428
6.1	6.734	2.453
6.2	6.787	2.478
6.3	6.839	2.502
6.4	6.888	2.526
6.5	6.937	2.549
6.6	6.983	2.571
6.7	7.029	2.593
6.8	7.073	2.614
6.9	7.115	2.635
7.0	7.157	2.655
7.1	7.197	2.673
7.2	7.236	2.694
7.3	7.274	2.713
7.4	7.312	2.731
7.5	7.348	2.749
7.6	7.383	2.766
7.7	7.417	2.783
7.8	7.450	2.800
7.9	7.483	2.816
8.0	7.515	2.832
9.0	7.793	2.974
10.0	8.016	3.090
11.0	8.198	3.186
12.0	8.350	3.268
13.00	8.478	3.338
14.00	8.588	3.399
15.00	8.683	3.452
16.00	8.767	3.499

Schedule of Reductions for Engraving Machine No. 3U		
Reduction	Lower Bar Millimeters	Upper Bar Millimeters
5.5	161.95	58.10
5.6	163.60	58.86
5.7	165.20	59.59
5.8	166.74	60.30
5.9	168.23	60.99
6.0	169.66	61.66
6.1	171.05	62.31
6.2	172.40	62.95
6.3	173.70	63.56
6.4	174.97	64.16
6.5	176.19	64.74
6.6	177.38	65.31
6.7	178.53	65.87
6.8	179.64	66.40
6.9	180.73	66.93
7.0	181.78	67.44
7.1	182.8[illegible]	67.94
7.2	183.80	68.43
7.3	184.77	68.90
7.4	185.71	69.37
7.5	186.63	69.82
7.6	187.32	70.26
7.7	188.39	70.70
7.8	189.24	71.12
7.9	190.07	71.53
8.0	190.87	71.94
9.0	197.94	75.53
10.0	203.60	78.48
11.0	208.22	80.93
12.0	212.08	83.01
13.0	215.34	84.78
14.0	218.13	86.32
15.0	220.56	87.67
16.0	222.68	88.86

28.236.02X



EXAMPLE

REQUIRED: THE SETTING IN INCHES FOR REDUCING 1.5 TO 1

FOR LOWER SLIDER BAR

STEP 1. DIVIDE TRACER ARM CENTERS BY THE REQUIRED REDUCTION THUS:

$$\frac{\text{TRACER ARM CENTERS } 10.0195''}{\text{REQUIRED REDUCTION } 1.5} = 6.679''$$

STEP 2. SUBTRACT THE QUOTIENT FROM THE LOWER BAR CONSTANT. 10.0195" − 6.679" = 3.340"

STEP 3. THE RESULT IS THE DISTANCE TO SET INDEX EDGE ON LOWER SLIDER BAR HEAD FROM GRADUATION 1 & 2.

FOR UPPER SLIDER BAR

STEP 1. DIVIDE UPPER SLIDER BAR CENTER DISTANCE BY THE REDUCTION REQUIRED PLUS A CONSTANT OF ONE.

REQUIRED REDUCTION 1.5
CONSTANT 1.0
2.5

$$\text{UPPER SLIDER BAR CENTERS } \frac{12.745''}{2.5} = 5.098''$$

STEP 2. SUBTRACT THE QUOTIENT FROM THE UPPER BAR CONSTANT 6.3725" − 5.098" = 1.2745"

STEP 3. THE RESULT IS THE DISTANCE TO SET INDEX EDGE ON UPPER SLIDER BAR HEAD FROM GRADUATION 1.

SCHEDULE OF VARIOUS REDUCTIONS BETWEEN 1:1 AND 2:1 ON MOD. 3U PANTOGRAPH WITH TRACING STYLUS IN NEAREST HOLE OF ARM.

MEASUREMENTS IN INCHES

REDUCTION	DISTANCE TO SET INDEX EDGE ON LOWER SLIDER BAR HEAD FROM GRAD. MARKS 1 & 2	DISTANCE TO SET INDEX EDGE ON UPPER SLIDER BAR HEAD FROM GRAD. MARK 1
1.0	0	0
1.1	.911"	.303"
1.2	1.670"	.579"
1.3	2.312"	.831"
1.4	2.863"	1.062"
1.5	3.340"	1.275"
1.6	3.757"	1.471"
1.7	4.126"	1.651"
1.8	4.453"	1.821"
1.9	4.746"	1.978"

FOR OTHER REDUCTIONS USE FORMULA

FOR GREATER REDUCTIONS USE SCHEDULE AS PER INSTRUCTION BOOK WITH TRACING STYLUS AT EXTREME END OF PANTOGRAPH ARM

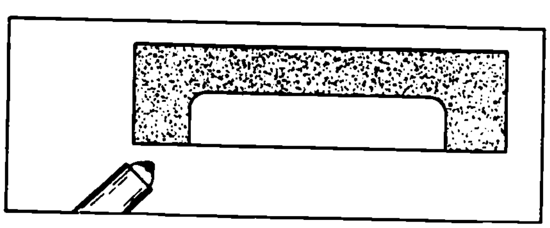
28.237X

Figure 12-22.—Formula for obtaining special reductions between 1 and 2.

the wheel as shown in figure 12-23. Then swing the diamond across the face of the wheel by rocking the toolhead in much the same manner as for grinding a cutter. In dressing the wheel, a cut of 0.001 or 0.002 inch should be the very maximum. If the diamond fails to cut freely, turn it slightly in the toolhead to present an unused portion of the diamond to the wheel.

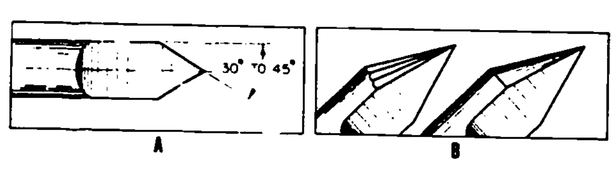
ROUGH AND FINISH GRINDING CONICAL POINT.—Set the grinder toolhead to the desired cutting edge angle (fig. 12-24A). This angle usually varies from 30° to 45°, depending upon the work desired. for most sunken letter or design engraving on metal or bakelite plates, a 30° angle is used. Now place the cutter in the toolhead and rough grind to approximate size by swinging across the wheel's face. Do not rotate the cutter while in contact with the face of the wheel but swing straight across, turning cutter slightly BEFORE or AFTER contact with wheel. This will produce a series of flats as in figure 12-24B. Now, grind off the flats and produce a smooth cone by feeding the cutter into the wheel and rotating the cutter at the same time. The finished cone should look like figure 12-24B, smooth and entirely free from wheel marks.

GRINDING FLAT TO CENTER.—The next operation is to grind the flat to center. For very



28.238X

Figure 12-23.—Position of diamond for truing a grinding wheel.



28.239X

Figure 12-24.—Grinding conical point. A. Cutter angle. B. Rough and finished conical shape.

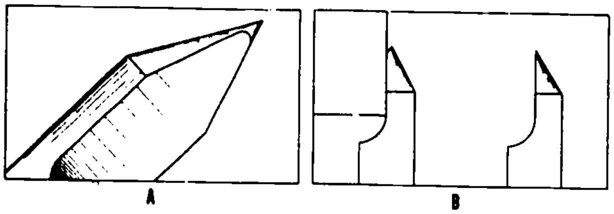
Table 12-3.—Cutter Speeds

Revolutions per minute for high speed steel cutters, single-flute type. Use two-thirds of speeds shown for 2- and 4-flute end mills; one-half speeds for 6-flute end mills.

Materials and Feeds	Cutter diameter (at cutting point)								
	1/32"	1/16"	1/8"	3/16"	1/4"	5/16"	3/8"	7/16"	1/2"
	Speeds (rpm)								
Hardwood (650-800 ft./min.)----	10,000 to 20,000	10,000 to 20,000	10,000 to 20,000	10,000 to 20,000	10,000 to 20,000	9,000	8,000	7,000	6,000
*Bakelite (170-250 ft./min.)----	10,000	8,000	6,000	4,000	3,000	2,200	1,800	1,500	1,300
**Engraver's brass and aluminum (375-425 ft./min.)	10,000 to 15,000	10,000 to 15,000	10,000 to 15,000	8,000	6.000	5,000	4,000	3,500	3,000
Cast iron (130-250 ft./min.)----	8,000	7,500	5,500	3,500	2,500	2,000	1,650	1,400	1,200
Hard bronze and machine steel (80-200 ft./min.)	7,000	6,000	3,000	2,200	1,600	1,200	975	800	700
Annealed tool steel (70-100 ft./min.)	5,000	4,500	2,300	1,600	1,200	1,000	850	725	600
Stainless steel, Monel (45-75 ft./min.)	3,500	2,750	1,400	1,050	700	575	500	435	350
Very hard die and alloy steels (30-45 ft./min.)	2,000	1,250	800	600	475	400	350	300	250

*Also celluloid, hard rubber, pearl, ivory, and synthetic plastics.

**Slightly lower speeds for ordinary brass, zinc, copper, silver, gold, soft bronze, and German silver.



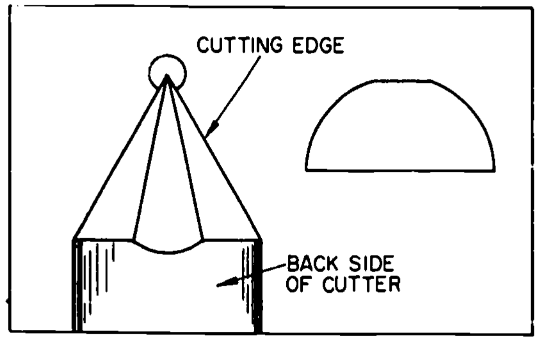
28.240X
Figure 12-25.—Grinding the flat. A. Flat not ground to center. B. Flat ground to center.

small, delicate work it is absolutely essential to grind this flat EXACTLY to center. If the flat is oversize, you can readily see it after grinding the cone, and the point will appear as in figure 12-25A. To correct this, grind the flat to center as in figure 12-25B.

GRINDING CHIP CLEARANCE.—The cutter now has the correct angle and a cutting edge, but has no chip clearance. This must be provided to keep the back side of the cutter from rubbing against the work and heating excessively, and to allow the hot chips to fly off readily. The amount of clearance varies with the angle of the cutter. The procedure for grinding chip clearance is as follows.

Gently feed the cutter into the face of the wheel. Do not rotate the cutter. Hold the back (round side) of the conical point against the wheel. Gradually feed in toward the wheel while rocking the cutter continuously across the wheel's face and without turning, until a flat is ground which runs out exactly at the cutter point (fig. 12-26). Check this very carefully, with a magnifying glass if necessary, to be sure you have reached the point with this flat. Be extremely careful not to go beyond.

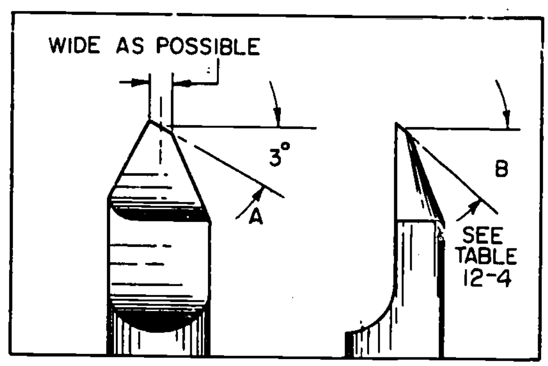
Rough away stock as in figure 12-24B. Rotate the cutter against the face of the wheel to grind away all stock on the back of the conical side, up to the cutting edge. Be extremely careful not to turn the cutter too far and thus grind away part of the cutting edge. Clean up all chatter marks. Be careful of the point; this is where the cutting is done. If this point is incorrectly ground, the cutter will not



28.241X
Figure 12-26.—First operation in grinding clearance.

work, regardless of how perfect it may be farther out on the taper of the cone.

TIPPING OFF THE CUTTER POINT.—For engraving hairline letters up to 0.0005 inch in depth, the cutter point is not flattened, or TIPPED OFF. For all ordinary work, however, it is best to flatten this point as much as the work will permit. Otherwise, it is very difficult to retain a keen edge with such a fine point, and when the point wears down, the cutter immediately fails to cut cleanly. Tipping off is usually done by holding the cutter in the hands at the proper inclination from the grinding wheel face and touching the cutter very lightly against the wheel, or by dressing with an oilstone. Angle A (fig. 12-27) should be



28.242X
Figure 12-27.—A tipped off cutter.

Table 12-4.—Rake Angles for Single-Flute Cutters

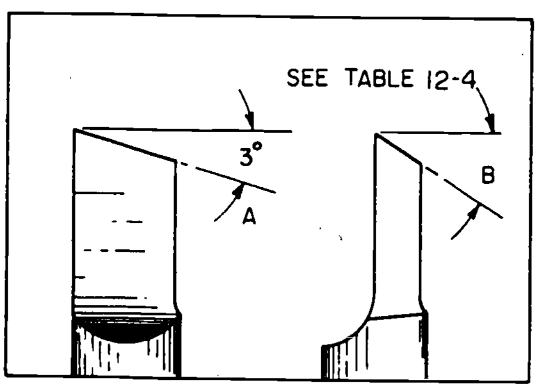
Material to be cut	Angle B (See figs. 12-27) and 12-28)
Tool steel	5-10 degrees
Machine steel	10-15 degrees
Hard brass	15-20 degrees
Aluminum	20-25 degrees
Bakelite, celluloid, wood, fiber	20-25 degrees

28.2

approximately 3°; this angle causes the cutter to bite into the work like a drill, when fed down. Angle B (fig. 12-27) varies, depending on the material to be engraved. Table 12-4 will serve as a guide in maintaining angle B.

Grinding Square-Nose Single-Flute Cutters

The properly ground square-nose single-flute cutter should be similar to the illustration in figure 12-28. When square-nose cutters are ground, they should be tipped off in the same manner as described in connection with figure 12-27. All square-nose cutters have peripheral clearance ground back of the cutting edge. After grinding the flat to center (easily checked wi micrometer), clearance is ground by feedin the required amount toward the wheel turning the cutter until all stock has t removed from the back (round side) right u the cutting edge. Table 12-5 prov information on chip clearance for various s cutters.

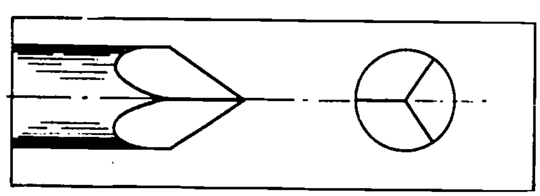


28.243X
Figure 12-28.—Square-nose cutter with properly ground tip.

Grinding Three- and Four-Sided Cutters

Three- and four-sided cutters (see fig. 12- are used for cutting small steel stamps and small engraving where a very smooth finis desired. The index plate on the toolhead cc spindle has numbered index holes for inde› to grind three- and four-sided cutters.

Set the toolhead for the desired angle. I the pin in the index hole for the desired num of divisions and grind the flats. Now with loosening the cutter in the toolhead collet, r



28.24
Figure 12-29.—Three-sided cutter.

Table 12-5.—Chip Clearance Table for Square-Nose Cutters

Cutter diameter	Clearance	Cutter diameter	Clearance
Inches	Inches	Inches	Inches
1/10	.004	1/4	.010
1/8	.006	5/16	.012
5/32	.006	3/8	.015
3/16	.008	7/16	.015
		1/2	.020

28.244.0

Table 12-6.—Clearance Angles for 3- and 4-Sided Cutters

Degrees of cutting	45°	40°	35°	30°	25°	20°	15°	10°	5°
Angle of clearance: (Degrees)									
3 sides	26 1/2	23	19 1/2	16	13	10 1/2	7 1/2	5	2 1/2
4 sides	35 1/2	23	25 1/2	22 1/2	18 1/2	14 1/2	10	7	3 1/2

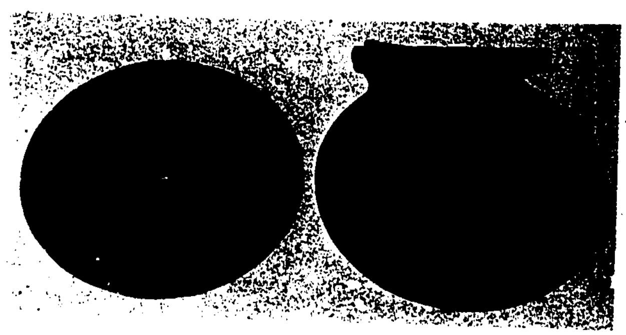
28.245.0

the toolhead to the proper clearance angle. Clearance angles are listed in table 12-6.

PANTOGRAPH ATTACHMENTS

Some attachments commonly used with the pantograph engraving machine are: copy dial holders, indexing attachments, forming guides, and rotary tables. The use of these attachments extends the capabilities of the pantograph engraving machine from flat, straight line engraving to include circular work, cylindrical work, and indexing.

The copy dial holder shown in figure 12-30 is used instead of the regular copyholder when a circular copy plate is used. This holder has a



28.245X
Figure 12-30.—Copy dial holder and plate.

spring-loaded indexing pawl which is aligned with the center pivot hole. This pawl engages in the notches in circular copy plates to hold the plate in the required position for engraving the character concerned.

An indexing attachment such as that shown in figure 12-31 may be used for holding cylindrical work to be graduated. In some cases the dividing head (used on the milling machine) is used for this purpose. The work to be engraved is held in this attachment and may be indexed for any number of divisions available on the plate. Figure 12-31 shows a micrometer collar being held for graduation and engraving.

A forming guide (sometimes called a radius plate) is used when engraving cylindrical

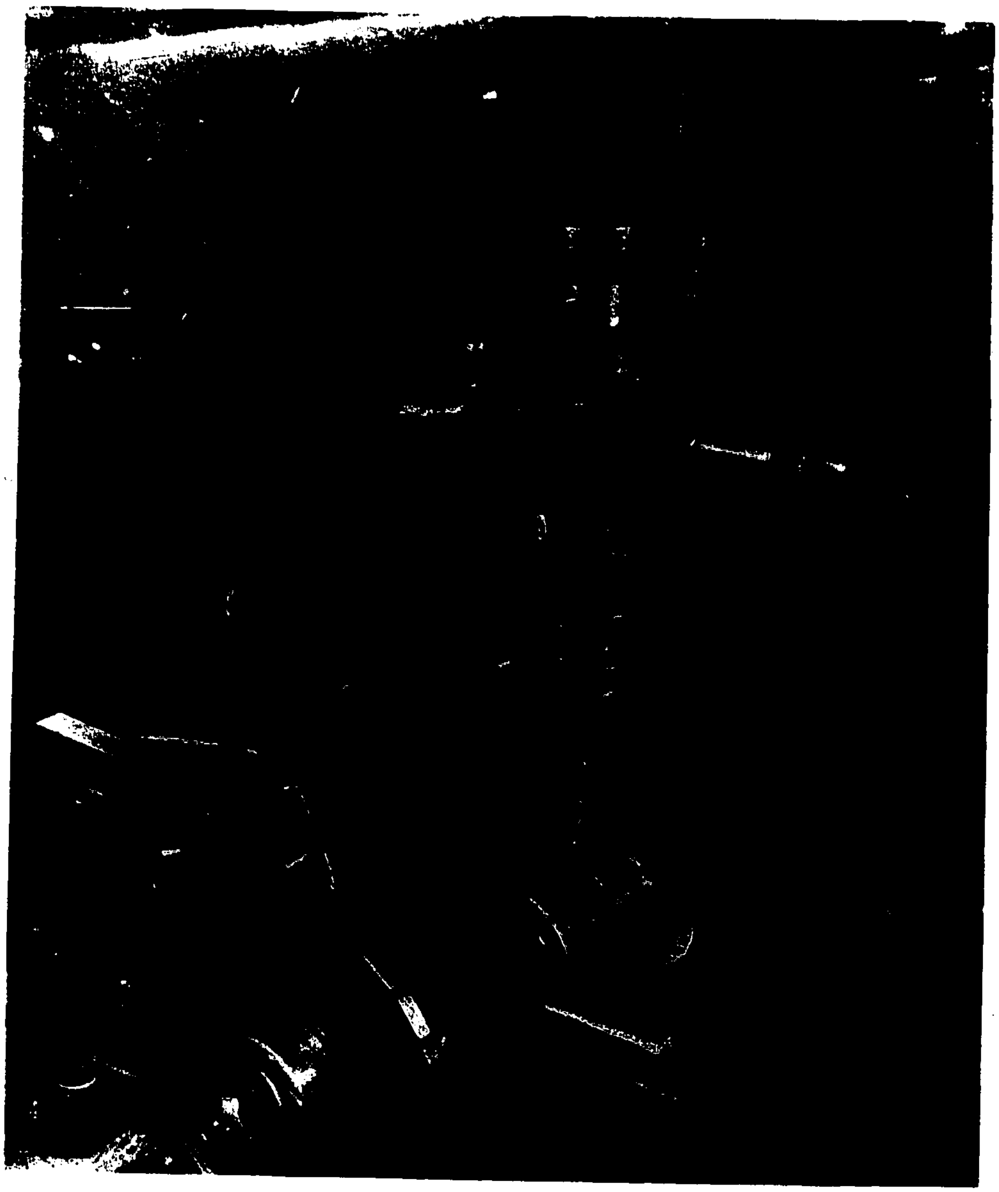


Figure 12-31.—Using an indexing attachment.

28.246X

surfaces. The contour of the guide must be the exact opposite of the work; if the work is concave the guide must be convex and vice versa. The forming guide is mounted on the forming bar. (See fig. 12-32.) When the spindle floating mechanism is released, the spindle follows the contour of the forming guide. This causes the cutter to move vertically in direct relation to the contour of the forming guide.

The rotary table shown in figure 12-32 is used for holding work such as face dials. It is similar to the rotary table used on milling machines. The rotary table is mounted directly on the worktable and provides a means of rapid graduation and of engraving the faces of disks.

USING A CIRCULAR COPY PLATE

The circular copy plate might be efficiently used in engraving a number of similar workpieces with single characters used consecutively. For example, to engrave 26 similar workpieces with a single letter, but with each piece having a different letter, the following setup can be used:

1. Set the workpiece conveniently on the worktable and clamp two aligning stops in place. These stops will not be moved until the entire job is completed.
2. Set the circular plate on the copyholder so that the plate can be rotated by hand. Check to ensure that the indexing pawl engages the notch on the rim so that the plate will be steady while tracing the character.
3. Set the machine for the required reduction and speed, and adjust the worktable so that the spindle is in position over the workpiece.
4. Clamp the first workpiece in place on the worktable. (Aligning stops, step 1, ensures accurate positioning.)
5. Rotate the circular plate until the letter A is under the tracing stylus and the index pawl is engaged in the notch.
6. Engrave the first piece with the letter A. Check the operation for required adjustments of the machine.
7. After the first piece is finished, remove it from the machine. Do not change the alignment of the aligning stops (step 1), worktable, or copyholder. Place the second workpiece in the machine. Index the circular plate to the next letter and proceed as previously described.
8. Continue loading the workpiece, indexing the plate to the next character, engraving, and removing the work, until completion of the job.

28.247X

Figure 12-32.—A rotary table.

ENGRAVING A GRADUATED COLLAR

To engrave a graduated collar, as shown in figure 12-31, a forming guide and indexing attachment are used. The circular copy plate can also be used to speed up the numbering process. After each graduation is engraved, the work is indexed to the next division until the graduating is completed. When engraving numbers with more than one digit, offset the work angularly by rotating the work so that the numbers are centered on the required graduation marks.

ENGRAVING A DIAL FACE

A rotary table and circular copy plate might be used in engraving the dial face shown in figure 12-33. Note that the figures on the right side of the dial are oriented differently from those on the left side; this illustrates the usual method of positioning characters on dials. The graduations are radially extended from the center of the face. The graduations also divide the dial into eight equal divisions.

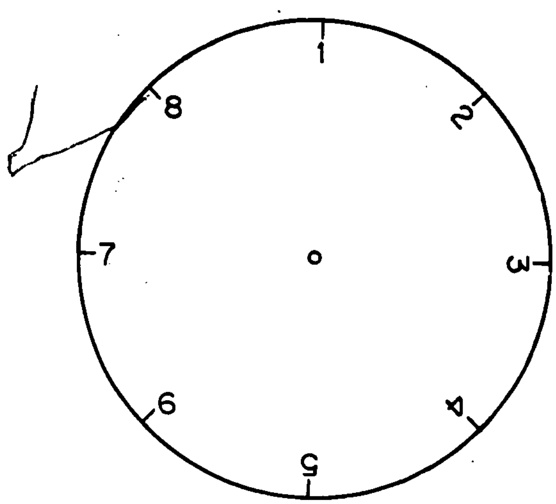
To set up and engrave a dial face, proceed as follows:

1. Set the reduction required. The size of the copy on the circular copy plate and the desired size of numerals on the work are the basis for computing the reduction.

2. Set the copy plate on the copyholder, ensuring that it is free to rotate when the ratchet is disengaged.

3. Mount a rotary table on the worktable of the engraver. Position the dial blank on the rotary table so that the center of the dial coincides with the center of the rotary table. Clamp the dial blank to the rotary table.

4. Place the tracing stylus in the center of the circular copy plate and adjust the worktable so that the center of the dial is directly under the point of the cutter.



28.248

Figure 12-33.—A dial face.

5. Rotate the copy plate until the copy character for making graduation marks is aligned with the center of the copy plate and the center of the work. Set the stylus in this mark. Now, by feeding the worktable straight in toward the back of the engraver, adjust the table so that the cutter will cut the graduation to the desired length.

6. Start the machine and adjust the engraver worktable vertically for the proper depth of cut. Then clamp the table to prevent misalignment of the work. Any further movement of the work will be made by the rotary table feed mechanishm.

7. Engrave the first graduation mark.

8. Using the rotary table feed wheel, rotate the dial to the proper position for the next graduation. As there are eight graduations here, rotate the table 45°; engrave this mark and continue until the circle is graduated. You will now be back to the starting point. (Note: The circular copy plate is not moved during the graduating process.)

9. To engrave numbers positioned as shown on the right side of the dial in figure 12-33, move the worktable so that the cutter is in position for engraving the numbers. Rotate the circular copy plate to the numeral 1 and engrave it. Rotate the rotary table 45° and the circular copy plate to 2, and engrave. Continue this process until all the numbers are engraved. If two (or more) digit numbers are required, offset the dial as previously described.

10. To engrave the numbers, as shown on the left side of the dial in figure 12-33, rotate the copy plate to the required number and then, using the crossfeed and longitudinal feed of the engraver table, position the cutter over the work at the point where the number is required. This method requires that the worktable be repositioned for each individual number. As previously stated, movement of the engraver worktable in two directions results in angular misalignment of the character with the radius of the face; in this example, angular misalignment is required.

CHAPTER 13

PRECISION GRINDING MACHINES

Modern grinding machines are versatile and are used to perform work of extreme accuracy. These machines are used primarily for finishing surfaces that have been machined in other machine tool operations. Surface grinders, cylindrical grinders, and tool and cutter grinders, installed in most repair ships, can perform practically all of the grinding operations required in Navy repair work.

A Machinery Repairman must demonstrate an ability to: (1) mount, dress, and true grind machine wheels; (2) perform precision grinding operations using a magnetic chuck; (3) grind cutter bits (tool bits) on a surface grinder for Acme and square threading; and (4) set up and grind milling cutters using a tool and cutter grinder.

To perform these jobs, you must have a knowledge of the construction and principles of operation of commonly used grinding machines.

You gain proficiency in grinding through practical experience. Therefore, you should take every available opportunity to watch or perform grinding operations from setup to completion.

There are many classes of each type of grinder. The SURFACE grinder may have a rotary table with a horizontal or a vertical spindle, or a reciprocating table with a horizontal or a vertical spindle. CYLINDRICAL grinders may be classified as plain, centerless, or internal grinders. The TOOL AND CUTTER grinder can be appropriately considered as a class of cylindrical grinder. Grinders generally found in the shipboard machine shop are the reciprocating table, horizontal spindle (planer type), surface grinder; the plain cylindrical grinder; the tool and cutter grinder; and sometimes a universal grinder. The UNIVERSAL grinder is similar to a tool and cutter grinder except that it is designed for heavier work and usually has a power feed system and a coolant system.

Before operating a grinding machine, you must understand the underlying principles of grinding and the purpose and operation of the various controls and parts of the machine. You must also know how to set up the work in the machine. The setup procedures will vary with the different models and types of machines. Therefore, you must study the manufacturer's technical manual to learn specific procedures for using a particular model of machine.

SPEEDS, FEEDS, AND COOLANTS

As with other machine tools, the selection of proper speed, feed, and depth of cut is an important factor in successful grinding. Also, the use of coolants may be necessary for some operations. The definitions of the terms "speed," "feed," and "depth of cut," as applied to grinding, are basically the same as for other machining operations.

IN FEED is the depth of cut that the wheel takes in each pass across the work. TRAVERSE (longitudinal or cross) is the rate that the work is moved across the working face of the grinding wheel. WHEEL SPEED, unless otherwise defined, means the surface speed in fpm of the grinding wheel.

WHEEL SPEEDS

Grinding wheel speeds commonly used in precision grinding vary from 5,500 to 9,500 fpm. You can change wheel speed by changing the spindle speed or by using a larger or smaller wheel. To find the wheel speed in fpm, multiply the spindle speed (rpm) by the wheel circumference (inches) and divide the product by 12.

$$\text{fpm} = \frac{(\text{circ. X rpm})}{12}$$

$$= \frac{\pi \text{ X D X rpm}}{12}$$

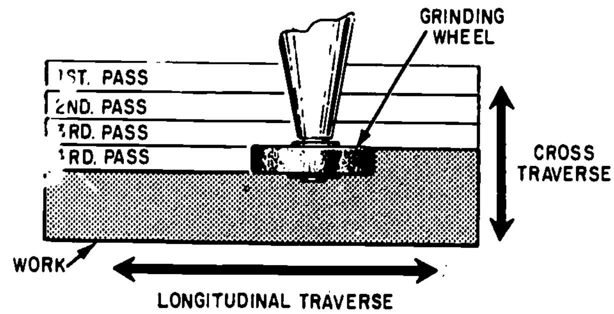
The maximum speed listed on grinding wheels is not necessarily the speed at which the wheel will cut best. The maximum speed is based on the strength of the wheel and allows for a margin of safety. Usually, the wheel will have better cutting action at a lower speed than that listed by a manufacturer as a maximum speed.

One method of determining the proper wheel speed is to set the wheel speed at approximately .000 rpm. Take a trial cut. If the wheel acts too soft (wears away too fast), increase the speed. If the wheel acts too hard (slides over the work or overheats the work), decrease the speed.

TRAVERSE (WORK SPEED)

During the surface grinding process, the work moves in two directions. As a flat workpiece is being ground (fig. 13-1), it moves under the grinding wheel from left to right (longitudinal traverse). The speed at which the work moves longitudinally is called work speed. The work also moves gradually from front to rear (cross traverse), but this movement occurs at the end of each stroke and does not affect the work speed. The method for setting cross traverse is discussed later in this chapter.

A cylindrical workpiece is ground in a manner similar to the finishing process used on a lathe (fig. 13-2). As the surface of the cylinder rotates under the grinding wheel (longitudinal traverse) the work moves from left to right (cross traverse).



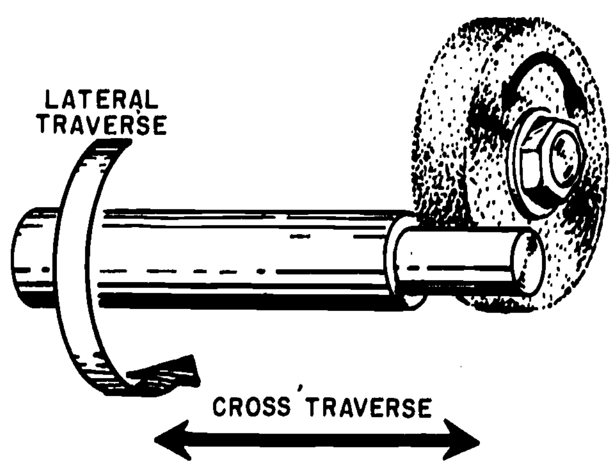
28.427
Figure 13-1.—Surface grinding a flat workpiece.

To select the proper work speed, take a cut with the work speed set at 50 feet per minute. If the wheel acts too soft, decrease the work speed. If the wheel acts too hard, increase the work speed.

Wheel speed and work speed are closely related. Usually by adjusting one or both, you can obtain the most suitable combination for efficient grinding.

DEPTH OF CUT

The depth of cut depends on such factors as material, heat treatment, wheel and work speed, and condition of the machine. Roughing cuts



28.428
Figure 13-2.—Surface grinding a cylindrical workpiece.

should be as heavy as the machine can take; finishing cuts are usually 0.0005 inch or less. For rough grinding, you might use a 0.003-inch depth of cut and then, after a trial cut, adjust the machine until you obtain the best cutting action.

COOLANTS

The cutting fluids used in grinding operations are the same types of fluids used in other machine tool operations. They consist of water, water and soluble oil, water solutions of soda compounds, mineral oils, paste compounds, and synthetic compounds. They also serve the same purposes as in other machine tool operations plus some additional purposes. As in most machining operations, the coolant helps to maintain a uniform temperature between the tool and the work, thus preventing extreme localized heating. In grinding work, excessive heat will damage the edges of cutters, cause warpage, or possibly cause inaccurate measurements.

In other machine tool operations, the chips will fall aside and present no great problem; this is not true in grinding work. If no means is provided for removing grinding chips, they can become embedded in the face of the wheel. This embedding, or loading, will cause unsatisfactory grinding and you will need to dress the wheel frequently. A sufficient volume of cutting fluid will help prevent loading. Other uses of the cutting fluid are to reduce friction between the wheel and the work and to help produce a good finish.

In most other machining operations, the primary factor of a cutting fluid is its lubricating properties. In grinding, however, the primary factor is the cooling property, with the lubricating property second in importance. For this reason, water is the best possible grinding coolant, but if used alone, it will rust the machine parts and the work. Generally, when you use water, you must add a rust inhibitor. The rust inhibitor has very little effect on the cooling properties of the water.

A water and soluble oil mixture gives very satisfactory cooling results and also improves the lubricating properties of the cutting fluid. The addition of the soluble oil to water will alter the grinding effect to a certain extent. Soluble oil decreases the tendency of the machine and the work to rust, thereby eliminating the need for a rust inhibitor. When you prepare a mixture of soluble oil and water as a grinding coolant, use a ratio of three parts water to one part of oil. This mixture will generally prove to be satisfactory.

The paste compounds are made of soaps of either soda or potash, mixed with a light mineral oil and water to form an emulsion. As a coolant, these solutions are satisfactory. However, they have a tendency to retain the grinding chips and abrasive particles, which may cause unsatisfactory finishes on the work.

Mineral oils are used primarily for work where tolerances are extremely small or in such work as thread grinding, gear grinding, and crush form grinding. The mineral oils do not have as great a cooling capacity as water. However, the wheel face will not load as readily with mineral oils as with most of the other coolants. This factor allows you to select a finer grit wheel and requires fewer wheel dressings.

When selecting a cutting fluid for a grinding operation, consider the following characteristics:

- It should have a high cooling capacity to reduce cutting temperature.
- It should prevent adhesion between the work and the chip material.
- It should be suitable for a variety of machine operations on different materials, reducing the number of cutting fluids needed in the shop.
- It should have long life and should not emit obnoxious odors or vapors harmful to personnel.
- It should not cause rust or corrosion.
- It should have a low viscosity to permit gravity separation of impurities and chips collected by the fluid as it is circulated in the cooling system.

- It should not oxidize or form gummy deposits which will clog the circulating system.

- It should be transparent, allowing a clear view of the work.

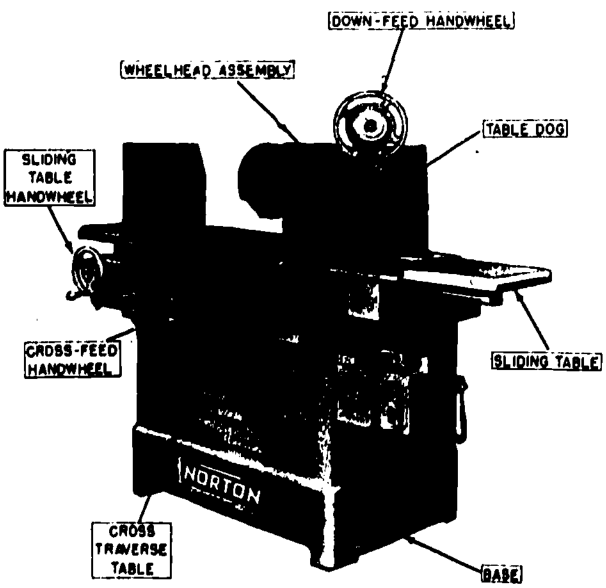
- It should be safe, particularly in regard to fire and accident hazards.

- It should not cause skin irritation.

The principles discussed above are basic to precision grinding machines. You should keep these principles in mind as you study about the machines in the remainder of this chapter.

SURFACE GRINDER

Most of the features of the surface grinder shown in figure 13-3 are common to all planer type surface grinders. The basic components of this machine are a base, a cross traverse table, a sliding worktable, and a wheelhead. Various controls and handwheels are used for controlling the movement of the machine during the grinding operation.



28.249X

Figure 13-3.—Surface grinder (planer type).

BASE

The base is a heavy casting which houses the wheelhead motor, the hydraulic power feed unit, and the coolant system. Ways on top of the base are for mounting the cross traverse table; vertical ways on the back of the base are for mounting the wheelhead unit.

The hydraulic power unit includes a motor, a pump, and piping to provide hydraulic pressure to the power feed mechanisms on the cross traverse and sliding tables. The smooth, direct power provided by the hydraulic unit is very advantageous in grinding. The piping from this unit is usually connected to power cylinders under the traverse tables. When the machine is operating automatically, control valves divert pressurized hydraulic fluid to the proper cylinder, causing the table to move in the desired direction. Suitable bypass and control valves in the hydraulic system let you stop the traverse tables in any position and regulate the speed of movement of the tables within limits. These valves provide a constant pressure in the hydraulic system so that you do not need to secure the system to stop the feed.

CROSS TRAVERSE TABLE

The ways on which the cross traverse table are mounted are parallel to the spindle of the wheelhead unit. These ways guide the cross traverse table in a path of movement parallel to the spindle. This allows the entire width of the workpiece to be traversed under the grinding wheel.

The piston in a power cylinder is fastened to the cross traverse table to provide power feed. Manual feed (by means of a handwheel attached to a feed screw) is also available. The amount of cross traverse feed per stroke of the reciprocating sliding table is determined by the thickness (width) of the grinding wheel. During roughing cuts, the work should traverse slightly less than the thickness of the wheel each time it passes under the wheel. For finish cuts, decrease the rate until you obtain the desired finish. When the power feed mechanism is engaged, the cross traverse table feeds only at each end of the stroke of the sliding table; the grinding wheel clears the ends of the workpiece before crossfeed is made, thereby decreasing side thrust on the grinding wheel and preventing a poor surface finish on the ends of the workpiece.

The total distance of cross traverse on grinding machines in shipboard machine shops is usually 12 inches or less. It is not necessary to traverse the full limits for each job. To limit the cross traverse to the width of the work being ground, use the adjustable cross traverse stop dogs which actuate the power cross traverse control valves. These dogs, when properly set, limit the cross traverse to the area being ground and secure the cross traverse when the grinding wheel reaches these limits.

SLIDING TABLE

The sliding table is mounted on ways on the top of the cross traverse table. Recall that the sliding table moves from left to right, carrying the workpiece under the grinding wheel.

The top of the sliding table has T-slots machined in it so that work or workholding devices (such as magnetic chucks or vises) can be clamped onto the table. The sliding table may be traversed manually or by power.

The arrangement for power feed of the table is similar to the cross traverse table. During manual traverse, a pinion turned by a handwheel engages a rack attached to the bottom of the sliding table.

Table stop dogs are provided to limit the length of stroke during manual operation of the sliding table. Table reverse dogs reverse the direction of movement of the table (when power feed is used) at each end of the stroke. The reverse dogs actuate the control valve to shift the hydraulic feed pressure from one end of the power cylinder to the other.

The rate of speed of the sliding table, given in feet per minute (fpm), can usually be adjusted within a wide range to give the most suitable speed for grinding.

WHEELHEAD

The wheelhead carries the motor-driven grinding wheel spindle. You can adjust the wheelhead vertically to feed the grinding wheel into the work. The adjustment is a lead screw type of mechanism similar to that used on the cross traverse table. A graduated collar on the handwheel lets you keep track of the depth of cut.

The wheelhead is not usually power fed because the depth of cut is quite small and any large movement of this part of the machine is needed only in setting up the machine. The adjusting mechanism is quite sensitive and the depth of cut can be adjusted in amounts as small as 0.0001 inch.

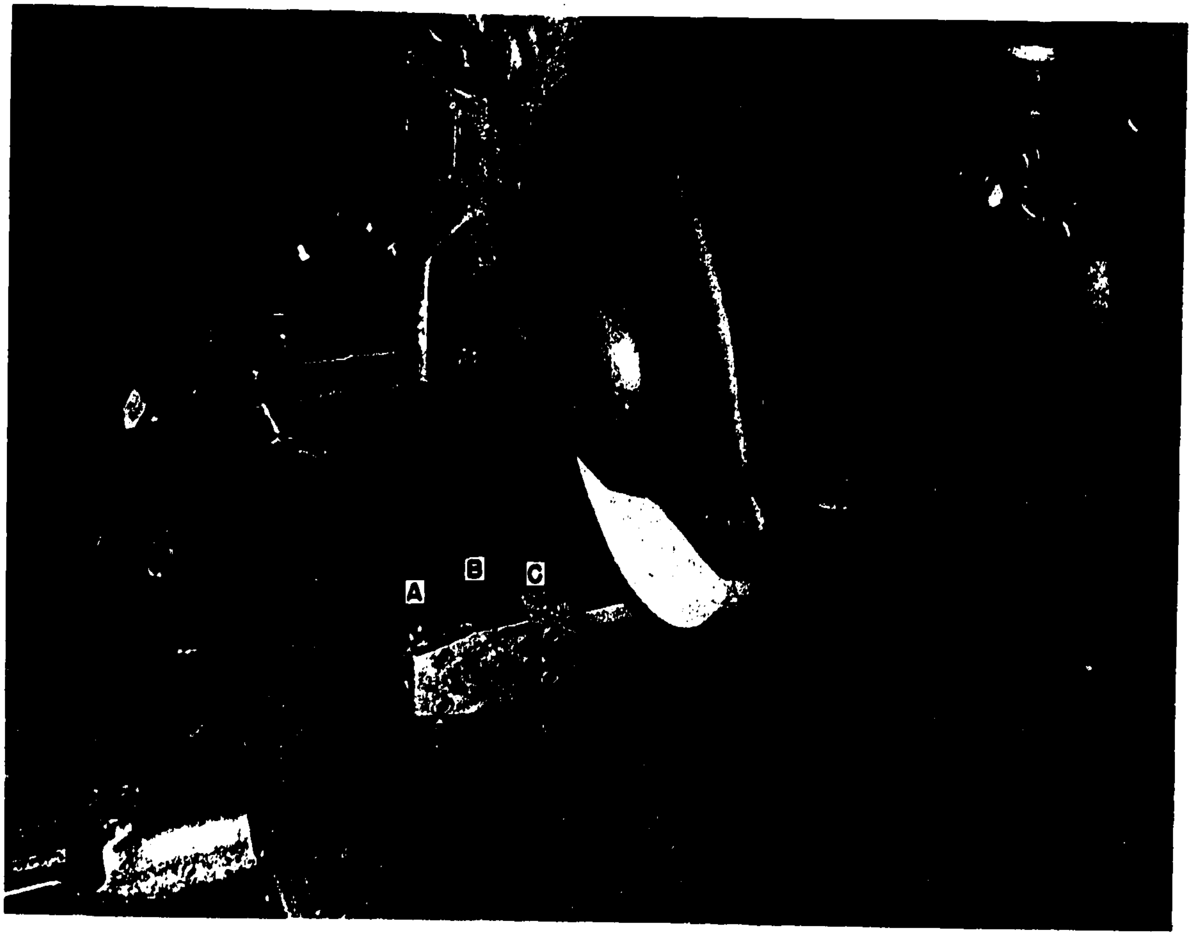
WORKHOLDING DEVICES

As surface grinding is usually done on flat workpieces, most surface grinders have magnetic chucks. These chucks are simple to use; the work can be mounted directly on the chuck or on angle plates, parallels, or other devices mounted on the chuck. Nonmagnetic materials cannot be held in the magnetic chuck unless special setups are used.

The universal vise is usually used when complex angles must be ground on a workpiece. The vise may be mounted directly on the worktable of the grinder or on the magnetic chuck.

Magnetic Chucks

The top of a magnetic chuck (see fig. 13-4) is a series of magnetic poles separated by nonmagnetic materials. The magnetism of the chuck may be induced by permanent magnets or by electricity. In a permanent type magnetic chuck, the chuck control lever positions a series of small magnets inside the chuck to hold the work. In an electromagnetic chuck, electric current induces magnetism in the chuck; the control lever is an electric switch. Work will not



28.260
Figure 13-4.—Magnetic chuck used for holding a tool grinding jig.

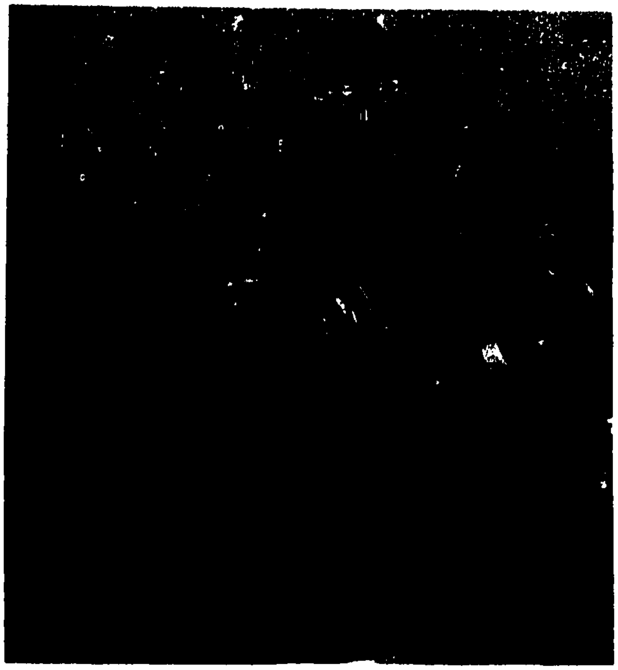
remain in place unless it is in contact with at least two poles of the chuck.

Work held in a magnetic chuck may become magnetized during the grinding operation. This is not usually desirable and the work should be demagnetized. Most modern magnetic chucks are equipped with demagnetizers.

Magnetic chuck will become worn and scratched after repeated use and will not produce the accurate results normally required of a grinder. You can remove small burrs by hand stoning with a fine grade oilstone. But you must regrind the chuck to remove low spots from wear and deep scratches. If the chuck is ever removed from the grinder, its table should be ground when it is replaced to ensure that it is parallel with the grinder table. To grind the table, use a soft grade wheel with a grit size of about 46. Feed the chuck slowly with a depth of cut not to exceed 0.001 or 0.002 inch. Use ample coolant to help reduce heat and flush away the grinding chips.

Universal Vise

The universal vise (fig. 13-5) can be used for setting up work, such as lathe tools, so that the surface to be ground can be positioned at any angle. The swivels can be rotated through 360°. The base swivel (A of fig. 13-5) can be rotated in



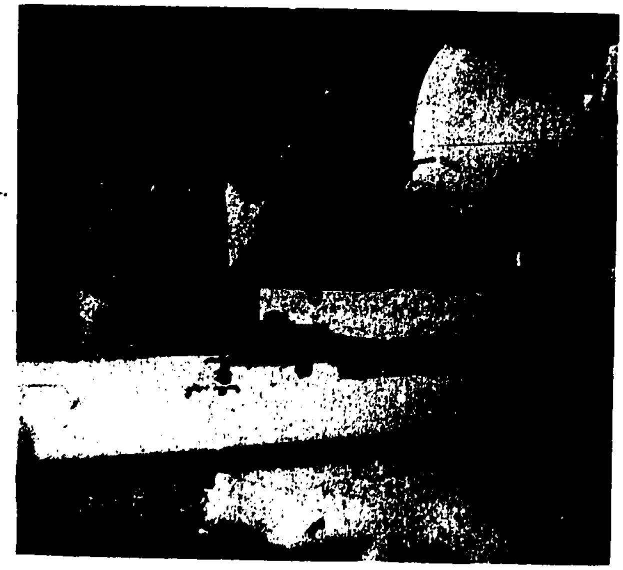
28.251X
Figure 13-5.—Universal vise (mounted on a tool and cutter grinder). (A) Base swivel; (B) Intermediate swivel; (C) Vise swivel.

a horizontal plane; the intermediate swivel (B of fig. 13-5) can be rotated in a vertical plane; the vise swivel (C of fig. 13-5) can be rotated in either a vertical or a horizontal plane depending on the position of the intermediate swivel.

USING THE SURFACE GRINDER

To grind a hardened steel spacer similar to that shown on the magnetic chuck in figure 13-6, proceed as follows:

1. Place the workpiece on the magnetic chuck. Move the chuck lever to the position that energizes the magnetic field.
2. Select and mount an appropriate grinding wheel. This job requires a straight type wheel with a designation similar to A60F12V.
3. Set the table stop dogs so that the sliding table will traverse the work clear of the wheel at each end of the stroke. If power traverse is to be used, set the table reverse dogs.



28.256
Figure 13-6.—Grinding a spacer on a surface grinder.

4. Set the longitudinal traverse speed of the worktable. For rough grinding hardened steel, use a speed of about 25 fpm; for finishing, use 40 fpm.

5. Set the cross traverse mechanism so that the table moves under the wheel a distance slightly less than the width of the wheel after each pass. (Refer to the manufacturer's technical manual for specific procedures for steps 4 and 5.)

6. Start the spindle motor; let the machine run for a few minutes and then dress the wheel.

7. Feed the moving wheel down until it just touches the work surface; then move the work clear of the wheel, using the manual cross traverse handwheel. Set the graduate feed collar on zero to keep track of how much you feed the wheel into the work.

8. Feed the wheel down about 0.002 inch and engage the longitudinal power traverse. Using the cross traverse handwheel, bring the grinding wheel into contact with the edge of the workpiece.

9. Engage the power cross traverse and let the wheel grind across the surface of the workpiece. Carefully note the cutting action to determine if you need to adjust the wheel speed or work speed.

10. Stop the longitudinal and cross traverse and check the workpiece.

Figure 13-5 shows a universal vise being used on a tool and cutter grinder in grinding a lathe tool bit. In this particular application, the base swivel (A) is set to the required side cutting edge angle, the intermediate swivel (B) is set to the side clearance angle, and the vise swivel (C) is set so that the vise jaws are parallel to the table. A cup type wheel is then used to grind the side of the tool. The universal vise is reset to cut the end and top of the tool after the side is ground.

The universal vise can also be used on a surface grinder for very accurate grinding of lathe cutting tools such as threading tools. For example, to grind an Acme threading tool, set the vise swivel at 14 1/2° from parallel to the table. Set the intermediate swivel to the clearance angle. Set the base swivel so that the tool blank (held in the vise jaws) is parallel to the spindle of the grinder. Remember to leave the tool blank extending far enough out of the end of the vise jaws to prevent the grinding wheel from hitting the vise. After grinding one side of the tool bit, turn the tool bit one-half turn in the vise and set the intermediate swivel to an equal but opposite angle to the angle set for the first side. This setting will result in a clearance equal to the clearance of the first side.

Another method for grinding single point tools is to hold the tool in a special jig as illustrated in figure 13-4. The jig surfaces are cut at the angles necessary to hold the tool so that the angles of the tool bit are formed properly.

When using either method for grinding tool bits, check the tool bit occasionally with an appropriate gage until you have obtained the correct dimensions. To save time, you can rough grind the tool bit to approximate size on a bench grinder before setting the tool bit in the jib.

CYLINDRICAL GRINDER

The cylindrical grinder is used for grinding work such as round shafts. Although many of the construction features of the cylindrical grinder are similar to those of surface grinders, there is a considerable difference in the functions of the components. Cylindrical grinders have no cross traverse table. An additional piece of equipment (the workhead) is mounted on the sliding table, and the wheelhead spindle is parallel to the sliding table. See figure 13-7.

As in the surface grinder, the base of this machine contains a hydraulic power unit and a coolant system. Longitudinal ways are for the sliding table. Horizontal ways (at right angles to the longitudinal ways) are provided to permit movement of the wheelhead toward or away from the workpiece. This horizontal movement is used for feeding the grinding wheel into the work for a depth of cut.

SLIDING TABLE

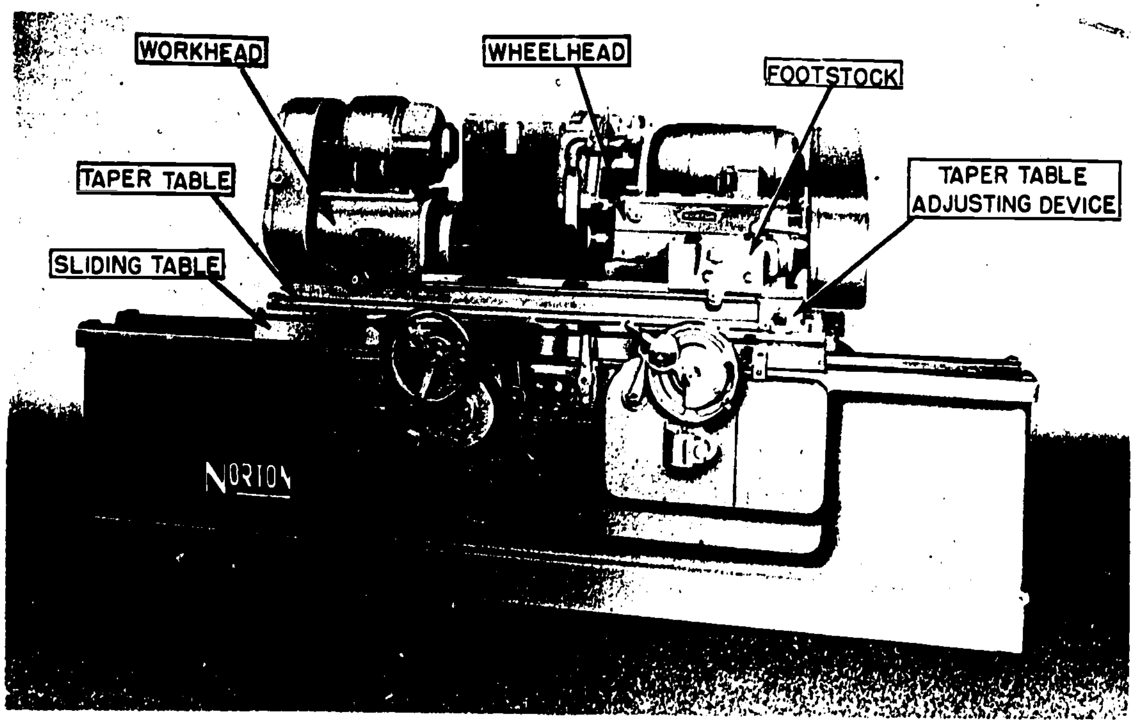
The sliding table of the cylindrical grinder is mounted directly on longitudinal ways on the base of the grinder. This table moves back and forth to traverse the work longitudinally along the width of the grinding wheel.

An adjustable taper table, located on top of the sliding table, is used for grinding long (small angle) tapers on the workpiece. The taper table is adjusted like the taper attachment on a lathe. Workholding devices are clamped on top of the taper table.

The motor-driven workhead is mounted on the taper table. This component holds and rotates the work during the grinding cut. Variable speed drive motors or step pulleys are provided for changing the rate of rotating speed of the workpiece to meet the requirements of the job.

A chuck, a center, or a faceplate can be used in mounting work on the workhead. Center rests and steady rests are also used in conjunction with the workhead for mounting long workpieces for cylindrical grinding.

On most cylindrical grinders used by the Navy, the workhead is mounted on a swivel base



28.252X

Figure 13-7.–Cylindrical grinder (with workhead and footstock mounted).

to provide a means of setting the work for grinding relatively large taper angles.

WHEELHEAD

The wheelhead of a cylindrical grinder moves on horizontal ways (platen). Since cylindrical grinding is done with the axis of the spindle level with the center of the work, no vertical movement of the wheelhead is necessary. Some wheelheads are mounted on swivel bases to provide versatility in taper and angle grinding setups.

USING THE CYLINDRICAL GRINDER

The methods used for setting up stock in a cylindrical grinder are similar to the methods used for lathe setups. Work to be ground between centers is usually machined to approximate size between centers on a lathe. The same center holes are then used for the grinding setup. Center rests or steady rests (as applicable) are used to support long work or overhanging ends. Short workpieces can be held in chucks. For internal grinding (on machines that have an internal grinding spindle), the work is held in a chuck; steady rests are used, if necessary, for support.

To set up a workpiece for grinding between centers proceed as follows:

1. Ensure that the centers in the workhead and footstock and the center holes in the workpiece are in good condition.
2. Clamp a driving dog onto the workpiece.
3. Position the workhead and footstock and set the traverse stop dogs so that when the workpiece is in place, the table will traverse (longitudinally) the proper distance to grind the surface.
4. Ensure that the workhead swivel, the taper table attachment, and the wheelhead

swivel are set properly for straight cylindrical grinding (or for the taper or angle required if an angle or taper is being ground).

5. Adjust the workhead speed mechanism to get the proper rotational speed. A slow speed is usually used for roughing, while a high speed is used for finishing.

6. Set the longitudinal traverse speed so that the work advances from 2/3 to 3/4 the thickness of the wheel during each revolution of the workpiece. Fast traverse feed is used for roughing and a slow feed is used for finishing.

7. Set the workpiece in place and clamp the footstock spindle after ensuring that both centers are seated properly and that the driving dog is not binding.

8. Select and mount the grinding wheel.

9. Start the spindle motor, hydraulic power pump, and coolant pump. After the machine has run for a few minutes, start the coolant flow and dress the wheel.

10. Using the cross traverse mechanism, bring the wheel up to the workpiece and traverse the table longitudinally by hand to see that the wheel will travel through the cycle without hitting any projections. (About one-half of the wheel width should remain on the work at each end of the longitudinal traverse stroke.) Clamp the table dogs in the correct positions to limit longitudinal traverse.

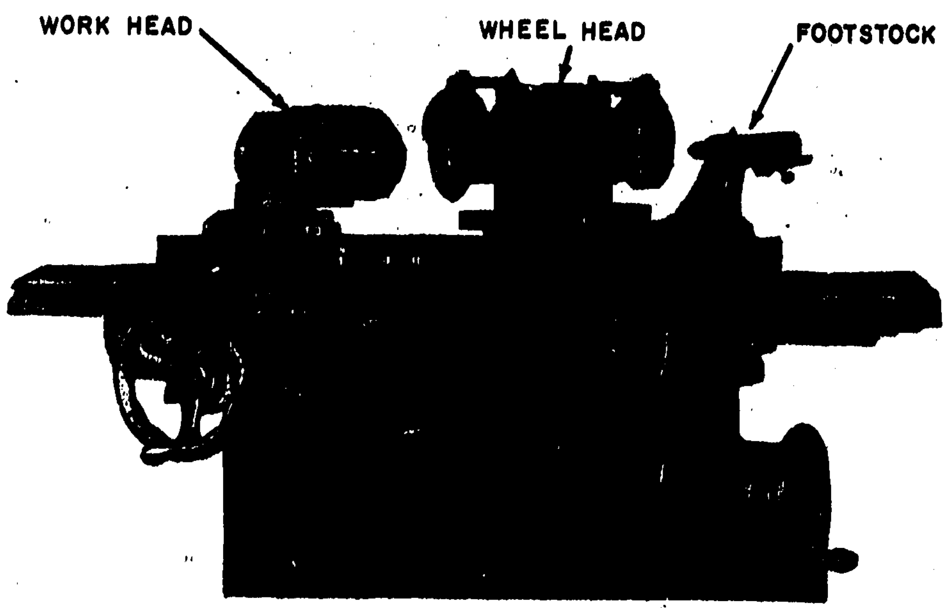
11. Start the workhead motor and feed the grinding wheel in sufficiently to make a cleanup cut (a light cut the entire length of the surface to be ground).

12. Using power longitudinal traverse, take a cut. Then disengage the power traverse, stop the workhead motor and wheelhead rotation, and check the workpiece for taper. Make any changes required. (If an adjustment of the taper table attachment is necessary at this point the wheel should be dressed again).

No information is given here on methods of setting the various controls and speeds as there is a variation in specific methods for each machine. The manufacturer's technical manual for the machine you are to use provides this information.

TOOL AND CUTTER GRINDER

A tool and cutter grinder (fig. 13-8) can best be described as having a combination of the



28.253

Figure 13-8.—Tool and cutter grinder (workhead and footstock).

features of the plain cylindrical and the planer type surface grinders. A tool and cutter grinder is used primarily for grinding multi-edged cutting tools such as milling cutters, reamers, and taps. The worktable has the same basic construction features as the surface grinder, but a taper table is mounted on the sliding table so you can grind tools that have small tapers such as tapered reamers.

WHEELHEAD

The wheelhead is adjustable in two directions. It can be moved vertically on its support column and will rotate through 360°. If you need to change the rotational direction of the grinding wheel, simply rotate the wheelhead 180°. Additionally, the spindle is double ended, allowing you to mount two wheels on the wheelhead.

WORKHEAD

The basic workholding devices used on the tool and cutter grinder are the workhead and the footstock (fig. 13-8). When a workhead is not provided, you can use a left-hand footstock similar to the right-hand footstock shown mounted on the table in figure 13-8. Also, a variety of tooth rests (for supporting and guiding the teeth of a cutter being sharpened) are usually provided.

A distinctive feature of most tool and cutter grinders is that there are control handwheels at both the back and the front of the machine. The dual controls permit you to stand in the most convenient position to view the work and still operate the machine. You can usually disengage the sliding table handwheel to push the table back and forth by hand. Graduated collars on the handwheels are a quick visible guide to indicate the amount of movement of the various feed components.

CUTTER SHARPENING

The working efficiency of a cutter is largely determined by the keenness of its cutting edge. Consequently, a cutter must be sharpened at the first sign of dullness. A dull cutter not only leaves a poorly finished surface, but also may be damaged beyond repair if you continue to use the cutter in this condition. A good rule for determining when to sharpen a cutter is to sharpen it when the wear land on the cutting edge is between 0.010 to 0.035 inch. Sharpening cutters at the first sign of dullness is both economical and a sign of good workmanship.

Cutters to be sharpened may be divided into two groups: (1) those that are sharpened on the relief and (2) those that are sharpened on the face. In the first group are such cutters as plain milling, side milling, stagger tooth, angle cutters, and end mills. In the second group are the various form cutters such as involute gear cutters and taps. The relief on the second type of cutter is provided when it is manufactured; the faces of the teeth are ground to sharpen them.

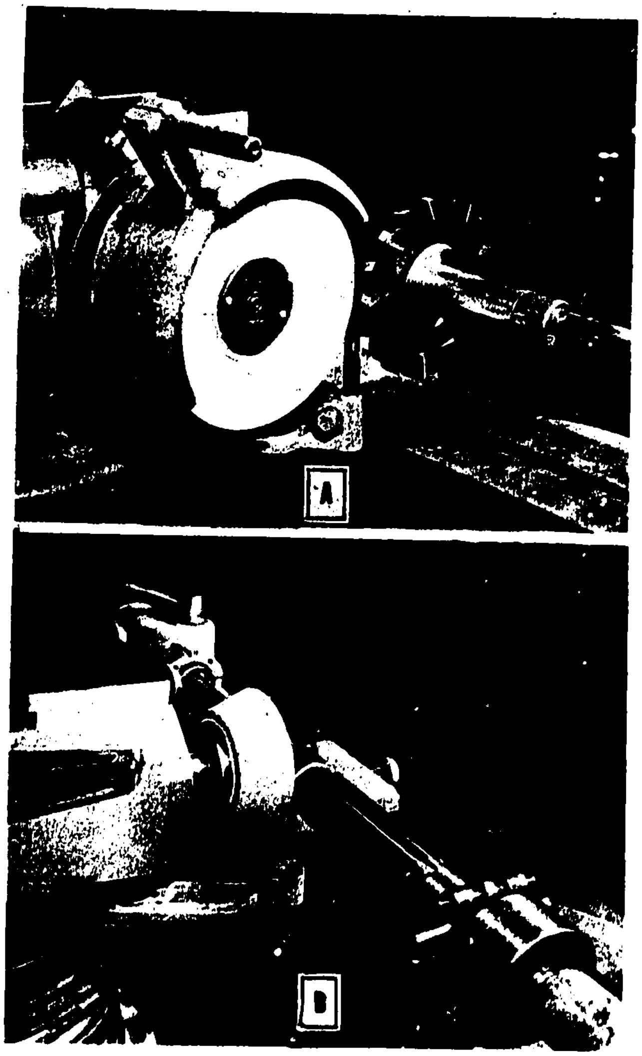
Figure 13-9 shows two methods for grinding cylindrical cutting tools on a tool and cutter grinder. Part A of figure 13-9 shows a setup for grinding a staggered tooth cutter using a straight wheel. Part B of figure 13-9 shows a setup for grinding a reamer using a cup type wheel. Either type of wheel can be used; the cup type wheel produces a straight clearance angle; the straight wheel produces a hollow ground clearance angle.

When you use the straight wheel, set the spindle parallel to the table. When you use a flaring cup wheel, turn the spindle at an angle of 89° to the table. This provides the necessary clearance for the trailing edge of the grinding wheel as it is traversed along the cutter.

When you grind a cutter, you should have the grinding wheel rotating as shown in B of figure 13-10. This method tends to keep the tooth of the cutter firmly against the tooth rest, ensuring a correct cutting edge. If this method causes too much burring on the cutting edge, you may reverse the direction of wheel rotation as shown in A of figure 13-10. If you use the latter method, ensure that the tooth being ground rests firmly on the tooth rest during the cut.

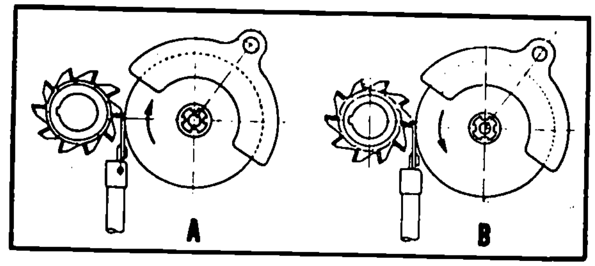
Dressing and Truing

Sharpening a high-speed steel cutter or reamer generally requires a soft grade wheel. A soft grade wheel breaks down easily and is



28.253A
Figure 13-9.—Tool grinding setups on a tool and cutter grinder: (A) Straight wheel grinding a milling cutter; (B) Cup wheel grinding a reamer.

therefore less likely to burn the cutter. You should true and dress the wheel prior to starting the sharpening operation and then re-dress as necessary, depending on the amount of wheel wear. As you grind each cutter tooth, the grinding wheel diameter decreases because of wear. The grinding machine does not automatically adjust itself for wear of the wheel. As a result, succeeding teeth have less metal removed and the teeth gradually increase in size.



28.257X
Figure 13-10.—Direction of wheel rotation: (A) Toward cutting edge; (B) Away from cutting edge.

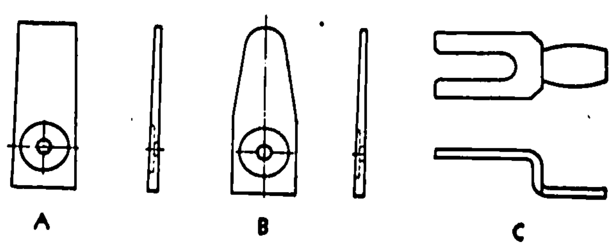
To compensate for wheel wear and to ensure that all the teeth are the same size, rotate the cutter 180° and grind all the teeth again. Be careful not to grind the cutter undersize.

To ensure a good cutting edge on the cutter, there must be a good finish on the clearance angle; therefore, you will occasionally need to dress the grinding wheel. The wheel truing attachment is used for this operation and for the initial truing and dressing operation on the wheel.

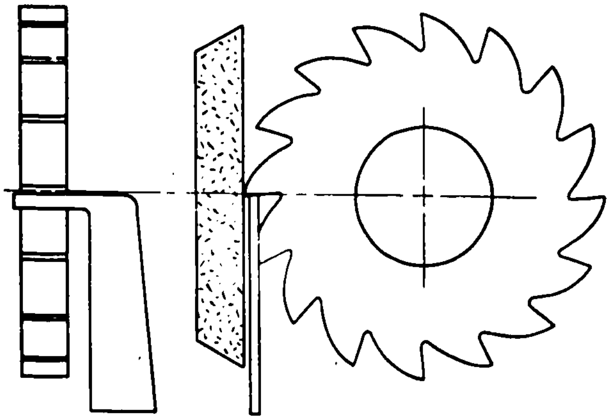
Tooth Rest Blades and Holders

Tooth rest blades are not carried in stock, so they must be made in the shop. You will find that once you understand the requirements for the blades, you will be able to readily fabricate various shapes to suit the types of cutters that you will be called upon to sharpen. It is normally recommended that these blades be made of spring steel.

The plain (straight) tooth rest blade (A in fig. 13-11) is used for sharpening side milling



126.46X
Figure 13-11.—Typical tooth rest blades.



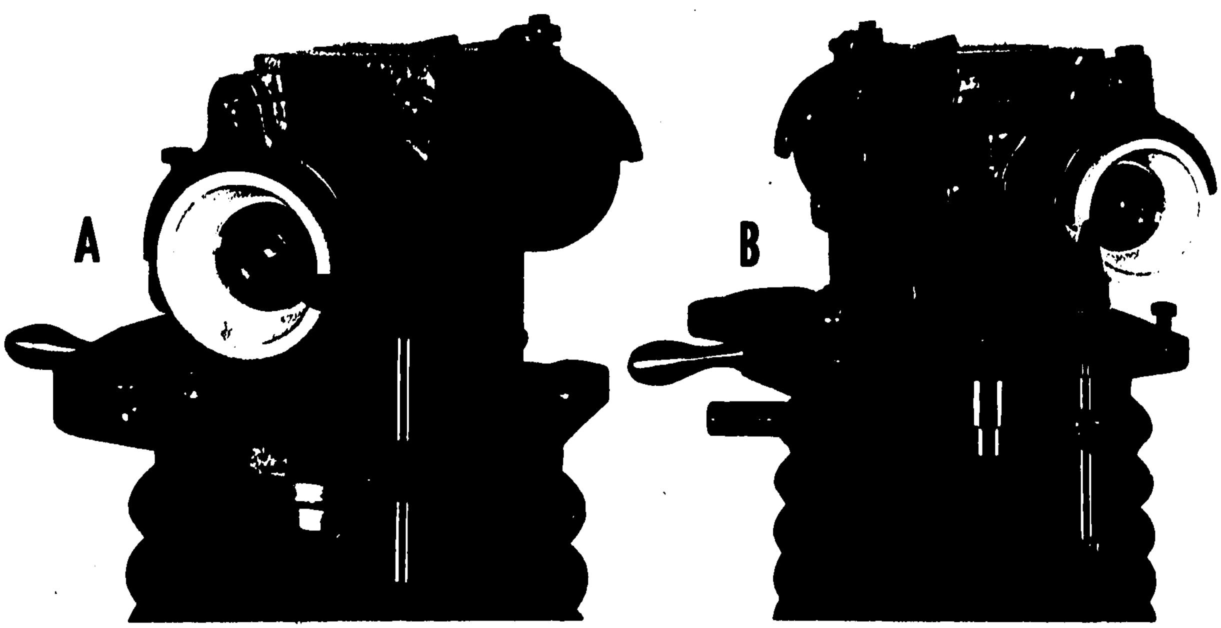
126.47X
Figure 13-12.—L-shaped tooth rest blade.

cutters, end mills, straight-fluted reamers, or any straight-fluted cutter. The rounded tooth rest blade (B in fig. 13-11) is used for cutters with the teeth on a helix, shell end mills, and small end mills. The offset tooth rest blade (C in fig. 13-11) is a universal blade that can be used for most applications. The L-shaped tooth rest blade for sharpening metal slitting saws and straight tooth plain milling cutters with closely spaced teeth is shown in figure 13-12. Other shapes of tooth rest blades may be made to fit the specific type of cutter or the cutter grinder being used.

Holders for the tooth rest blades may be either plain or universal. Figure 13-13A shows a tooth rest blade in plain holder and figure 13-13B shows a tooth rest blade mounted in a universal type holder. The universal tooth rest holder has a micrometer adjustment at its bottom to enable you to make precise up and down movements in the final positioning of the blade.

SETTING THE CLEARANCE ANGLE

Correct clearance back of the cutting edge of any tool is essential. With insufficient clearance, the teeth will drag, producing friction and slow cutting. Too much clearance produces chatter and dulls the teeth rapidly. The cutting edge must have strength, and the correct clearance



126.48X
Figure 13-13.—A. Tooth rest blade in plain holder; B. Tooth rest blade in a universal holder.

will provide this strength. Figure 13-14 shows a typical cutter tooth and the angles produced by grinding.

The primary clearance angle is the angle ground when a cutter requires sharpening. The number of degrees in the primary clearance angle varies according to the diameter of the cutter and the material being cut. A large diameter cutter requires less clearance than a small cutter. Cutters of hard materials such as alloy and tool steels require less clearance than cutters of softer materials such as brass and aluminum.

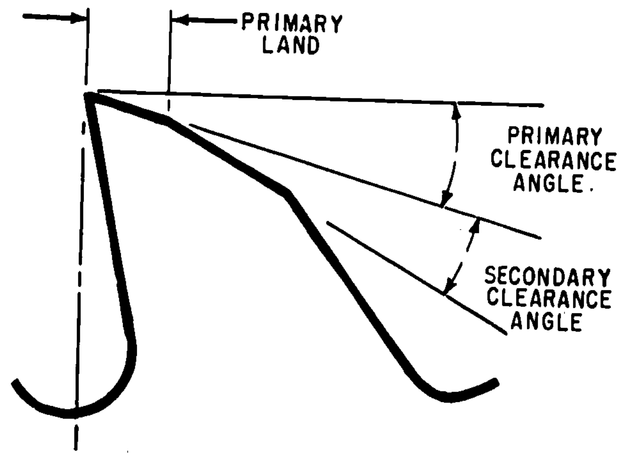
The primary clearance angles range from 4° for a large cutter to 13° for a smaller cutter. Some manufacturers of tool and cutter grinders have charts that can assist you in determining the correct clearance angle. The width of the primary land (the surface created when the primary clearance angle is ground) varies according to the size of the cutter. Primary land widths range from 0.0005-0.015 inch for a small cutter to 0.030-0.062 inch for a large cutter. You should grind the lands very carefully. A land that is too narrow will allow the cutting edge to chip or wear rapidly. A land that is too wide will cause the trailing side of the land (heel) to rub the work.

When the width of the primary land becomes excessive due to repeated grindings, you must grind the secondary clearance angle to reduce it. The secondary is normally 3° to 5° greater than the primary clearance angle.

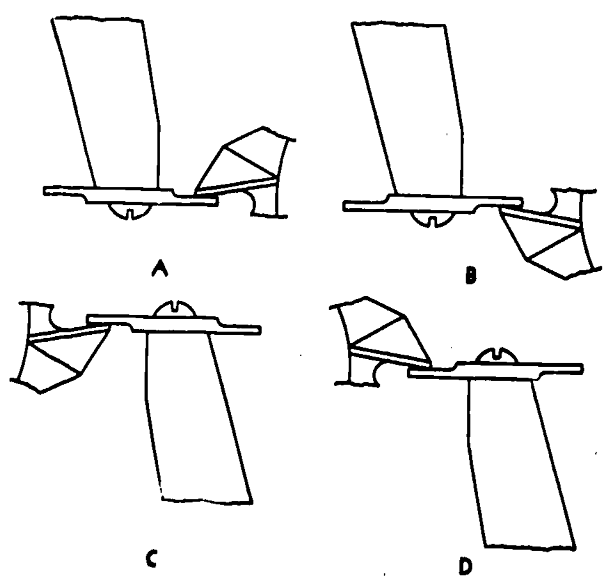
The desired clearance angle is obtained by the positioning of the grinding wheel, the cutter, and the tooth rest. The general procedure is to position the center of the wheel, the center of the work, and the tooth rest all in the same plane and then raise or lower the wheel head the proper distance to give the desired clearance angle.

When using the straight wheel, bring the center of the wheel and the center of the work into the same plane by using the centering gage (fig. 13-15) or by using a height gage. Then, fasten the tooth rest to the machine table and adjust to the same height as the center of the work. Raise or lower the wheelhead a predetermined amount to give the correct clearance angle. Determine the amount to raise or lower the wheelhead by multiplying the clearance angle (in degrees) by the diameter of the wheel (inches) and then multiply this product by the constant 0.0087.

When using a cup wheel, mount the tooth rest on the wheelhead. Position the center of the cutter in the same plane as the tooth rest. Then



28.429
Figure 13-14.—Cutter clearance angles.



126.49X
Figure 13-15.—Centering gage.

raise or lower the wheelhead the proper amount to give the desired clearance. Determine the amount to raise or lower the wheelhead by multiplying the clearance angle (in degrees) by the diameter of the cutter (in inches) and then multiply this product by the constant 0.0087.

Some tool and cutter grinders have a tilting wheelhead or a clearance setting device. Where a tilting wheelhead is provided, simply tilt the wheelhead to the desired clearance angle. If you use a clearance setting device, follow the steps listed below.

1. Clamp a dog to the mandrel on which the cutter is mounted.

2. Insert the pin on the side of the dog into the hole in the clearance setting plate that is mounted on the footstock.

3. Loosen the setscrew in the clearance setting plate and rotate the cutter to the desired setting (graduations found on the clearance setting plate).

4. Tighten the setscrew.

5. Remove the dog.

When you grind the teeth of end mills, side milling cutters, or stagger tooth cutters, use the graduated dials on the workhead to set the clearance angle.

CUTTER SHARPENING SETUPS

Tool and cutter grinders vary in design and in the type of accessory equipment; however, most tool and cutter grinders operate in the same way. By using only the standard workhead, footstocks, and tooth rest blade holders, practically any cutter can be sharpened. Most cutters can be sharpened using essentially the same method. A thorough study of the following sections, along with a little ingenuity and forethought, will enable you to sharpen any cutter that may be sent to your shop for sharpening.

PLAIN MILLING CUTTERS (HELICAL TEETH)

Since essentially the same method may be used for sharpening many cutters, we will discuss in some detail the steps involved in setting up the machine for a specimen cutter. The plain milling cutter with helical teeth will serve very well as an example. The steps are:

1. Remove all accessory equipment from the machine table.

2. Clean the table and the bottoms of the footstocks.

3. Mount the footstocks on the table, allowing just enough space between them to accommodate the mandrel with a slight amount of tension on the spring-loaded center.

4. Swivel the wheelhead to 89°. (This allows the end of the cutter to clear the opposite cutting face when you use a cup type wheel.)

5. Mount the wheel and wheel guard.

6. Thin the cutting face of the wheel to not more than 1/8 inch, using a dressing stick. True the wheel, using a diamond truing device.

7. Using the centering gage, bring the wheelhead axis into the same horizontal plane as the axis of the footstock centers.

8. Mount the cutter on a mandrel. (A knurled sleeve on the end of the mandrel will help you to maintain an even, effective grip while the cutter is being ground.

9. Mount the mandrel between the footstock centers, preferably in such a position that the grinding wheel cuts onto the cutting edge of the teeth.

10. Mount the plain tooth rest holder (with a rounded tooth rest blade) on the wheelhead.

11. With the centering gage on top of the wheelhead and the tip of the gage directly in front of the cutting face of the wheel, adjust the tooth rest blade to gage height. (This brings the blade into the same horizontal plane as the footstock centers.)

12. Traverse the saddle toward the wheelhead until one tooth rests on the tooth rest blade; then lock the table into position.

13. With a cutter tooth resting on the tooth rest, lower the wheelhead until the desired clearance is indicated on the clearance setting plate. If no clearance setting device is available, calculate the distance to lower the wheelhead using the method previously described.

Before starting the sharpening operation, run through it without the machine running. This will let you get the feel of the machine and also ensure that there is nothing to obstruct the grinding operation. Traverse the table with one hand while the other hand holds the cutter against the tooth rest blade. On the return movement, the tooth rest blade will cause the mandrel to turn in your hand, thereby eliminating the necessity of moving the table away from the wheel on the return traverse.

In sharpening the teeth of any milling cutter, grind one tooth, then rotate the cutter 180° and grind another tooth. Check the teeth with a micrometer to ensure that there is no taper being ground. If there is taper, you must remove it by swiveling the swivel table of the machine.

As the width of the land increases with repeated sharpenings, you will need to grind a secondary land on the cutter. Never allow the primary land to become greater than 1/16 inch wide because the heel of the teeth may drag on the work. To control the width of the primary land, double the clearance angle and grind a secondary land.

SIDE MILLING CUTTERS

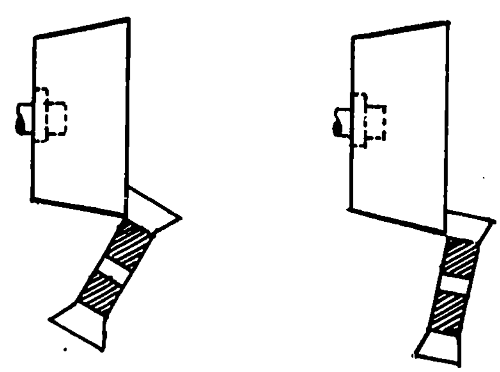
The peripheral teeth of a side milling cutter are ground in exactly the same manner as the teeth of a plain milling cutter, with the exception that a plain tooth rest blade is used.

To sharpen the side teeth, mount the cutter on a stub arbor and clamp the arbor in a universal workhead. Then mount a universal tooth rest holder onto the workhead so that when the workhead is tilted the tooth rest holder moves with it (fig. 13-16).

28.430

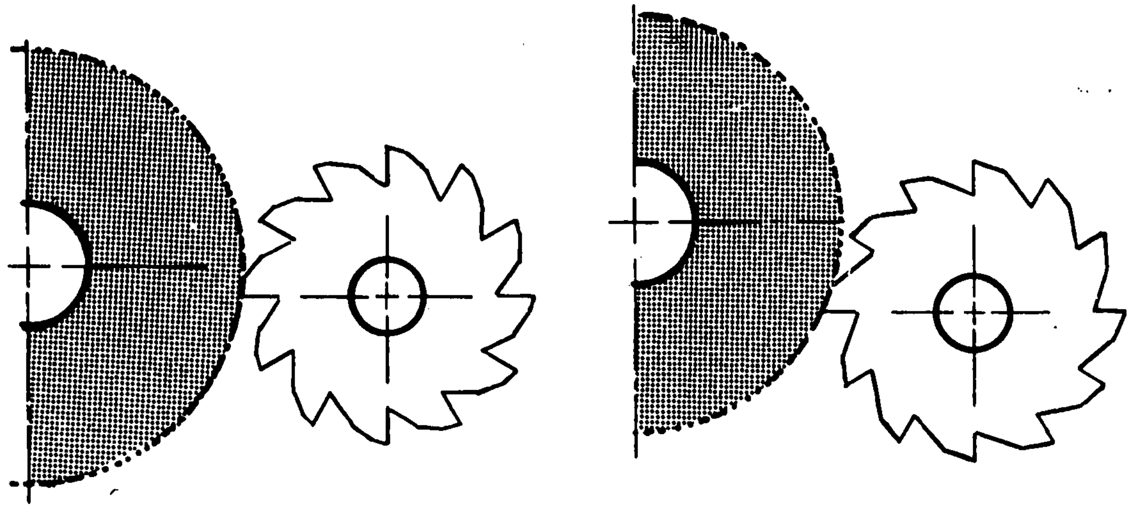
Figure 13-16.—Grinding side teeth of side-milling cutter (Norton Co.).

The procedure for grinding clearance angles varies, depending on the type of grinding wheel used. If you are using a cup wheel, swivel the workhead vertically to move the tooth toward or away from the wheel. The clearance angle increases as the tooth is swivelled away from the wheel. (Fig. 13-17). If you use a straight wheel, set the cutter arbor horizontally and raise or lower the wheel to change the clearance angle. The clearance angle increases as the wheel is raised (fig. 13-18).



28.431

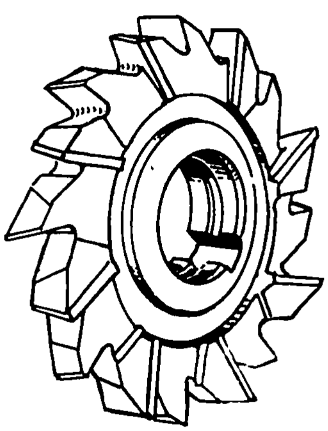
Figure 13-17.—Changing clearance angle by swiveling the cutter in a vertical plane.



28.432
Figure 13-18.—Changing the clearance angle by raising the grinding wheel.

STAGGERED TOOTH CUTTERS

Staggered tooth milling cutters (fig. 13-19) may be sharpened in exactly the same manner as plain milling cutters with helical teeth (fig. 13-20). If you use this method, grind all of the teeth on one side of the cutter. They turn the cutter over and grind all of the teeth on the other side.

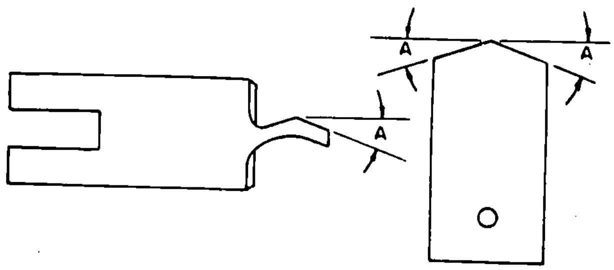


28.433
Figure 13-19.—Staggered-tooth side milling cutter (National Twist Drill).

There is, however, a method for sharpening all of the cutter's teeth in one setting (see setup, fig. 13-9A).

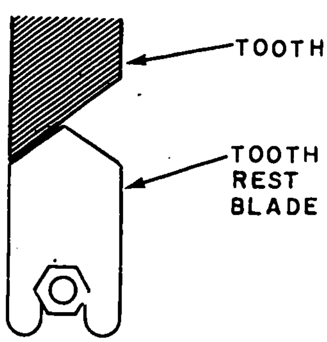
1. Mount the cutter on a mandrel held between centers.

28.434
Figure 13-20.—Tooth rest mounted on wheelhead in grinding a helical-tooth cutter (Brown & Sharpe Mfg. Co.).



126.50
Figure 13-21.—Tooth rest blades for staggered tooth cutters.

2. Fasten the tooth rest holder to the wheelhead.
3. Grind the tool rest blade to the helix angle of the cutter teeth on each side of the blade (fig. 13-21).
4. Position the high point of the tooth rest blade in the center of the cutting face of the wheel.
5. Align the wheelhead shaft centerline, the footstock centers, and the high point of the tooth rest blade in the same horizontal plane.
6. Raise or lower the wheelhead to give the desired clearance angle.
7. Rest the face of a tooth on its corresponding side of the tooth rest blade (fig. 13-22).
8. Move the cutting edge of the tooth across the face of the wheel. On the return cut, rest the next tooth on the opposite angle of the



28.435
Figure 13-22.—Resting the face of a tooth on its corresponding side of the tooth rest blade.

tooth rest. Continue alternating teeth on each pass until you have sharpened all the teeth.

ANGULAR CUTTERS

To sharpen an angular cutter, mount the cutter on a stub arbor and mount the arbor in a universal workhead. Then swivel the workhead on its base to the angle of the cutter. If the cutter has helical teeth, mount the tooth rest on the wheelhead. But if the cutter has straight teeth, mount the tooth rest on the table or on the workhead. To set the clearance angle for both types of teeth, tilt the workhead the required number of degrees toward or away from the grinding wheel. Then use a centering gage to align the cutting edge of one tooth parallel with the cutting face of the wheel. Take a light cut to check your settings and make fine adjustments until you obtain the desired clearance angle.

END MILLS

A damaged end mill may be salvaged by cutting off the damaged portion with a cylindrical grinding attachment, as shown in figure 13-23. When salvaging an end mill in this manner, use a coolant if possible to avoid removing the temper at the end of the cutter. The center of the end must be relieved in the same way as the original cutter.

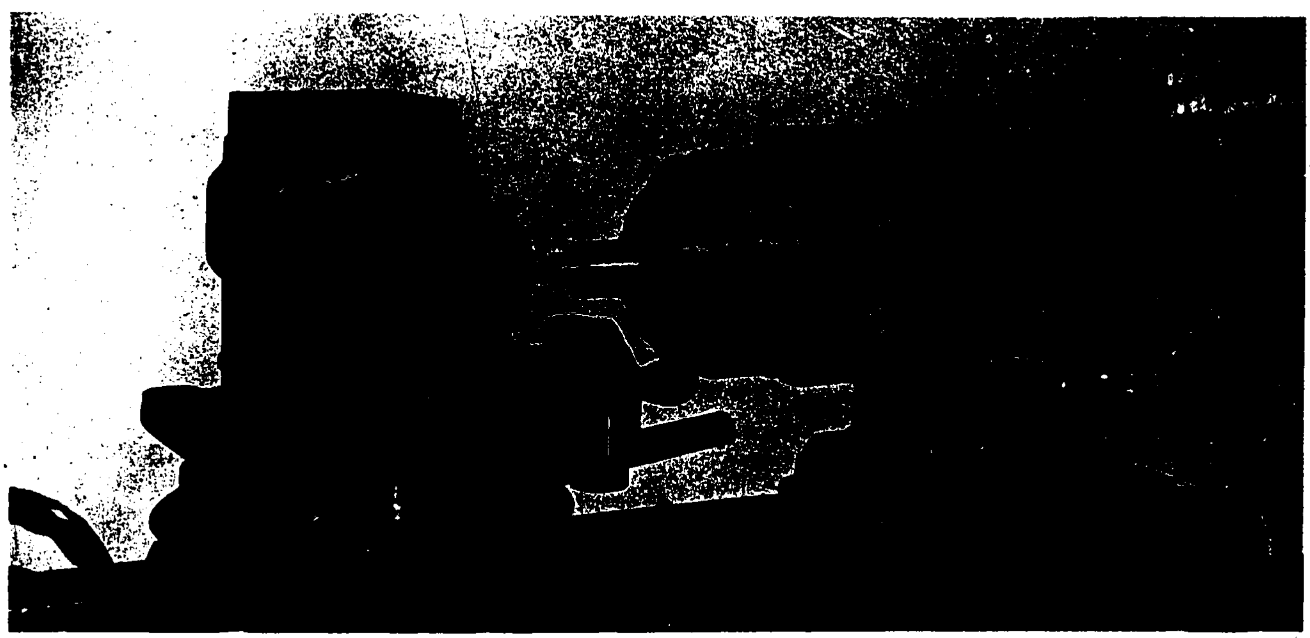
Generally, it will not be necessary to sharpen the peripheral teeth. If, however, the peripheral teeth must be ground, use the same procedure that you would use to sharpen a plain milling cutter except for the method of mounting the cutter. Mount the end mill in a universal workhead (fig. 13-24) instead of between centers. You must remember that whenever you grind the peripheral teeth of an end mill you change the size (diameter) of the cutter. You, therefore, must indicate that the cutter size has been changed. Either mark the new size on the cutter or grind off the old size and leave the cutter unmarked.

Use the following steps to sharpen the end teeth:

1. Mount the mill in a universal workhead.
2. Swivel the wheelhead to 89°.

126.51X

Figure 13-23.—Cutting off the damaged end of a helical end mill.



126.52X

Figure 13-24.—Grinding the peripheral teeth of an end mill.

3. Bring the cutting edge of a tooth into the same horizontal plane as the wheelhead spindle axis by using a centering gage. Place the gage on top of the wheelhead and raise or lower the wheelhead sufficiently to place the blade of the gage on the tooth cutting edge. This will at the same time align the cutting edge with the centerline of the wheel.

4. Lock the workhead spindle in place to prevent the cutter from moving.

5. Clamp the tooth rest blade onto the workhead so that its supporting edge rests against the underside of the tooth to be ground.

6. Swivel the workhead downward to the desired clearance angle and clamp it in position. At this point, make sure that the tooth next to the one being ground will clear the wheel. If it does not, raise or lower the wheelhead until the tooth does clear the wheel.

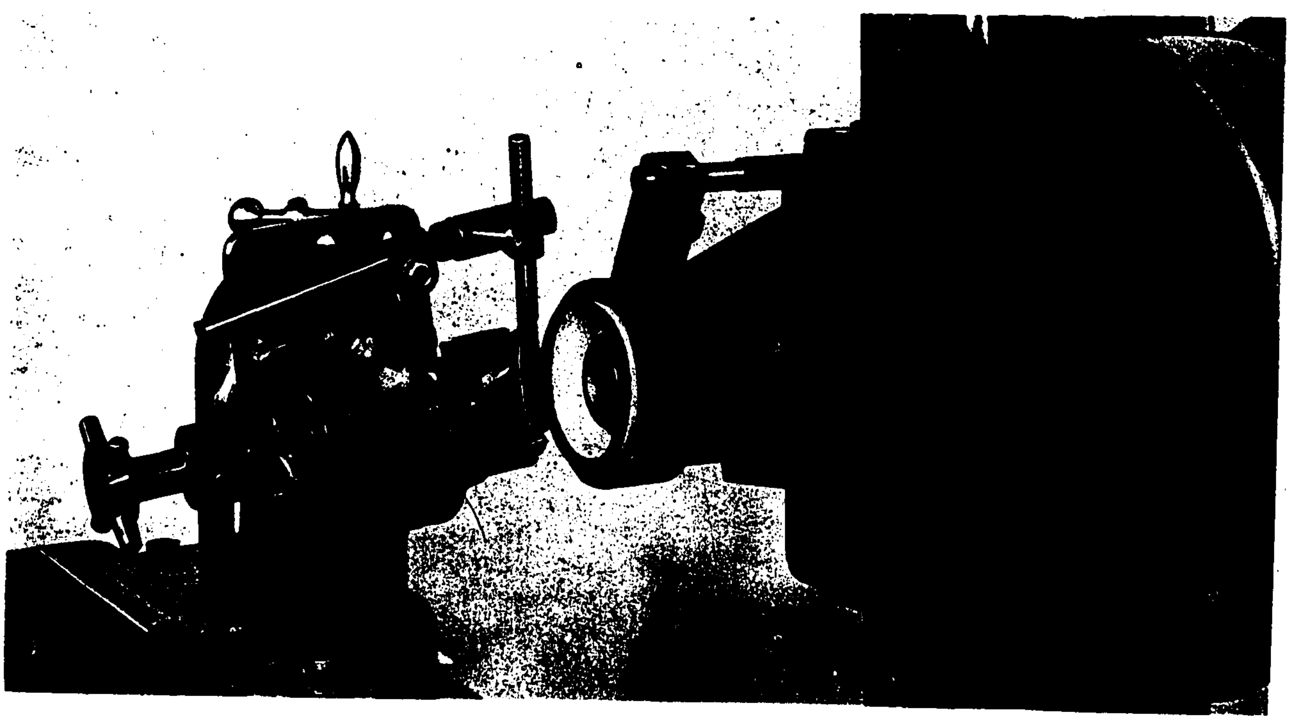
7. Unclamp the workhead spindle and begin grinding the mill.

8. After you have ground all of the primary lands, tilt the workhead to the secondary clearance angle and grind all the secondary lands.

On large diameter wheel end mills, it is often a good idea to back off the faces of the teeth toward the center of the cutter, similar to the teeth of a face mill. An angle of about 3° is sufficient, allowing a land of 3/16 to 5/16 inch long.

It is important that you use as much care when grinding the corners of the teeth as when grinding the face of the peripheral teeth; otherwise, the cutting edges will dull rapidly, and a poor finish will result. The corners of the teeth are usually chamfered 45° by swiveling the workhead or table and are left 1/6 to 1/8 inch wide.

To sharpen the end teeth of a shell end mill (fig. 13-25), mount the cutter on an arbor set in a taper shank mill bushing. Then insert the bushing into the taper shank mill bushing sleeve held in the universal workhead. To obtain the



126.53X

Figure 13-25.—Grinding the end teeth of a shell end mill.

desired clearance angle, swivel the workhead in the vertical plane and swivel it slightly in the horizontal plane to grind the teeth low in the center of the cutter. Turn the cutter until one of the teeth is horizontal; then raise the wheel until that tooth can be ground without interference.

FORMED CUTTERS

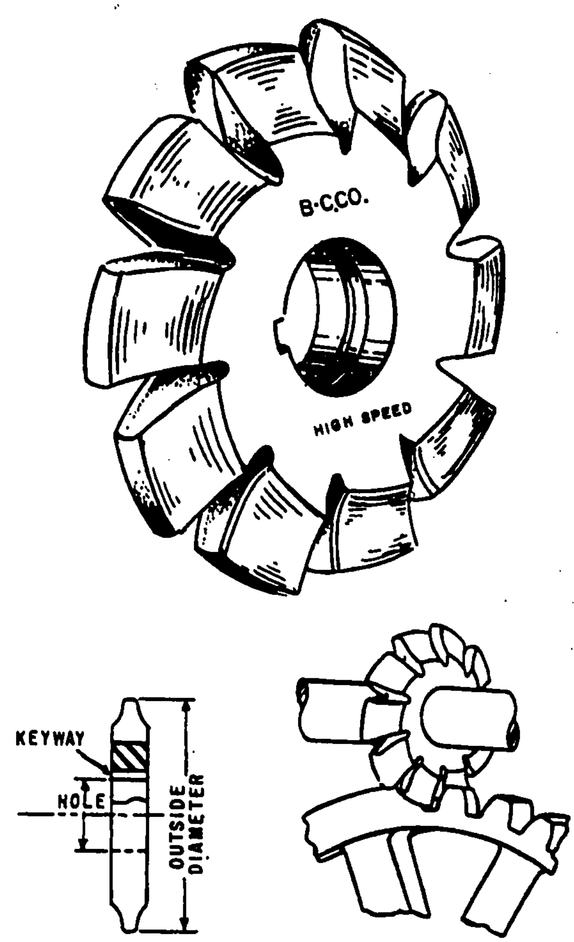
There are two methods commonly used to sharpen formed milling cutters. The first method, using a formed cutter sharpening attachment, is by far the most convenient to use. The second method consists of setting up the cutter on a mandrel, grinding the backs of the teeth and then reversing the cutter to sharpen the cutting face.

The involute gear cutter (fig. 13-26) will serve as an example. Since these cutters are form relieved, the only correct way to sharpen them is to grind the faces of the teeth. This type of cutter is ground radially on the cutting face after the backs of the teeth have been ground. Be sure to grind the backs of the teeth. If the teeth are to be ground uniformly, they must all have the same thickness from the back to the cutting face.

To sharpen a formed cutter using the formed cutter sharpening attachment, attach the wheelhead shaft extension to the shaft and mount a dish-shaped wheel on the extension. With the wheelhead swiveled to 90°, clamp the attachment to the table with the pawl side of the attachment away from the wheel. Place the cutter on the stud in the reverse position, so that the backs of the teeth can be ground.

The backs of the teeth must be in the same plane as the face of the wheel. This is accomplished by placing the centering gage on top of the wheelhead and adjusting the head vertically until the cutter and the gage are about central. Then remove the gage. Adjust the saddle in or out and at the same time rotate the cutter (by hand) on the stud. Place the edge of the pawl on the relief, behind the cutting face of the tooth being ground, and clamp in place by tightening the locking knob.

After grinding the back of the first tooth, index for grinding the next tooth by traversing the table to move the cutter away from the wheel. Lift the cutter off the stud and turn it to the next tooth. Be sure to hold the cutter against the solid pawl with one hand while grinding.



28.436
Figure 13-26.—Involute gear cutter.

Due to the deformations set up in hardening, the amount ground off one tooth may be greater than the amount ground off the next tooth, but after the backs of the teeth have been ground, there will be a uniformity between the backs of the teeth and the outside diameter.

To sharpen the cutting faces of the teeth, reverse the cutter on the stud and line up the cutting face of a tooth with the attachment centering gage. Loosen the pawl locking knob and adjust the pawl to the back of the tooth. Then adjust the saddle to bring the face of the tooth in line with the face of the grinding wheel. Once this adjustment has been made, do not readjust the saddle except to compensate for wheel wear. After grinding one tooth, move the saddle away from the wheel, index to the next tooth, and grind. If, after all teeth have been ground once, the teeth have not been ground enough, rotate the tooth face toward the wheel and make a second cut on each tooth.

If a cutter has been initially provided with a radial rake angle, this angle must be retained or the cutter will not cut the correct form. To sharpen this type of cutter, line up the point of one cutter tooth with the attachment gage, swivel the table to the degree of undercut, adjust the saddle to bring the face of the tooth in line with the face of the wheel, and grind.

If a formed cutter sharpening attachment is not available, formed cutters may be sharpened using a setup similar to that of a plain milling cutter; between centers on a mandrel. Using this method the setup for grinding a radial tooth formed milling cutter and for grinding a tap are essentially the same. We will use a tap in this example.

Grinding a Tap

To grind a tap, use the following steps:

1. Mount the wheelhead shaft extension and the dish wheel on the machine.
2. True the wheel with the diamond truing device.
3. Line up the face of the wheel with the footstock centers. Place a straightedge across the face of the wheel and adjust the saddle toward the wheelhead until the wheel face is centered.
4. Place the tap between centers.
5. Fasten the tooth rest to the table, with the blade against the back of the blade to be ground.
6. Adjust the tap to the wheel with the micrometer adjustment on the tool rest.
7. Grind the tap.

To produce accurate results in grinding taps, grind the backs of the teeth before you grind the faces.

HONES AND HONING

In honing, the cutting is done by abrasive action. Honing may be used to remove stock from a drilled, bored, reamed, or ground hole to correct taper, out-of-roundness, or bow (bell mouthed barrel shape or misalignment). Honing is also used to develop a highly smooth finish while accurately controlling the size of the hole.

You may do cylindrical honing on a honing machine or on some other machine tool by attaching the honing device to the machine spindle, or you may do it by hand. Regardless of the method you use, either the hone or the work must rotate, and the honing tool must move back and forth along the axis of rotation.

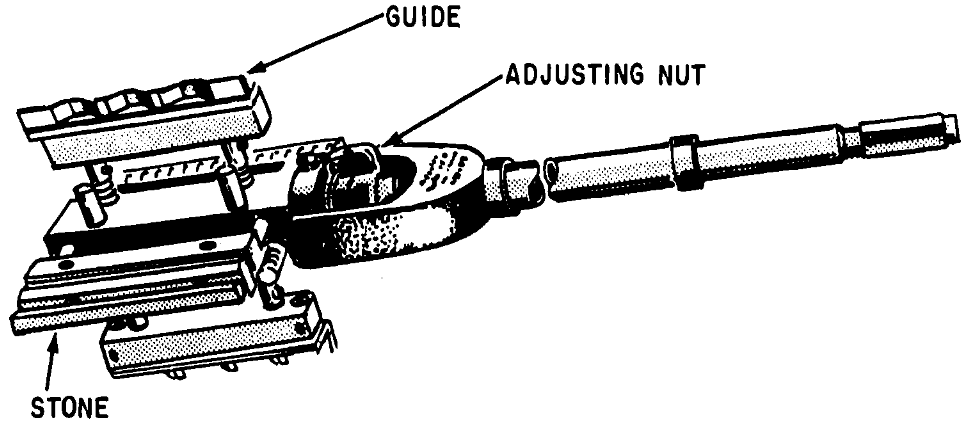
PORTABLE HONING EQUIPMENT

The portable hone shown in figure 13-27 is similar to the type used in most Navy machine shops. It is normally available in sizes ranging from 1 3/4 to 36 inches with each hone set being adjustable to cover a certain range within those sizes. The hone illustrated has two honing stones and two soft metal guides. The stones and the guides advance outward together to maintain a firm cutting action during honing. An adjusting nut just above the stone and guide assembly is used to regulate the size of the honed bore. Accuracy to within 0.0005 inch is possible when the proper operating procedures are observed.

To use the portable hone, follow these basic steps:

1. Clamp the hone shaft in the drill press chuck.
2. Clamp the workpiece to the drill press table.
3. Put the hone into the hole to be polished. Use honing compound as required.
4. Turn on the drill press and use the drill press feed handle to move the rotating hone up and down in the hole.

When a lathe (vertical or horizontal) is used to hone, the work can be mounted in a chuck or on a faceplate and rotated. The honing tool is



28.437

Figure 13-27.–Portable hone.

held in the tailstock with a chuck and moved back and forth in the workpiece bore by the tailstock spindle.

On a milling machine or a horizontal boring mill the workpiece is mounted on the table and the honing tool is mounted in the spindle. The hone is passed back and forth in the workpiece bore by moving the machine table.

Another method is to use a hand held power drill to rotate the hone in the workpiece. Move the rotating hone in and out of the hole by hand.

Each of these methods requires that the hone be allowed to self-align with the workpiece bore. To assist in this, place one or two universals between the hone shaft and the device or spindle which will hold or drive the hone. These universals and shaft extensions are usually available from the hone manufacturer.

When honing large boxes, use a device that attaches to the hone and lends support to the stones and guides to ensure a rigid setup.

STATIONARY HONING EQUIPMENT

Stationary honing equipment is not used as often in the machine shop as the portable hone. Con[illegible] often found in too many shop[illegible] are usually self-contained hones [illegible] built-in honing oil pump and reservoir, a workholding device, and a spindle to rotate and stroke the honing stones. Controls to adjust the rpm, rate of stroke, and the pressure feeding the stones to the desired size are usually standard. Some models have a zero setting dial indicator that lets you know when the desired bore size is reached. This indicator must be preset with a gage that is the same size as the required size of the bore.

STONE SELECTION

The honing stone is made of grit, a bond, and air voids similar to a grinding wheel. The grit is the cutting edge of the tool. It must be tough enough to withstand the pressure needed to make it penetrate the surface, but not so tough that it cannot fracture and sharpen itself. The bond must be strong enough to hold the grit, but not so strong that it rubs on the bore and interferes with the cutting of the grit. Air voids in the structure of the stone aid the coolant or honing oil in clearing chips and dissipating heat.

Honing stones are available in either aluminum oxide grit for ferrous metals or silicon carbide grit for nonferrous metals and glass. Grit sizes from 150 to 400 are available. If a large amount of metal must be removed, a coarse grit stone such as a 150-grit should be used to bring the base to within 0.0002 to 0.001 inch of the finish size. A finer grit stone should then be used to obtain a smooth finish.

Specific recommendations for stone selection is available from the hone manufacturer.

STONE REMOVAL

Honing does not change the axial location of a hole. The centerline of the honing tool aligns itself with the centerline of the bore. Either the tool or the part floats to ensure that the tool and the base align. Floating enables the tool to exert equal pressure on all sides of the bore.

As the honing tool is stroked through the bore, the pressure of the grits is greatest at the tight spots. Thus all taper and out-of-roundness are taken out before any stock is removed from the larger section of the bore. Also any bow is taken out. Since the honing stones are rigid throughout their length, they cannot follow a bow—they bridge the low spots and cut deeper on the high spots, tending to straighten out a bow.

After you have honed out the inaccuracies, you must abrade every section of the bore equally. To ensure that this happens, the rotating and reciprocating motions must be at an odd ratio to each other. Thus, every part of the bore is covered before any grit repeats its path of travel.

If a bore requires honing to correct taper or out-of-roundness, about twice as much stock should be left for honing as there is error in the bore. It is sometimes practical and economical to perform two honing operations: (1) rough honing to remove stock and (2) finish honing to develop the desired finish. As previously mentioned, from 0.0002 to 0.001 inch should be left for finish honing. If a machined bore must be heat treated, rough hone it before heat treating to produce an accurately sized, round, and straight bore. After heat treating, finish hone to correct any minor distortion and to produce the desired finish.

Honing produces a crosshatch finish. The depth of cut depends upon the abrasive, speed, pressure, and the coolant or honing oil used. To produce a finer finish, you can do one or all of the following:

1. Use a finer grit stone.
2. Increase the rotating speed.
3. Decrease the stroking speed.
4. Decrease the feed pressure.
5. Increase the coolant flow.

CHAPTER 14

METAL BUILDUP

Metal buildup is a rapid and effective method of applying practically any metal to a base material with a spray gun to restore worn mechanical equipment, salvage mismachined or otherwise defective parts, and protect metals against corrosion. As compared to original component replacement costs, metal buildup is a low cost, high quality method of restoration.

As you advance in the MR rating you must know how to prepare a surface for metal buildup and be able to set up and operate the equipment used in the thermal spray systems and the contact electroplating process. In this chapter, we shall discuss the thermal spray systems and the contact electroplating process.

Additional information on metalizing can be found in Mil Std 1687(SH) Thermal Spray Process and in NAVSHIPS 0919-000-6010, Instructions for Metalizing Shafts or Similar Objects.

THERMAL SPRAY SYSTEMS

There are four different thermal spray processes: wire oxygen-fuel spray, wire-consumable electrode spray, plasma-arc spray, and powder oxygen-fuel gas spray. In general, all four processes perform the same basic function: They heat the wire or powder that will be applied to its melting point, atomize the molten material with either high velocity gas or air, and propel it onto a previously prepared surface.

The rapid rate at which metal coatings can be sprayed and the portability of the equipment have increased the use of thermal spray processes. Metal coatings are especially useful in rebuilding worn shafts and other machine parts not subject to tensile stress, in hard surfacing where resistance to wear and erosion are desired, and in protecting metal surfaces against heat and corrosion. Naval shipyards, Intermediate Maintenance (IMA), and repair ships use thermal spray processes to coat metallic and nonmetallic surfaces with practically any metal, metal alloy, ceramic, or cermet that can be made in wire or powder form. (Cermet is a strong alloy of a heat resistant compound and a metal used especially for turbine blades.)

In this chapter we shall discuss the wire oxygen-fuel spray process and the powder oxygen-fuel gas spray process with emphasis on the latter. These are the two thermal spray processes you will most likely use as an MR3 or MR2.

APPROVED APPLICATIONS

Thermal spray coatings have been approved by NAVSEA for several applications. Case by case approval is not needed for use of thermal spraying in the applications listed below. Procedures used for these applications are limited to those which have been approved by NAVSEA.

1. Repair of seal (packing) areas of shafts used in oil and freshwater systems to obtain original dimensions and finish.
2. Repair of bearings' interference fit areas of shafts to restore original dimensions and

finish (except for motors and generators where chrome plating is permissible).

3. Buildup of pump shaft wear ring sleeves to original dimensions.

4. Repair of miscellaneous static fit areas, such as those on electric motor end bells, to restore original dimensions, finish and alignment.

SAFETY PRECAUTIONS

- Metalizing melts metal in wire form in a flame and atomizes it by a jet of compressed air into a fine spray. The metal particles may be inhaled easily by anyone present. Personnel using metalizing equipment must wear respirators that have been approved for this kind of work.

- You must wear safety glasses or face shield and proper protective clothing at all times during thermal spraying operations.

- Cleaning solvents are toxic and hazardous to your health. Use only in a well-ventilated area.

- Warning signs must be posted near the operation to warn personnel.

- Adhere strictly to the safety precautions noted in the *Welding Handbook*, Sixth Edition, Section 1 Chapter 9, published by the American Welding Society, and the manufacturer's handbook.

QUALIFICATION OF PERSONNEL

Thermal spray operations must be performed only by qualified personnel. The operator must prepare test specimens for visual, microscopic, bend and bond tests using qualified procedures developed for the particular coating and thermal spray process. In addition, the operator will be responsible for setting up the spraying equipment according to the parameters (gun-to-work distance, air, fuel gas, etc.) required by the spraying procedure.

An operator failing the initial qualification tests may be permitted one retest for each type of test that failed to meet the requirements.

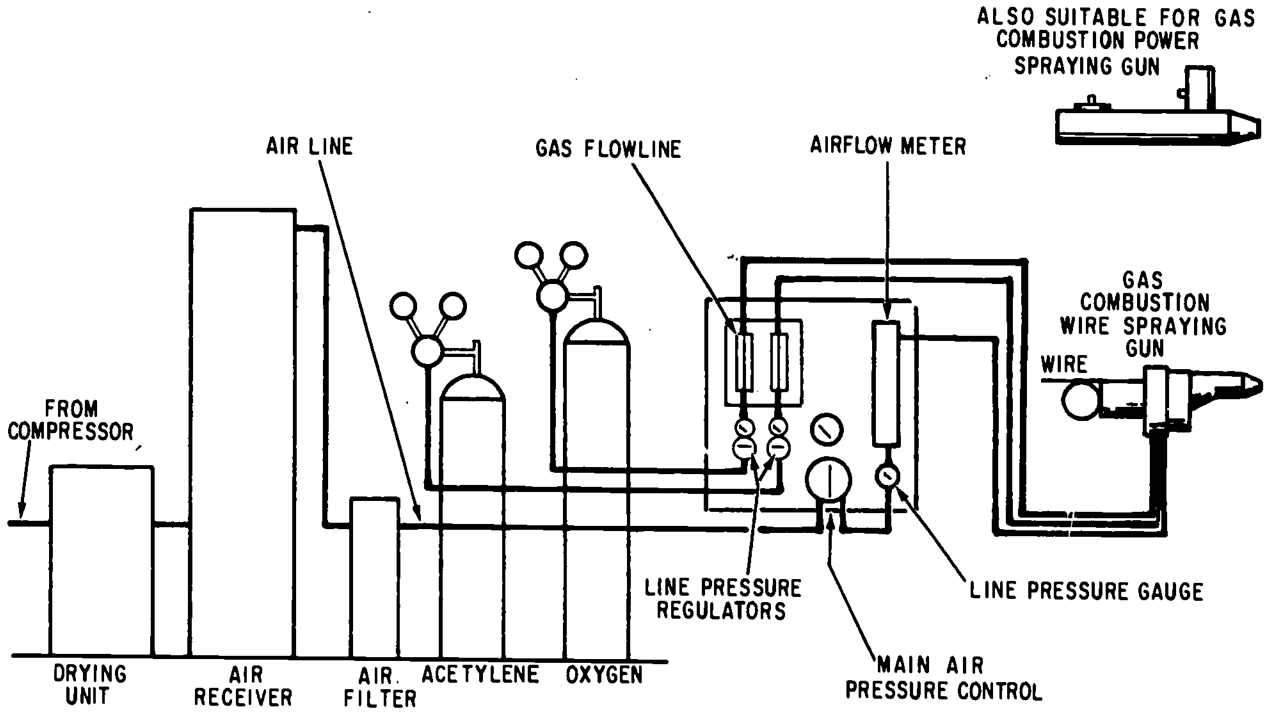
Operators meeting the requirements for the performance tests will be certified to perform manual spraying with the coating system and spray process used in qualification testing.

Certification of the operator will be retained unless 6 months have elapsed since the last use of the thermal spray process. Operators whose certification has lapsed may be requalified by satisfactorily completing the qualification tests. Complete information regarding certification can be found in MIL-STD-1687.

WIRE OXYGEN SPRAY

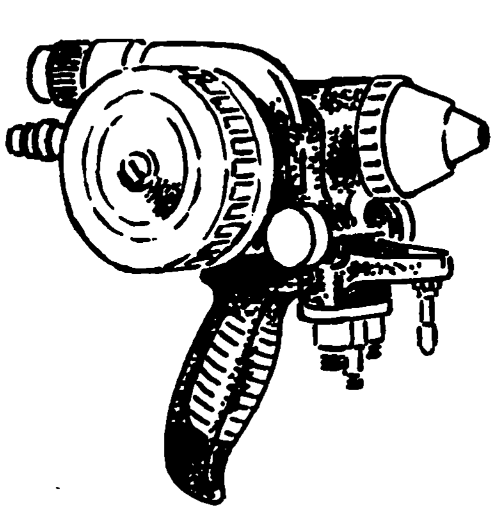
The wire oxygen-fuel spray process is suitable for all purpose use. It offers variable, controlled wire feed rate within the ranges required for all commonly used metallizing wires, and it can be used in both hand held and machine mounted applications. Figure 14-1 shows a typical installation.

The Type 12E Flame Spray Gun (fig. 14-2) can spray metalizing wires, such as aluminum, zinc, copper, Monel, nickel, etc., in wire sizes ranging from 3/16 inch down to 20 gage using acetylene, propane, natural gas, manufactured gas, or MPS as the fuel gas. The wire is drawn through the gun and nozzle by a pair of wire feed ..e rolls, powered by a self-contained compressed air turbine. At the nozzle, the wire is continually melted in an oxygen/fuel gas flame. Then, a controlled stream of compressed air blasts the molten tip of the wire, producing a fine metal spray. Systems of this type are commonly used to spray aluminum wire coatings for shipboard corrosion control applications such as steam valves, stanchions, exhaust manifolds, deck machinery, equipment foundation, etc.



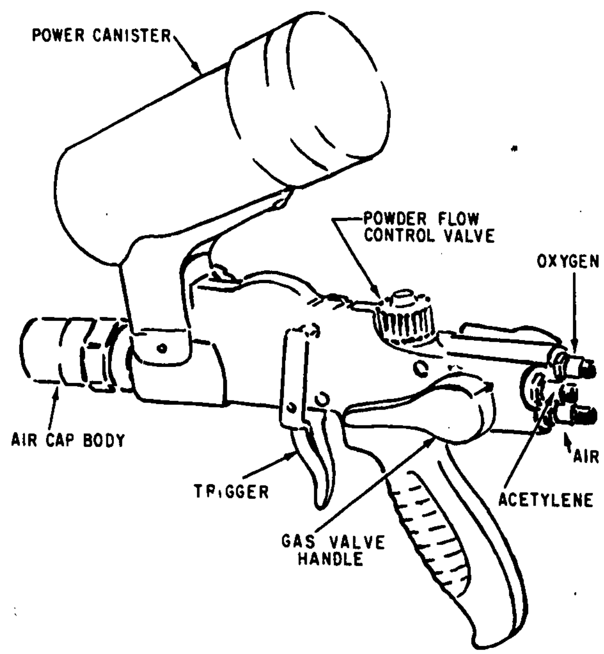
28.440

Figure 14-1.–Typical installation for combustion gas spraying.



28.441

Figure 14-2.–Type 12E spray gun.



28.442

Figure 14-3.–Gravity feed oxygen-fuel powder spray gun.

POWDER OXYGEN-FUEL SPRAY

Figure 14-3 shows a powder spray gun. The powder feeds by gravity through a metering valve and is drawn at reduced pressure into an aspirator chamber. From the chamber the powder is propelled through the flame where it melts and then deposits on the work in the form of a coating. The Type 5P Thermo Spray Gun will spray metal, ceramic, cermet and exothermic powders.

Exothermic coatings composites are those that produce an exothermic (heat evolved) reaction derived from their chemical composition. These coating materials include METCO 402 and 405 wires and 442, 444, 445, 447, 450 powders. When the composites reach a certain temperature in the spray gun flame, nickel and aluminum, for example, react to form nickel aluminide and evolve a great deal of heat. The extra heat provided to the molten particles by the exothermic reaction, coupled with the high particle velocity of the thermal spray process, accounts for the self-bonding characteristics of the coating and its exceptional strength, both perpendicular and parallel to the substrate (base).

Exothermic materials are often referred to as one-step coatings. They produce self-bonding, one-step buildup coatings that combine metallurgical bonding with good wear resistance. They also eliminate the need for separate bond and buildup coatings.

The gravity fed oxygen fuel powder spray gun must be used in a horizontal position. Deposit efficiencies are very high, almost as high as 100% in some cases. Only a minute amount of the powder is lost by being blown away or consumed in the flame.

PREPARING THE SURFACE

Surface preparation is a very important step in the metallizing process. An improperly prepared surface may cause failure of the part under operating conditions. Preparing the surface consists of three distinct operations: (1) cleaning, (2) undercutting, and (3) surface roughening.

We cannot overemphasize the importance of proper surface preparation. Although it is often the most critical part of the total job, it is frequently given the least consideration. Even in the case of coatings that are to be fused, very thorough preparation is sometimes necessary. The type of roughening treatment used and the degree of roughness will depend on the type and thickness of coating to be applied and on the anticipated conditions of service. Frequently, preparation is inadequate simply because proper preparation is not convenient or the necessary equipment may be lacking. But even the best and most elaborate surface preparation is the least costly part of the job. The operator should see to it that the required equipment is available and that the job of preparing the surface is carefully and thoroughly done.

Cleaning Methods

To ensure a good bond between the sprayed coating and the base material to which it is applied, be sure the areas to be coated and the adjacent areas are free from oil, grease, water, paint, and other foreign matter which may contaminate the coating.

Solvent Cleaning

Prior to any blasting or spraying, clean with solvent all surfaces that have come in contact with any oil or grease. (Vapor degreasing is preferred, but you may use solvent washing.) When using solvent, be very careful to prevent any detrimental attacks on the base material; do NOT leave any residue film on the base surfaces. METCO-Solvent Trichloroethane O-T-620 or Toluene TT-T-548 are suitable solvent cleaners. Because of the flammable and toxic nature of most solvents, you MUST follow proper precautions during solvent cleaning. You must also be very careful to protect any parts that may be attacked by the solvents.

Abrasive Cleaning

You can use abrasive blasting to remove heavy or insoluble deposits. Do not use the abrasive blasting equipment you use for general cleaning operations for surface roughening operations.

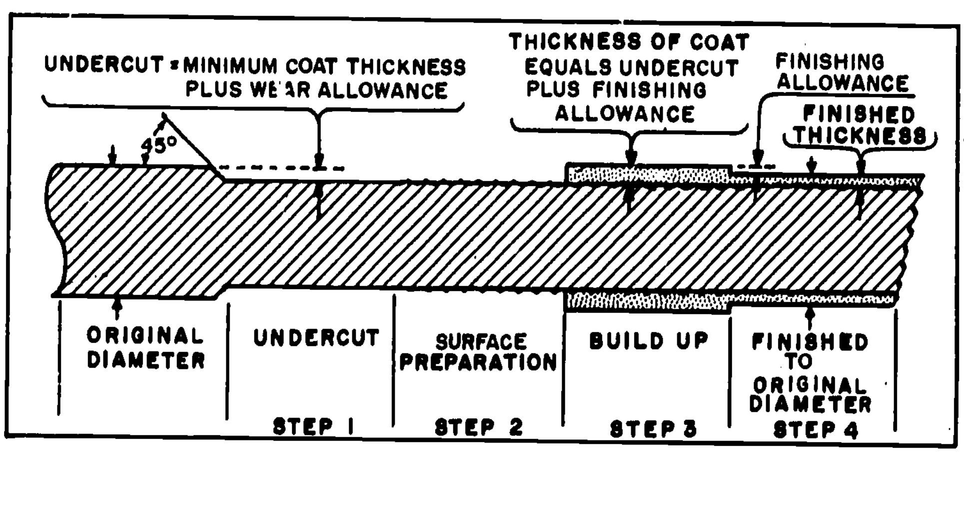
Heat Cleaning

Solvent clean porous materials that have been contaminated with grease or oil and then heat them for 4 hours to char and drive out the foreign materials from the pores. Heat steel castings at 550°F (288°C) maximum; heat aluminum castings, except age hardening alloys, at 300°F (149°C) maximum. In thin sections, use lower temperatures to minimize warpage.

UNDERCUTTING

To obtain a satisfactory thickness of metalized deposit on the finished job, usually you need to undercut the surface to be built up. (See fig. 14-4.) Undercutting must be a dry machining operation, as any cutting lubricants or coolants used will contaminate the surface of the workpiece. When building up shafts, be extremely careful to ensure that the undercut section is concentric to the original axis of the shaft. The length of the undercut should extend beyond both ends of the sleeve or bearing or the limits of the carbon or labyrinth ring, or the packing gland in which the shaft will operate. However, you must be careful not to remove any fillets at points where the shaft section diameter changes. The ends of the undercut should be straight or chamfered at 45° to the base metal.

The depth to which a shaft should be undercut is determined by a number of factors. Some of these factors include the severity of service, the amount of wear expected in service, the depth of metal loss, the remaining thickness of the load carrying member, and the limits of the particular coating. In general, the minimum specified depth of undercutting should be at least equal to the recommended minimum thickness for the particular coating, plus the wear or corrosion tolerance for the application. Undercutting and surface roughening reduce the effective structural cross section of the part to be metalized. Also, sharp grooves and re-entrant angles may produce stress risers. When preparing for thermal spraying, carefully examine from a design standpoint all parts subjected in service to high stresses, shock loads, or critical applications to determine that adequate strength is maintained in the structure. Metal spray deposits cannot be depended upon to restore such qualities as tensile strength or resistance to fatigue stress. NOTE: Shot peening may be used in applications that require high fatigue resistance of the coating system. Shot peening is done by impinging a high-velocity stream of metal or glass particles suspended in compressed air onto the metal substrate. Shot peening is normally performed by dry blasting with cast steel shot with a hardness of Rockwell C 40 to 50. Steel shot must not be used on aluminum or stainless steel; glass beads should be used for aluminum or stainless steel alloys. When



126.1

Figure 14-4.—Major steps in restoration of dimensions with thermal spray.

required, shot peening is performed following machining and before abrasive blasting.

SURFACE ROUGHENING

After undercutting the shaft, you must roughen the undercut section to provide a bond for the metal spray. During undercutting and roughening, do NOT use a lubricant or coolant. Keep the surface clean and dry. Even touching the surface with your hands would contaminate it. If, for any reason, the surface becomes contaminated, you must thoroughly clean and degrease it. The cleanliness and roughness greatly affect the strength of the bond between the base metal and the sprayed coating. You must always carefully control cleanliness to ensure adequate bond strength for the service to which the part will be subjected. Two methods of surface roughening are abrasive blasting and macroroughening for restoration of dimensions greater than 1/2 inch where exothermic materials cannot be used.

Abrasive Blasting for Surface Preparation

Prior to thermal spraying, condition the surfaces to be coated by abrasive blasting. The required amount of surface roughness is related to the configuration of the part. Where part configuration permits, a roughness of 200-300 microinches is desired. Blasting pressure is normally 60 to 80 pounds per square inch (psi) for suction type equipment and the nozzle-to-work distance is about 3 to 6 inches. Blasting must not be so severe as to distort the part. When distortion can occur, such as with thin walled sheet metal parts, reduce roughening as necessary to a minimum surface roughness of 63 microinches and regulate the blasting pressure as necessary.

Abrasive blasting particles used for surface preparation may be either angular nonmetallic grit (e.g. aluminum oxide) or angular chilled iron grit. To prevent rusting, the abrasive particles cannot contain any feldspar or other mineral constituents which tend to break down and remain on the surface in visible quantities. Chilled iron grit must be kept dry during storage and use. Grit designated for coating preparation should not be used for any other purpose. Use the following ranges of grit size as a guide in selecting the desired grit.

GRIT SIZE	GRIT SIZE MESH	USE
Coarse	(-10 to +30)	Use where the coating thickness will be greater than 0.010", and where the roughest blasted surface is required
Medium	(-14 to +40)	Use where the coating thickness will be less than 0.010", and where the roughest blasted surface is not required or cannot be tolerated
Fine	(-30 to +80)	Use under thin coatings which will be used as sprayed or finished lightly by brush blasting

General Notes on Blasting

Clean, dry air is essential. Traces of oil in the air which cannot be readily detected can seriously affect the bond. A good test is to put a drop of cleaning solvent, or equivalent fast drying solvent, on the blasted surface. A distinct dark ring after the solvent dries usually indicates oil in the air.

Keep the blast angle within 10° or 15° from the perpendicular. Where access to the surface is difficult and you must blast from a steeper angle, apply the spray from the same approximate angle. If you blast at an angle from one direction and spray from an angle in the other direction, the bond strength may be close to zero.

Thorough blasting is important. It is good practice to blast until the surface appears fully blasted, that is, until further blasting produces no visible change, and then to blast further for a short period. In other words, blast every piece about 150% as long as appears to be necessary.

Masking for Grit Blasting

All areas of a component that are not to be grit blasted must be covered and masked to prevent damage or contamination by the abrasive blasting media and debris. Rebound grit from the walls of the blast room or blast cabinet may scratch and damage areas of the work which are not to be blasted unless they are adequately covered. Masking for blasting may be an expensive part of the operation and this should be taken into account when selecting the masking method. Following abrasive blasting, remove any masking material that is unsuitable as a masking material for the thermal spray process and replace it with masking material suitable for thermal spraying.

Metal masks and blasting jigs are commonly developed for this purpose. You can sometimes fit the work into a jig in such a way that the part to be blasted is the only part exposed. Where necessary, you must use additional covers or metal masks. One great disadvantage in using metal for masking in blasting, however, is the metal mask itself which blasts away rapidly and must be replaced frequently.

Rubber has proved to be much more successful in masking for blasting purposes, and you should use it wherever possible. Sometimes it is quite practical to construct whole jigs from blocks of rubber rather than from metal. Rubber or aluminum masking tape is very satisfactory for all operations where hand masking can be used economically. Since rubber is not cut by the blasting operation, you can use rubber jigs almost indefinitely. You can use thin rubber tape for heavy blasting protection.

Macroroughening

Macroroughening is a lathe operation performed on bearing areas of shafts or similar surfaces. It consists of cutting a narrow, low-pitch, shallow groove or thread in the surface.

APPLICATION OF COATING

Spray the component using the parameters (e.g. gun-to-work distance, rotational or linear speed of gun to work piece, air, fuel, gas, primary and secondary pressures, and power output) contained in the approved procedure for the material being sprayed.

You must start the spraying operation within 2 hours after completing the preparation of the surface. If more than 15 minutes, but not over 2 hours is expected to elapse between the preparation of the surface and the spraying operation, or when the part must be removed to another location, you must protect the prepared surface from oxidation, contamination, and handling and fingermarks. Clean paper (free of newsprint) will usually provide adequate protection.

Whenever possible (or practical) preheat the work to 200°-225°F to eliminate surface moisture. (Higher preheats may be required on a case basis for specific substrates/coatings which may develop defects due to thickness of coating, thermal gradients, or differences in coefficients of expansion.) Take temperature readings with a contact pyrometer. Do NOT use temperature sticks or similar devices in thermal spray area. If you preheat with a gas flame, do not apply the flames directly onto the area to be sprayed to avoid possible surface oxidation and contamination from carbon deposits.

To safeguard against the possibility of cracks that may occur in the sprayed deposit due to difference in the expansion of the substrate and the sprayed metal, do not spray on substrates with a temperature below 60°F.

Interrupt the spraying operation only to measure thickness or temperature, to change spraying material from bond or undercoat to finish coat, or to permit cooling to prevent overheating. During spraying, do not allow the temperature of the work to exceed 350°F or the tempering/aging temperature of the substrate, whichever is lower. For cooling use a blast of clean air, carbon dioxide, or other suitable gas introduced near but not directly upon the area being sprayed.

In general, keep the direction of the metal spray as close as possible to a 90° angle with the surface being coated and never less than 45°. Apply the coating in multiple passes of 0.005 ±0.001 inch for wire spray and 0.003 ±0.001 inch for powder spray. Cover the entire prepared surface with a pass of spray before proceeding to the next pass.

When you use the macroroughening method of surface preparation, apply at least the first four layers of deposited metal in each direction with the spraying stream directed at 45° to the perpendicular, alternately from left to right, in order to deposit metal onto each face of the thread. Then complete the work by spraying at right angles to the surface.

For cylindrical parts, direct the spray stream at the axis at all times. Coat the part at a rotational speed of 40 to 100 surface feet per minute or as otherwise specified.

When a machined finish is required, spray a minimum of 0.005 inch thickness of sprayed metal in excess of that required for finished dimensions on the surface to provide for machining or grinding. For shafts, make the measurements on the radius. Follow the coating manufacturer's recommendations.

Allow the work to cool normally to room temperature after spraying. If it is necessary to cool the work more quickly, direct an air blast against it. Do not quench with a spray of water or other liquid.

Application of Sealant

To prevent corrosive attack or fluid leakage, sprayed coatings must be treated with a sealant. The particular sealant selected will depend upon the maximum use temperature of the component and the purpose of sealing the coatings. The sealant is applied after spraying and before finish machining. For severe applications, a sealant should be applied again, following finish machining.

Sealants used in thermal spray processes may be of the following types:

1. Paraffin wax
2. Resins
   a. Air dried
   b. Baked (heat cured)
   c. Pressurized
   d. Vacuum impregnated
3. Inorganic

Masking for Spraying

You can use tapes, liquid-masking compounds, silicone rubber, or metal shielding as thermal-spraying masking materials. Tapes used for spray masking must be designed for high-temperature use. Masking materials must not cause corrosion or contamination of the sprayed coatings.

More generally, however, masking tape and masking compound are used for masking materials to be sprayed. Use a pressure sensitive masking tape which is designed to withstand the usual spray temperatures.

Masking compound (METCO or equivalent) is designed for masking where a liquid masking material is more convenient. It is a water soluble material which can be brushed onto any surface to prevent adhesion of sprayed material. Approved masking compound will not run or bleed at the edges.

Masking compound may also be used to protect the spray booths and other equipment which is subject to overspray, such as rotating spindles, chucks, lathes, and the like. When used for this purpose, the surfaces should be cleaned on a regular schedule and the compound should be re-applied, since it will eventually dry out and the sprayed material will then stick to the substrate. For instances when holes, slots, keyways, or other types of recesses cannot be protected by tapes or shields, inserts of carbon or metal calk or rubber should be used. These inserts are installed before abrasive blasting and spraying, left in place throughout the thermal spray operation, and removed after completion of surface finishing prior to the final sealer application.

SURFACE FINISHING

The structure of sprayed metal deposits is granular rather than homogeneous. In spraying, the minute particles of metal strike the surface at high velocity, flatten out, and build up on each other. This structure, which by its relatively low coefficient of friction and oil-retaining qualities makes sprayed metal ideal for all bearing surfaces, creates a problem in finishing. Experimentation and research indicate that with understanding and appreciation of the characteristics of sprayed metal, machining and grinding can be done in the toolroom or on the production line with less trouble than is caused by many alloy materials in solid or wrought form.

A machinist unfamiliar with sprayed metal will grind the tool bit and set it according to past experience on similar metal in its solid or wrought form. As a result, crumbly chips similar to those from cast iron will occur regardless of the metal or the tool setting, and the surface will appear full of "pin pricks" and decidedly porous.

A grinding wheel operator will tend to use the grain and grade wheel used on the same material in wrought form. Regardless of the manner in which the operator dresses the wheel, it will load up immediately and produce a spiralled and discolored surface. If the operator continues and attempts to remove stock with a loaded or glazed wheel, surface checks that cannot be removed will appear. Sufficient working data for both machining and grinding have been established to permit production finishing of all of the commercially used metals that have been developed for thermal spraying. Naturally, some finish better than others, but commercial finishes within commercial tolerances can and are being obtained on all thermal spray alloys.

Because of the possibility of plucking out individual particles during the finishing operation, the finishing parameters are most important with sprayed coatings than with solid materials. With many sprayed materials, maintaining grinding wheel sharpness, for instance, and adhering to proper feeds and speeds may be quite critical. Most applications for sprayed materials consist of fairly thin-sprayed coatings over a substrate. Grinding and finishing operations should take this structure into account and avoid overheating the coatings or seriously deflecting them. For instance, if the coating material is a refractory material with low heat conductivity, there is some danger of developing hot spots during grinding. Machinists who are accustomed to grinding metals should be cautioned to grind slowly enough and apply sufficient coolant to avoid local overheating of such materials. Where thin coatings have been applied over relatively soft substrates, the finishing operations must be done in a way to avoid loads on the coatings that could seriously deflect it.

Requirements

Thermal sprayed coatings are sufficiently different than the same material in wrought form that different grinding wheel and finishing tool recommendations are almost always required. Therefore, tools and wheels should not be chosen on experience with the parent material in wrought or cast form. Selection of the finishing method depends on the type, hardness, and thickness of the coating and the finish desired. Softer coatings are often finished by machining with a carbide tool, using speeds and feeds for cast iron. Harder coating materials are generally finished by grinding.

Wet grinding is usually recommended over dry grinding if the proper wheel is used. Wheels with coarse grain and low bond strength are used to grind sprayed coatings to prevent loading the wheel. When a coolant is used, it should contain a rust inhibitor, and it must be kept clean and free of foreign matter. The grinding wheel must not remain immersed in the coolant because it will become unbalanced due to the absorption of moisture.

The coating manufacturer's finishing recommendations must be consulted and followed in selecting the finishing technique, including the proper tool, feeds and speeds.

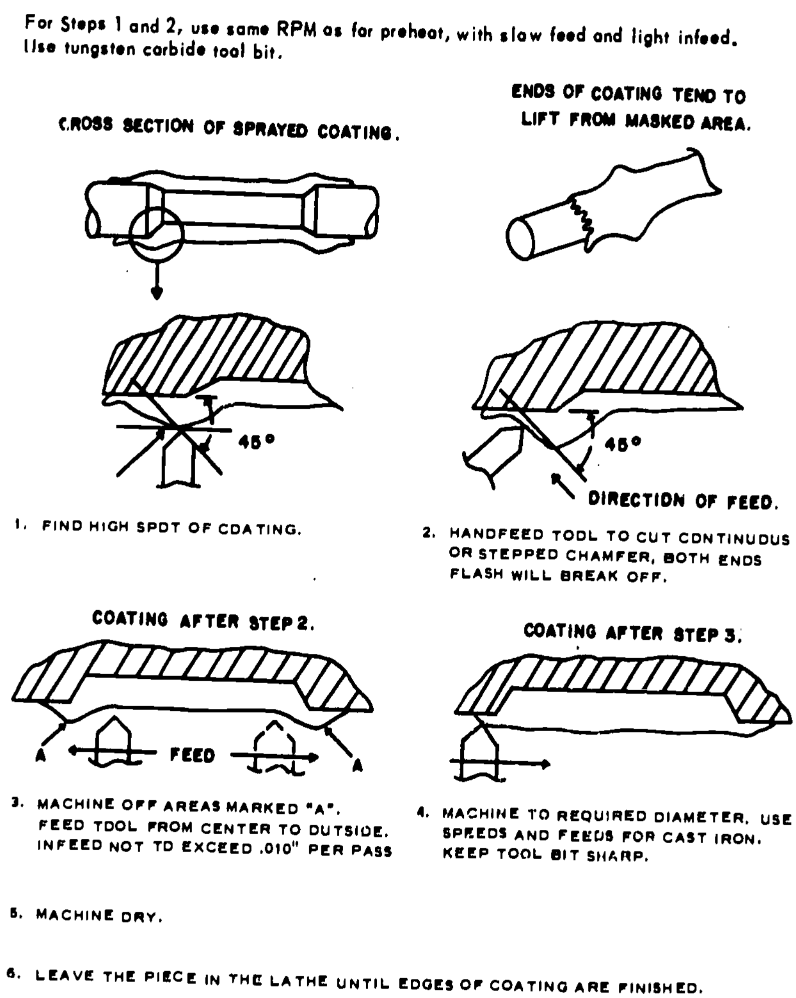
Masking materials should be removed prior to surface finishing. Finishing dimensions are as required by specification or drawing.

Where finishing difficulties do arise even though proper finishing techniques were followed, the spraying operation itself should be reviewed. It is quite obvious that if particles pluck out, for instance, the fault may not be in the grinding but rather in substandard coatings. Excessive moisture or oil in the air supply during the spraying operation can cause this trouble. Maintaining proper gun-to-work distance in accordance with the instruction manuals and spraying at the proper angle to the substrate surface are typical parameters which may affect the structure of the coating adversely and show up as finishing difficulties.

Machining

The sprayed coating stream has an appreciable area (approximately 1/8 to 1/2 inch) and, therefore, the sprayed coating cannot be terminated sharply at the end of the undercut section. At the end of the undercut section (at the shoulders in the case of a shaft), the coating will build up on top of the surface adjacent to the undercut just as thick as in the undercut. If the undercut is 1/8 inch, then something over 1/8 inch of sprayed material will be built up at the shoulder. This buildup at the shoulder of the undercut is emphasized in this discussion because it requires special attention during machining.

The buildup at the shoulder usually has a ragged edge and, if the tool is set to "hog it off", the sprayed material will crack off in chucks, possibly starting a crack which will penetrate the main section of the coating. To avoid this trouble, it is good practice to remove the ragged edge by machining it separately, with a series of fairly thin cuts, until the surface is nearly down to the shoulder before proceeding to take the full cut across the entire surface. (See figure 14-5.)



28.443

Figure 14-5.—Finishing machining of a thermal spray coating.

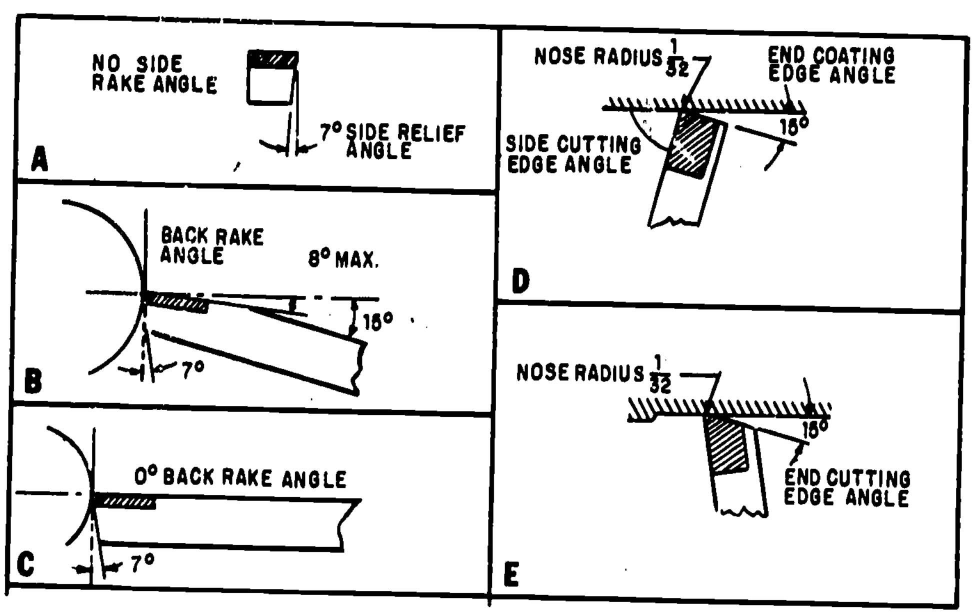
A general guide to finishing is to avoid applying pressure in directions that tend to lift the coating from the workpiece. In many cases, a sprayed coating is subject to more stress in finishing than it is in service. The procedures described above minimize the machining stresses.

Machining sprayed metal is not difficult. A tungsten carbide tool bit, sharpened for cast iron, will be satisfactory. The sprayed coating contains hard oxides. High work speed, slow traverse and light infeed are required. When it is necessary to hold dimension to tight tolerance, tool bit wear must be taken into account. Cemented carbide tools have been found more satisfactory than softer tools for machining most sprayed metals. Even the softer sprayed metals which can easily be cut with high-speed steel tools, have an abrasive action on the tool tip. Carbide tools are necessary for the harder materials where machining is to be done.

Figure 14-6 illustrates proper tool configuration for machining sprayed materials. The usual rules that apply to the use of carbide tools for heavy machining work should not be followed since they do not apply to machining sprayed materials. For instance, when machining sprayed materials, it is never necessary to take a cut deeper than about 0.025". The side cutting angle (see figure 14-6) is not important since the cutting is done by the tool on the radius at the nose of the tool. No back rake is required but it may be as much as 8°.

Grinding

Wherever the ground surface is to be used for a journal or bearing surface it is most important that the final surface is clean and not contaminated with grinding abrasive. While such surfaces can be cleaned by scrubbing after grinding, it is often much more satisfactory to seal the surface prior to grinding. Sealers, such as METCO-SEAL AP and METCO 185 Sealer, have been developed for this purpose. The use of sealer before grinding prevents contamination of the pores of the sprayed coating and also helps to provide a cleanly cut ground surface instead of a surface with the particles smeared or drawn into feathers.



28.444

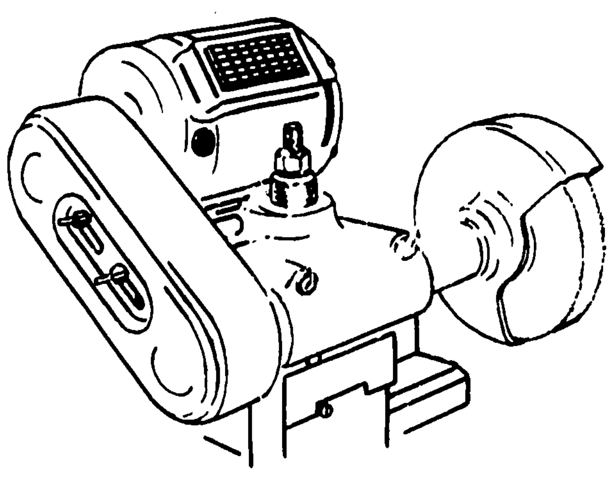
Figure 14-6.—Cutting tool angle for machining thermal spray-metal coating.

Heavy grinding equipment with carefully trued concentric wheels should always be used. (See fig. 14-7.) Pounding from an eccentric wheel or vibration due to the use of equipment that is too light for the job will damage the coatings or produce a poor finish.

Wet-grinding is recommended whenever suitable equipment is available. When proper equipment is used no special difficulties arise in grinding sprayed materials as compared to grinding these same materials in other forms. Of course, attention should be given to the special problems resulting from the structure of sprayed materials as discussed earlier. It should be remembered, however, that sprayed materials are not the same materials structurally as those in solid form and that different wheels, feed, speeds, etc. should be used in accordance with the coating manufacturer's recommendations.

The softer sprayed materials, particularly the sprayed metals, tend to "load" a wheel. Wheels with relatively coarse grain and low bond strength are necessary for such materials so that the wheel will break down before loading.

Ground surfaces should be thoroughly cleaned after grinding whenever the surface is to be used for a journal surface or a mating machine element part. This procedure is emphasized because the porous structure of most sprayed coatings are more inclined to retain abrasives on the surface than solid materials. As mentioned earlier, sealing prior to grinding will ensure clean final surfaces.



28.445
Figure 14-7.—Lathe grinder for dry grinding of thermal spray coating.

Figures 14-8, 14-9, and 14-10 illustrate the proper techniques for finishing keyways, holes and other openings, and the ends of coatings.

CONTACT ELECTROPLATING

Contact electroplating (brush-on) is a method of depositing metal from concentrated electrolyte solutions without the use of immersion tanks. The solution is held in an absorbent material attached to an inert portable electrode (anode) of a d.c. power pack. The cathode lead of the power pack is connected to the workpiece to provide the ground, completing the plating circuit. Electroplating deposits metal by contact of the anode with the work area. Constant motion between the anode and the work is required to produce high quality uniform deposits.

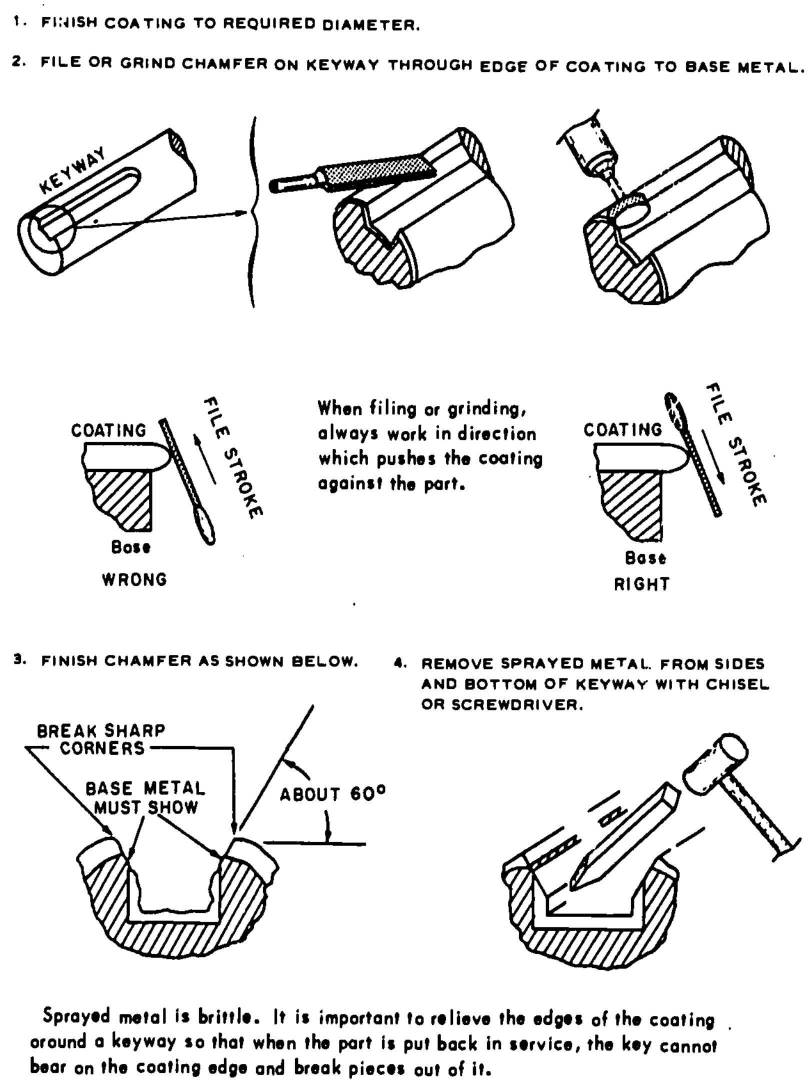
Contact electroplating (also referred to as contact plating) can be used effectively on small to medium size areas to perform the same functions as bath plating; for example, corrosion protection, wear resistance, lower electrical contact resistance, repair of worn or damaged machine parts, etc. This process is not recommended to replace bath plating. However, there are some advantages which make contact electroplating superior in some fields of application:

- The equipment is portable; plating can often be done at the job site.

- It can reduce the amount of masking and disassembly required.

- It permits plating of small areas of large assembled components or parts too large for available plating tanks.

- By plating to the required thickness, it can often eliminate finish machining or grinding of the plated surface.



28.446

Figure 14-8.—Finishing keyways.

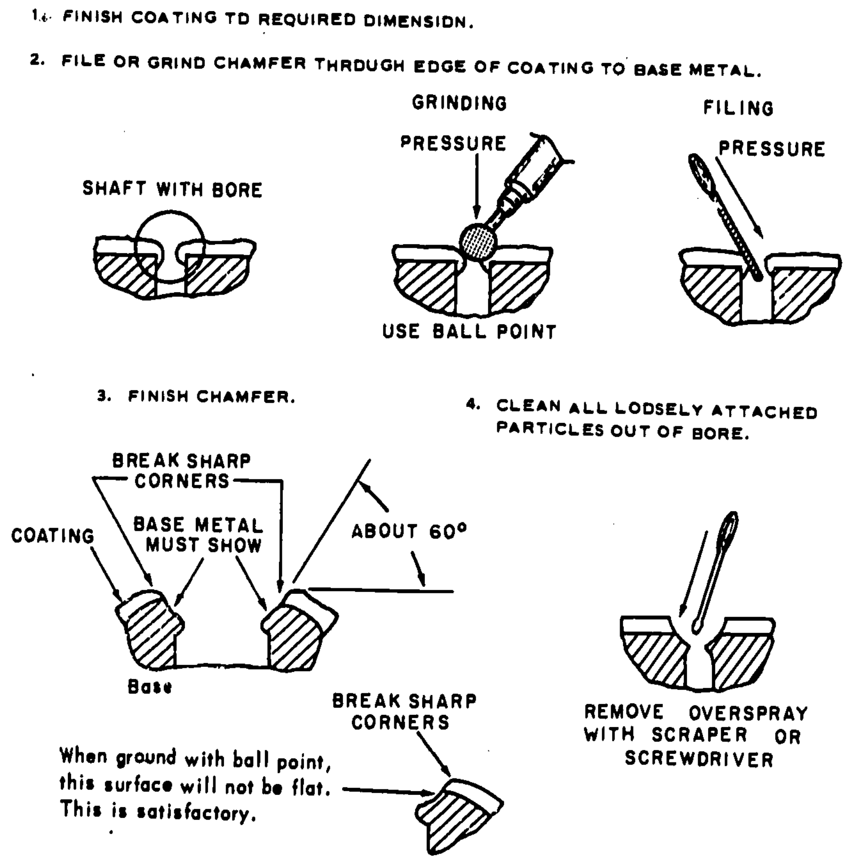
- Damaged or defective areas of existing plating can be touched up, instead of complete stripping and replating of the entire part.

Although the contact electroplating equipment—power pack, plating tools, solutions, plating tool coverings—are discussed in detail throughout this chapter, the following sections contain brief descriptions which you need at this point.

POWER PACK

Contact plating power packs are available in direct current output ranges of 0-15 amperes at 0-20 volts to 0-150 amperes at 0-40 volts. These power packs operate on 115- or 230-volt 60-Hz single- or three-phase a.c. input.

The intermediate sizes, 25 to 100 ampere maximum output, are most commonly used.



The edges of the coating must be relieved around oil holes, slots or other openings in the part, so that there is no possibility of pieces of sprayed metal breaking off and getting between mating surfaces.

CAUTION: Clean the metallized piece thoroughly before putting it back in service. Any loose particles of sprayed metal might cause trouble.

28.447

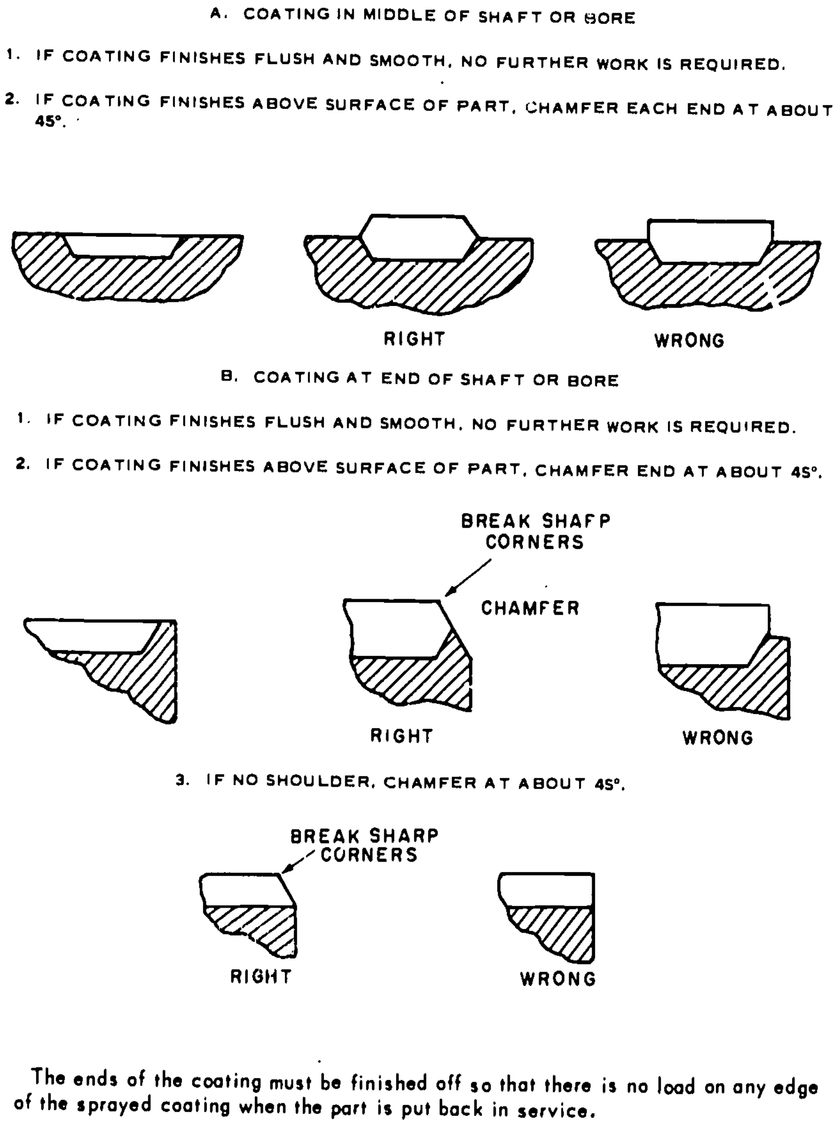
Figure 14-9.—Finishing holes and other openings.

The units in this range are portable, weighing less than 150 lbs, yet can provide the required power for most shipboard and shop type work. A unit in the 60- to 100-ampere range is recommended as basic contact plating shop equipment. Even though subsequent workload demand may require supplementing it with smaller or larger units, a unit of this size will always remain useful. There are units available up to 200 amperes.

Power packs are transportable through both a 25-inch diameter opening and a 20 X 35-inch rectangular opening.

PLATING TOOLS

Contact plating tools consist of a stylus handle with a conductive core, which is insulated for operator safety, and an insoluble anode normally of high quality graphite. Since



28.448

Figure 14-10.—Finishing ends of coating.

considerable heat is generated during plating operations there must be a means of cooling the plating tool. The handles of plating tools have cooling fins to dissipate heat. In some cases, large tools may require the use of plating solution or water as a cooling medium. Graphite anodes are brittle and are not practical for use in locations where a very small diameter anode is required. For plating holes less than 1/2 inch in diameter, or narrow slots and keyways, anodes made of 90% platinum and 10% iridium material are recommended.

The removable anodes are available from the equipment manufacturers in a wide range of standard sizes and three basic shapes: cylindrical or convex—for plating inside diameters; concave—for outside diameters; and, flat or spatula shaped.

Graphite material may also be purchased for manufacturing special tools.

SOLUTIONS

The solutions used in contact plating include preparatory solutions for cleaning and activating the surface to be plated, plating solutions for depositing pure or alloy metals, and stripping solutions for removing defective plating. These solutions are manufactured and sold by the process equipment manufacturers. Solutions of any trade name can be used if the deposits meet the applicable plating specification and if they are qualified by procedure tests. However, plating and preparatory solutions of different manufacturers must not be intermixed or substituted in a plating procedure.

For plating operations, solution is either poured into shallow glass or porcelain dishes or beakers for dipping or into a pump for dispensing through solution-fed tools.

PLATING TOOL COVERINGS

Cotton batting of surgical grade U.S.P. long fiber, sterile cotton is the most common tool covering. It is fastened to the anode to hold and distribute the solution uniformly. Cotton batting alone can be used for jobs involving a few short preparing and plating operations or to facilitate maximum tool to workpiece conformance for plating in corners or on irregularly shaped areas. When longer tool cover life is desired, cotton, Dacron or cotton-Dacron tubegauze sleeving should be used over the cotton batting. In addition to cotton batting and tubegauze, Dacron batting, Pellon and treated "Scotchbrite" may also be used as plating tool coverings.

OPERATOR QUALIFICATION

Only qualified operators are permitted to perform production plating. The cognizant plating shop and the quality control department maintain a list of qualified operators. Qualification of operators is the responsibility of the performing activity and is based on the operator's ability to:

1. Successfully complete a process equipment manufacturer's training course, in-house, or other approved training course. To qualify the operator must show proficiency in the contact plating process which includes the following:

   (a) Preparation of metal surface for contact plating
   (b) Selection of the proper power settings, tools and solution
   (c) Proper masking technique
   (d) Proper plating technique
   (e) Calculation of plating thickness
   (f) Proper surface finishing technique

2. Successfully plate mock-ups, simulating typical plating work required at the facility, to the specified quality requirements and thickness range indicated in NAVSHIPS 0919-000-6010.

Completion of an approved training course and certification will not always assure that the operator is skilled enough to accomplish all possible jobs that may be encountered. Much of the required skill can be gained only from actual plating experience. Newly trained and certified operators should generally work under the guidance of an experienced operator for a minimum of 30 days. If there are no experienced operators at the facility, experience can be gained by limiting the plating work to simple applications at first, avoiding jobs requiring heavy plating buildup, especially for critical and rubbing contact applications, and gradually progressing to more difficult tasks. In either event, the plating vendor or distributor should be consulted whenever plating problems arise. If the problem cannot be resolved by consultation or if a plating job is beyond in-house capability, vendor services should be required to assist with the actual plating and to provide concurrent on-the-job training.

HEALTH AND SAFETY PRECAUTIONS

The plating solutions may be poisonous and may produce fumes which are irritating to the eyes. For these reasons, you must take the following precautions:

- You must wear safety glasses or face shield, rubber gloves and rubber apron or laboratory clothing at all times when electroplating.

- NEVER let your skin come in contact with the solutions. If it does occur, wash your skin thoroughly with soap and water.

- ···en electroplating in air conditioned compartments, nonventilated compartments, confined areas of ventilated compartments, or in compartments with only minimal ventilation, be sure that portable ventilation exhaust blowers are installed and operating BEFORE you begin. Direct the exhaust hose from these blowers to an adequately sized exhaust terminal or discharge directly to the weather where practical.

- Warning signs must be posted near the operation to warn personnel that toxic and poisonous chemicals are being used.

- Adhere strictly to the safety precautions noted in the caution plate on the equipment or specified in the manufacturer's operating procedures.

TERMINOLOGY

Contact electroplating is highly technical and introduces many terms of which you probably have little knowledge. The next few pages contain definitions which will be of great benefit as you study the process of contact electroplating. Read them carefully and then refer to them as necessary as you progress through the remainder of the chapter.

ACTIVATE: Removing passive film which is normally present or which forms quickly on certain metals. This is conducted using an appropriate solution and forward current. Activating improves adhesion of the plating to follow.

ADHESION: The degree to which an electroplate is bonded or "sticks" to the base material.

ANODIZED COATING: An oxide coating formed on aluminum by making it the anode in an appropriate solution. Thickness varies from 0.000020 to 0.001 inch depending upon the application.

ALLOY: Metallic combination of two or more elements.

ALTERNATING CURRENT (a.c.): Electrical current that has an alternating direction of current flow, usually 60 times per second.

AMPERE-HOURS (also AMP-HR or Ah): A measure of a total quantity of electrical current such as may be contained in a battery. Comparable to a quantity or volume of water.

AMPS, AMPERES, or AMPERAGE: A measure of the quantity of electrical current flowing through a conductor such as wire or a conductive solution. Comparable to rate (gal per minute) at which water flows through a pipe.

ANODE: Positive terminal in a conductive solution. Metal ions in the solution flow away from the positive terminal. In the reverse direction, the workpiece is positive and there is a tendency to remove material or "etch" the workpiece. In the forward direction, the workpiece is negative and metal ions flow to the part; that is, the workpiece is plated.

ANODE-TO-CATHODE SPEED: The rate of movement of the plating tool relative to the surface being plated. The relative movement can be obtained by moving the tool, by moving the workpiece, or by moving both.

ANODIC CORROSION PROTECTION: Corrosion protection offered by a deposit more reactive than the base material. The deposit, therefore, corrodes preferentially to the base material. The coating, therefore, does not have to be pore-free.

BAKE: Heating a part for several hours at approximately 400°F, usually to remove entrapped gases such as hydrogen.

BATH PLATING: Electroplating by immersing the workpiece in a tank of plating solution.

BHN: Brinell Hardness Number.

BURNED DEPOSIT: A loose, powdery, defective deposit applied by improper plating. Burned deposits tend to occur first at high current density areas, such as masked edges and sharp external corners, and can be recognized by being distinctly darker in color. A burned deposit can be covered, but the burned layer will have poor cohesion and the final surface will be rougher. Moderate, localized burning can be tolerated in most applications. Severe, overall burning requires that the plating operation be stopped to allow chemical or mechanical removal of the burned layer. Plating then can be resumed after the surface is properly prepared.

CARBURIZED: A part that has been case hardened by impregnating carbon in the surface and then heattreating the part.

CASE HARDEN: Hardening an iron base alloy, such as steel or cast iron, so that the surface layer or case is substantially harder than the interior.

CATHODE: Negative terminal in an electrolyte. Metal in an electrolyte flows to the negative terminal. In the "forward" or plating direction, the workpiece is negative and metal flows to it.

CATHODE EFFICIENCY: The percentage of current flow (amperes) or quantity of current (ampere-hours) used to electroplate metal. (See NOBLE METALS.) The remainder of the total amperes or ampere-hours (current) deposits hydrogen which dissipates in the atmosphere.

CATHODIC CORROSION PROTECTION: Corrosion protection offered by a deposit more noble than the base material. The deposit must be pore-free, since the base material will corrode in preference to the coating.

CHROMATE COATING: A coating applied on many metals, often zinc and cadmium. The color of the coating varies from almost transparent to yellow or brown. It is applied for additional corrosion protection, decorative reasons, or as a base for paints.

COHERENT: Holds firmly together as one piece; that is has high resistance to breaking apart in pieces.

CONSTANT FACTOR: The factor is constant and is not affected by plating conditions, such as current density, temperature, etc. A certain number of amp-hr, therefore, always deposits a certain volume of metal from the solution.

CONTACT AREA: The area of contact made by a plating tool on the workpiece measured in square inches.

CURRENT DENSITY: The plating current being passed per square inch of contact area. The value is determined by dividing the plating current by the contact area. When 10 amps are drawn with a tool making 5 square inches of contact with a part, the current density is 2 amps per square inch.

DENSE: Has no voids, cracks, or pores.

DESMUT: To remove a loose, powdery, darker surface film formed by a previous etching operation.

DIFFUSION: The movement of atoms in a solid, liquid, or gas; usually tends to make the system uniform in composition.

DIRECT CURRENT (d.c.): Electrical current that flows in only one direction.

DPH or DIAMOND PYRAMID HARDNESS: A microhardness test that is suitable for testing the hardness of thin or small areas, such as an electrodeposit. It develops square impressions. DPH hardnesses are converted to more familiar Brinell or R_c values using conversion charts.

DRAG-OFF: The solution left on the workpiece when plating is completed. This solution will be lost in the following rinse operation.

DUCTILITY: The property of a material that permits it to be stretched permanently without fracture. The opposite of brittleness.

ELECTROLYTE: A solution that will conduct electricity.

ELECTROPOLISH: To polish a surface while electrochemically etching in a special solution.

ETCH: To electrochemically remove material from a surface. Conducted with appropriate solution and reverse current.

"F" or FACTOR: The ampere-hours required to deposit the volume of metal equivalent to a 0.0001-inch thickness on 1 square inch of area.

FORWARD CURRENT: Direction of d.c. current flow in which metal ions tend to flow away from the anode and toward the workpiece. The anode is positively charged and the workpiece is negatively charged.

FRETTING: Wear that occurs between two adjacent surfaces caused by a minute back and forth rubbing movement or vibration.

FRETTING CORROSION: Wear that occurs between two adjacent surfaces caused by a minute back and forth movement or vibration. The wear is accompanied by formation of oxides from the small, worn off particles.

GALLING: The damaging of one or both metallic surfaces by removal of particles from localized areas during sliding friction.

GASSING: Evolution of hydrogen gas bubbles on the workpiece, either by activating, plating, or by chemical attack of the activator on chromium.

GRAIN STRUCTURE: All metals have a granular or cellular structure. The grain size varies from invisible to the naked eye to perhaps 1/8 inch in diameter. Grain structure refers to the overall appearance of the arrangement of grains.

HARDCOAT: An oxide coating formed on aluminum by making it the anode in an appropriate solution. Thickness varies from 0.001 to 0.005 inch. The coating is used primarily for wear resistance.

HARDNESS: The ability of a material to resist indentation. Brinell and R_c are common hardness tests.

HYDROGEN EMBRITTLEMENT: Embrittlement of a material caused by absorption of hydrogen. Occurs only with certain materials such as steel over 40 R_c, titanium, and certain harder stainless steels.

IMMERSION DEPOSIT: A metallic deposit which forms on more reactive metals by chemical reaction with certain plating solutions. No flow of d.c. current is required. Immersion deposits ordinarily have poor adhesion.

IONS: Electrically charged atoms or groups of atoms in a solution. Metal atoms are charged positive and migrate toward the cathode.

KNOOP: A microhardness test which is suitable for hardness testing thin or small areas such as an electrodeposit. Knoop hardness values are converted to more familiar Brinell or R_c hardness values, using conversion charts.

LITER: Volume equal to 1.0567 quarts.

MATTE: A dull, satiny appearance resulting from a fine microroughness.

MICROCRACKED: A type of deposit structure in which there are numerous fine surface-to-base metal cracks. Cracks are so numerous and fine that they can be seen only at high magnifications.

MICROPOROUS: A type of deposit structure in which numerous fine pores exist. The pores are so numerous and fine that they can be seen only at high magnification.

MICROSTRUCTURE: The structure of deposit when viewed at 50X magnification and over.

MILKY: A type of deposit appearance that is almost bright but has a cloudy appearance due to a very fine microroughness.

NITRIDED: Case hardened surface on certain steels formed by heating in a nitrogen containing material. Nitrogen diffuses into the surface, causing a hard case.

NOBLE METALS: Metals may be classified according to their tendency to be corroded or chemically attacked. The noble metals are less easily corroded or chemically attacked. They include metals such as copper, nickel, and gold.

NODULAR: Type of electrodeposit that has rounded projections on the surface, visible to the naked eye upon close examination.

OHMS or SYMBOL Ω: A unit of measure of resistance to the flow of electrical current.

OXIDE: Reaction product of a metal with oxygen.

PASSIVATE: The formation of a thin, invisible oxide film on certain metals which impairs adhesion of a following electroplate.

pH: A measurement value on a scale of 0 to 14 of the acidity or alkalinity of a solution. 0 for example is strongly acidic, 4 less acidic, 7 neutral, 10 mildly alkaline, and 14 strongly alkaline.

PLATING RATE: The rate at which a deposit builds up. In this manual it is expressed in inches per hour.

PORES: Small random holes in a deposit just barely visible to the naked eye.

POROUS: A type of deposit that contains pores.

PREPLATE: A thin preliminary plating applied using plating solution other than the final desired solution. Preplates are used to improve adhesion.

PREWET: Applying plating solution to the surface before applying current. The operation improves adhesion of deposits from certain solutions by ensuring that plating begins on a surface covered all over with full strength solution.

R_C: Rockwell C Hardness.

REACTIVE METALS: Metals may be classified according to their tendency to be corroded or chemically attacked. The reactive metals are more easily corroded or chemically attacked. They include metals such as aluminum, steel, and zinc.

REVERSE CURRENT: Direction of d.c. current flow in which metal ions tend to flow away from the workpiece and toward the anode. The anode is negatively charged and the workpiece is positively charged.

SACRIFICIAL CORROSION PROTECTION: Anodic corrosion protection.

SCALE: Surface oxidation on a metal caused by heating in air or an oxidizing atmosphere.

SEIZING: Same as GALLING.

SMEARED METAL: Deformed material near the surface caused by machining, grinding, or wear.

STRESS: Pressure (force per unit area) existing in a deposit. Tensile stress is a "pulling apart" type of stress. Compressive stress is a "pushing together" type of stress.

STRESS CRACK LIFTING: The type of deposit structure caused by the development of surface-to-base metal cracks which then curl up on the edges because of poor adhesion. Can be seen visually or at low magnification. Similar in appearance to a dried up clay lake bed.

STRESS CRACKS: Cracks running from the plated surface to the base material. Can be seen visually or at low magnification. Normally detrimental only when corrosion protection is desired of the plating.

STRIPPING: Removing an electroplate from a workpiece by chemical or electrochemical means.

TANK PLATING: Same as BATH PLATING.

THROWING POWER: The ability of a plating solution to provide a uniform deposit on a part that has surface irregularities readily visible to the naked eye. A solution with good throwing power is particularly useful for pit filling since relatively more plating is applied at the bottom of the pit.

VARIABLE FACTOR: Factor is not constant but varies depending on plating conditions such as current density and temperature. A given number of amp-hr, therefore, will deposit different amounts of metal, depending on plating conditions. Plating conditions, therefore, must be controlled to get desired thickness of deposit.

VOLTS: A measure of the electrical force applied. Comparable to water pressure.

WATERBREAKS: The breaking of a water film into beads such as on a waxed car.

APPLICATIONS

The contact plating process is a relatively new and rapidly expanding field. When used for depositing a corrosion resistant coating, electroplating has shown sufficient success to permit almost unrestricted use of the process in this area. However, the vast field of possible applications for repairing worn or damaged parts of high-speed rotating and reciprocating machinery has just begun to be explored and warrants a more cautious approach. Requirements for contact plating are specified in Table 14-1 which defines the areas of

Table 14-1.—Requirements For Production Contact Plating

Class	Allowable Thickness (Max)	Restrictions	Qualification Requirements
I	No limit[1]	None	See Operator Qualifications
II	0.030"[2]	None	
III	0.020"[2]	Excluding Class IV and V	Original qualification plus plating of a mock-up simulating the production plating. The plated mock-up must be approved by the Quality Control Department of the performing facility.
IV V		NAVSEA Approval required on a case basis.	

[1]Limitations to be governed by practical and economical use of the metals deposited. The material manufacturer's recommendations should not be exceeded.

[2]Thickness limit does not apply to filling-in pits, scores, dents, etc. where the total surface area comprises 10% or less of the area to be plated. The maximum allowable plating thickness shall not exceed that recommended by the material manufacturer.

permissible use of contact plating. For simplification, applications are classified as follows:

Class I: Plating used for decorative or corrosion preventive functions only.

Class II: Parts that remain in static contact with other plated or unplated parts.

Class III: Parts that make rubbing contact with other plated or unplated parts, excluding those which apply to Class IV.

Class IV: Rubbing contact parts in elements of turbine/reduction gear, turbo or diesel electric power generating units, and main propulsion shaftings.

Class V: Parts under the cognizance of the Nuclear Power Division.

List of Successful, Typical Repair Applications

- Bearing Seats, Saddles, and Supports

Ball Bearings: Plating of shafts and bores to reestablish close tolerance fits. The use of an outer layer of tin (0.002 to 0.003 inch thick) has produced significant results in reducing fretting of bearing bores in electric motor end bells and also contributes to noise reduction.

Sleeve Bearings: Plating of seats, saddles, and supports to correct for oversize machining and out-of-roundness caused by distortion.

- Flanges and Flat Surfaces

Steam turbine casing joint flanges: Repair steam cuts and erosion damage.

Diesel engine cylinder blocks: Restore mating surfaces damaged by fretting.

Wave guide plumbing: Plating flange seal areas to provide corrosion resistant metallic gaskets.

- O-Ring Grooves and Sealing Surfaces

Repair of pits, scratches, and gouges on parts used for air, oil, salt- and freshwater service.

- Close Tolerance Mating Parts

Pump impellers: Repair of worn bores and keyways to restore design size and fit on shaft.

- Hydraulic Equipment

Scored, scratched pitted or gouged surfaces of cylinder walls, tailrods, steering gear rams, spool valves, and O-ring seal grooves.

- Masts, Periscopes, Antennas, and Associated Hull Fittings

- Shafting

Areas worn by contact with seals and packing.

- Steam Valves

Repair turbine nozzle control valve seat's hard facing by plating 0.003-0.005 inch thickness of cobalt over copper and nickel substrates. Apply a thickness as required to repair steam cutting and erosion damage and restore valve seat geometry.

- Applications Approved by NAVSEA on a Case Basis

Repair of steam turbine rotor bearing journals.

Repair of diesel engine crankshaft main bearing journals.

LIMITATIONS

- Cracks: Plating cannot be made over areas containing cracks. Cracks must be completely removed by grinding or other mechanical means. Fill shallow grooves by copper plating and then plate the area with the specified material. Repair deep grooves by welding.

- Chromium plating on existing bath chromium deposits: Brushing chromium plating on existing bath chromium deposits has not been consistently successful. The problem experienced by vendors and users alike has been poor bonding. For this reason, selective deposition of chromium on existing bath chromium deposit is not recommended for engine parts that make rubbing contacts. To plate such parts, completely remove previous chromium deposits prior to contact plating. As an alternate, apply a nickel flash over the existing bath plated chromium and follow with contact chromium or other plating material.

- Babbitt restoration: The feasibility of contact deposition of babbitt on worn or damaged babbitt surfaces of bearings is being evaluated by a government laboratory. Pending affirmative recommendations, babbitt restoration by the contact plating process should not be made.

- Deposition of chromium: Contact plating solutions can produce deposits with mechanical properties which will satisfy the requirements for most plating work. Therefore, selective plating coatings can normally be used to repair or as a substitute for bath plated coatings. The exception to this is the use of chromium to refurbish worn parts. Deposition of chromium by contact electroplating is not recommended because the deposit is much softer than that deposited by bath electroplating, the thickness of the buildup is limited, and the process is tedious and slow. As an alternate, other metals such as cobalt or nickel will provide wear resistance and hardness properties which are suitable for most applications where chromium would normally be used. For areas requiring extensive buildup, deposit copper up to about 0.020 inch of the final dimension, and then deposit an outer layer of cobalt, nickel-tungsten or cobalt tungsten for greater wear resistance and surface hardness.

PROCESSING INSTRUCTIONS

The equipment and solution manufacturers have prepared comprehensive instruction literature covering the use of their products. You should closely follow these instructions, especially those concerning procedures for preparing base metals for plating and the use of individual plating solutions, to ensure satisfactory plating results. A list of vendors' literature is shown in table 14-2. Detailed, step by step contact plating procedures for the most commonly used metals are also found in Engineered Uniform Method and Standard No. 3426-801. (Copies may be obtained from Commander, Mare Island Naval Shipyard, Vallejo, California 94592). Another Government document on this subject is MIL-STD-865 (USAF). (Copies may be obtained from Commander, Hill Air Force Base, OOAMA/OONEO, Utah 84401.)

Refer questions arising from difficulty with equipment or solutions to the manufacturer or his nearest local sales representative and send a report, identifying the problem and its resolution, to NAVSEC (Code 6101D) for information.

QUALITY CONTROL

Quality control is comprised of several factors: documentation, process control, general (all plating) inspection, and liquid penetrant inspection of plating for rubbing contact service.

Documentation

The cognizant quality control department will ensure that plating produced meets the requirements of the applicable specifications listed below:

Deposit	Specification
Cadmium	QQ-P-416
Chromium	QQ-C-320
Copper	Mil-C-14550
Gold	Mil-G-45204
Nickel	QQ-N-290
Silver	QQ-S-365
Tin	Mil-T-10727
Tin-lead	Mil-P-81728
Zinc	QQ-Z-325

Table 14-2.—Publications of Major Vendors of Contact Plating

The major vendors of contact plating equipment and material are listed below. These vendors also provide consultant and operator training services.

VENDORS	PUBLICATIONS*
Dalic Process SIFCO Metachemical Division of Steel Improvement and Forge Company 935 E. 63rd Street Cleveland, Ohio 44103 Piddington & Associates Ltd. 3221 E. Foothill Boulevard Pasadena, Calif. 91107	Operating Instruction Manual Containing Technical Bulletins: IM-1, 2, 3, 10 and 11 through 20 IM-200, 202 through 210 IM-302, 303, 305, 307, and 308 Equipment and Material price list
Selectron Process Selectrons Ltd. 116 E. 16th Street New York, N.Y. 10003 Vanguard Pacific Inc. 1655 Ninth Street Santa Monica, Calif. 90406	Technical Instruction Manuals SI-115 and SI-130 Technical Bulletins SL-81, SL-82, SP-1023 and Navy-Fact File Selectron "Plating Guide" slide rule Equipment and Material price list

*Publications may be obtained on request.

Process Control

All parts to be plated will be handled in accordance with a written Process Control Procedure approved by the individual activity. Plating work should be set up to ensure a smooth flow of plating work from initial engineering approval through final inspection.

Adequate records must be kept of work performed by the plating shop. Processing information recorded should include the following:

1. Name of ship, date and job order number when applicable.
2. Description of part to be plated by proper name and drawing piece numbers.
3. A sketch of the area requiring plating.
4. Identification of base metal.
5. Final thickness of deposit required.
6. Plating material(s) to be used.
7. Step by step processing procedure.
8. Method of surface finishing (grinding, honing, etc.)
9. Final inspection, including method and dimensional checks when applicable.

Items 1 through 6 above should be engineering and job planning functions and represent the minimum information required by the plating shop.

Process control records of completed work are a ready reference for handling repeat jobs and for assessing the capability of the plating shop.

General (All Plating)
Inspection Procedure

• Visual Inspection: All platings must be smooth and free of blisters, pits, nodules, porosity, excessive edge buildup, and other defects which will affect the functional use of the plated part. The finished plating must conform to the required design surface finish for the part and must be free of burnings and stress concentrations. Burning is defined as rough, coarse grained, or dull plates caused by localized high current density or arcing. Highly stressed deposits are normally indicated by cracks or crazing.

• Adhesion Test: The adhesion test is performed with Scotch #250 tape or an equivalent high tack strength pressure sensitive tape as follows:

1. Thoroughly clean and dry the plated surface.
2. Cut a piece of 1 inch wide unused tape approximately 6 inches longer than the width of the plated area.
3. Stick the tape across the width of the plated area so that approximately 1 1/2 inches of the base metal on each side is taped. Tamp the tape down to ensure that it sticks thoroughly.
4. Grip the loose end of the tape and rip rapidly upward (at right angle to the plating) removing the tape with a single jerk.
5. Inspect the tape. Any plating adhering to the tape is cause for rejection.

Platings for Rubbing Contact Service

In addition to the general inspection, platings for rubbing contact service must meet the liquid penetrant inspection.

• Liquid Penetrant Inspection: The finished plating will be inspected with Group I liquid penetrant in accordance with MIL-STD-271. Indications must not be greater than 1/16 inch and the concentration of indications must not exceed 3 in any square inch area. For chromium plating only, because of the inherent crazying characteristic of the material, water washable penetrant material (Group III or IV of MIL-STD-271) may be used for liquid penetrant inspection.

POWER PACK COMPONENTS

The equipment must contain safety features in accordance with MIL STD 454. Operations that could create personnel hazards or result in damage to the equipment or work will be noted on a caution plate permanently attached to the front of the equipment.

The parts of the power pack–ammeter, d.c. circuit breakers, voltmeter, ampere-hour meter, start and stop buttons, output terminals, forward-reverse switch, output leads–are discussed below and labeled in figure 14-11.

AMMETER

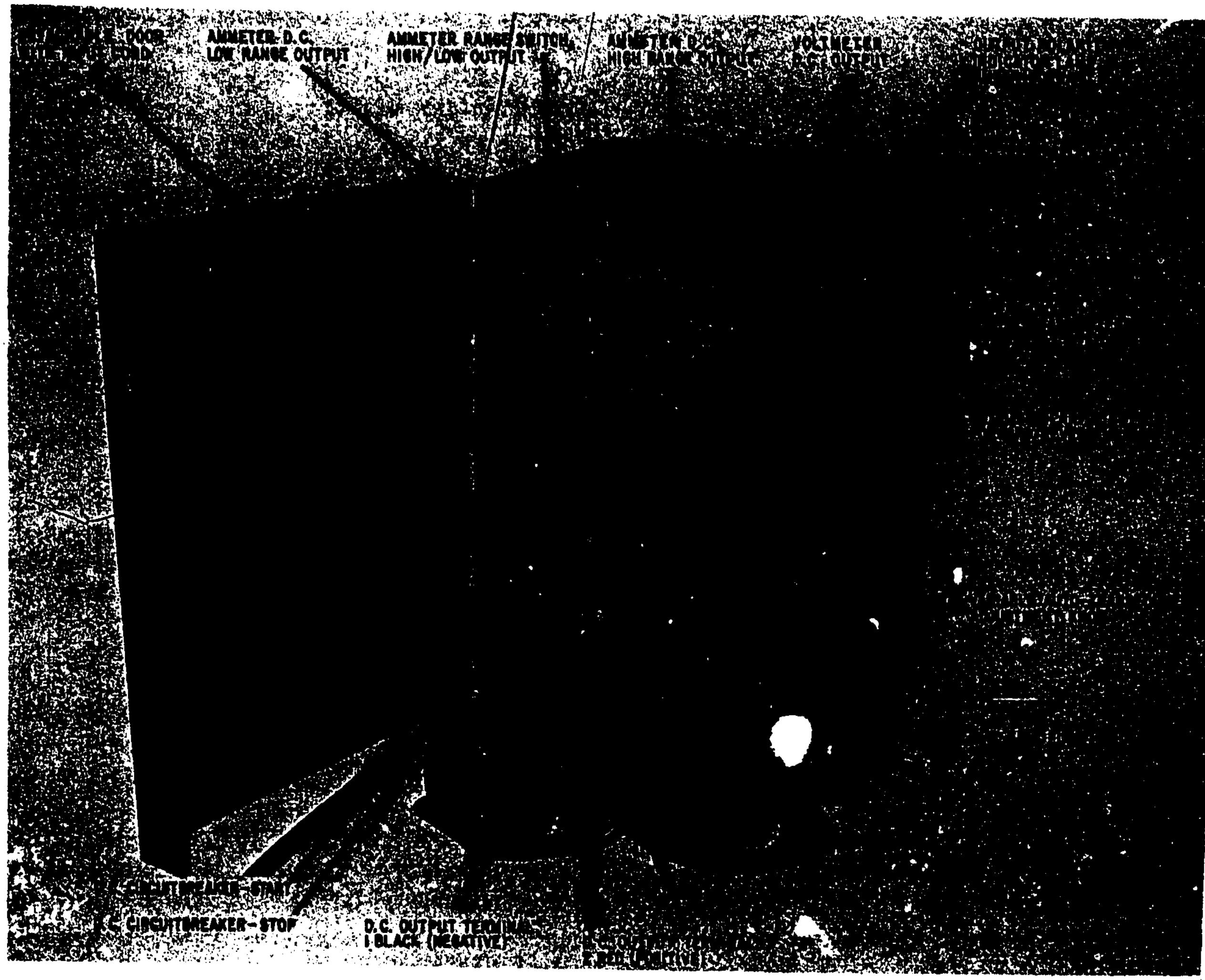
There is at least one ammeter on the power pack. The ammeter measures the rate of current flow through the plating tool. Since the rate at which metal is being applied is exactly or nearly proportional to the rate of current flow, the ammeter gives a second to second idea of how fast you are plating.

D.C. CIRCUIT BREAKERS

All power packs have at least one d.c. circuit breaker. Its purposes are to prevent overloading the power pack and to minimize damage to the workpiece in case there is an accidental direct shorting of a lead or a tool on the workpiece.

VOLTMETER

The voltmeter measures the voltage (electrical pressure) applied across the d.c.



28.449

Figure 14-11.—Dalic power packs—digital display.

circuit or through the solution. Different voltage ranges are used with different solutions. The "volts" control knob makes the adjustments for applied voltage, which is the initial step in obtaining the proper plating conditions.

AMPERE-HOUR METER

The ampere-hour meter measures the quantity (amps X time) of current passed through the d.c. circuit and allows control of the thickness of deposits. The formula for determining ampere-hours will be discussed later in this chapter. The meter also has a zero reset. The reset button is pushed after cleaning, etching, etc. are finished. When the computed amp-hours are passed, the plating operation has been completed. The white dot below the numbers indicates the decimal point; example 0012.61 means 12.61 amp-hours have been passed.

START BUTTON

The start button energizes the circuit breaker and makes the d.c. circuit operative.

STOP BUTTON

The stop button deenergizes the d.c. circuit and makes it inoperative.

OUTPUT TERMINALS

Each power pack has at least one black and one red output terminal. Larger power packs have a number of black and red terminals, sometimes of various sizes. Plating tool leads, usually color coded red, are always connected to a red terminal. The alligator clamp lead, usually color coded black, is always connected to a black terminal. A lead can be connected to any terminal if the color and size are compatible.

FORWARD-REVERSE SWITCH

The forward-reverse switch changes the direction of current flow in the d.c. circuit.

OUTPUT LEADS

Larger power packs have a number of wire leads of different sizes. Small leads are correlated with small terminals for small tools where low amperages will be drawn. Large size wire leads are correlated with large terminals for large tools where high currents will be drawn.

SELECTION OF POWER PACK

The power pack size is determined by the solution used and the plating tool contact area. Use Table 14-3 in selecting the size. It lists (1) the plating tool contact area desirable with a given solution and power pack and (2) the power pack size required for a given solution and plating tool contact area.

EXAMPLES IN USING TABLE 14-3:

a. You are to use a 60-35 power pack and code 2050 solution on a given job. If possible, you should use a plating tool that gives 20 square inches of contact area.

b. You are to use code 2080 solution on a job where the contact area is up to 5 square inches. Use a 30-25 power pack or larger on this job.

OPERATION OF POWER PACK

PRIOR TO PLATING, take the following steps on the power pack you will use:

1. If the power pack has an external ground post, connect the post with sufficient size wire to a suitable ground.
2. If necessary, connect the suitable male connector to the a.c. input line, and connect to proper (volts and phase) a.c. power supply.
3. Turn the "volts" control to extreme "low" position.
4. Connect the appropriate size output leads for the plating tools you will use to the appropriate terminals on the power pack. (Black alligator clamp lead to black terminal; red plating tool lead to red terminal.)

DURING PLATING OPERATION.

- Press the "start" button to energize d.c. circuit.

- Adjust the "volts" control and the "forward-reverse" switch as necessary for various preparatory and plating steps.

- Press the amp-hour meter button to reset the indicator to zero just prior to plating.

- When plating is completed, press the "stop" button to deenergize d.c. circuit.

PLATING TOOLS–SELECTION AND PREPARATION

Selection and preparation of the proper preparatory and plating tools is a VERY IMPORTANT factor in determining how rapidly and effectively you carry out a particular job. In plating operations (preparation of the surface or plating), work is done only where and when the

Table 14-3.—Optimum Plating Tool Contact Areas (sq in.) for Various Plating Solution and Power Packs

Solution	Code	Power Packs: 10-15 10 Amp	15-20 15 Amp	30-25 30 Amp	60-35 60 Amp	100-40 100 Amp	150-40 150 Amp	200-25 200 Amp
Antimony	2000	2	6	12	24	40	60	80
Bismuth	2010	4	10	20	40	67	100	134
Cadmium	2020	1	3	5	10	17	25	34
Cadmium	2021	3	8	15	30	50	75	100
Cadmium	2022	2	5	9	18	29	43	57
Cadmium	2023	2	4	8	15	25	38	50
Chromium	2030	1	3	5	10	17	25	34
Chromium	2031	3	8	15	30	50	75	100
Cobalt	2043	1	2	5	9	15	22	29
Copper	2050	2	5	10	20	34	50	67
Copper	2051	2	5	9	18	29	43	57
Copper	2052	2	5	9	18	29	43	57
Copper	2054	1	4	7	14	23	34	45
Copper	2055	0.5	2	3	5	8	12	16
Iron	2061	1	3	5	10	17	25	34
Lead	2070	3	8	15	30	50	75	100
Lead	2071	3	8	15	30	50	75	100
Nickel	2080	1	3	5	10	17	25	34
Nickel	2085	1	2	5	9	15	22	29
Nickel	2086	1	3	6	12	20	30	40
Nickel	2088	1	3	5	10	17	25	34
Tin	2090	3	8	15	30	50	75	100
Tin	2092	3	8	15	30	50	75	100
Zinc	2100	2	5	10	20	34	50	67
Zinc	2101	1	2	5	9	15	22	29
Zinc	2102	1	2	5	9	15	22	29
Zinc	2103	1	2	5	9	15	22	29
Gallium	3011	4	10	20	40	67	100	134
Gold	3020	4	10	20	40	67	100	134
Gold	3021	4	10	20	40	67	100	134
Gold	3022	4	10	20	40	67	100	134
Gold	3023	20	60	120	240	400	600	800
Indium	3030	3	8	15	30	50	75	100
Palladium	3040	2	5	10	20	34	50	67
Platinum	3052	1	3	5	10	17	25	34
Rhenium	3060	2	5	10	20	34	50	67
Rhodium	3072	2	5	10	20	34	50	67
Rhodium	3074	3	8	15	30	50	75	100
Silver	3080	2	4	8	15	25	38	50
Silver	3081	5	15	30	60	100	150	200
Silver	3082	2	6	12	24	40	60	80
Silver	3083	2	6	12	24	40	60	80
Nickel-Cobalt	4002	1	3	5	10	17	25	34
Tin-Indium	4003	3	8	15	30	50	75	100
Tin-Lead-Nickel	4005	4	10	20	40	67	100	134
Cobalt-Tungsten	4007	1	3	5	10	17	25	34
Nickel-Tungsten	4008	1	3	5	10	17	25	34
Babbitt-SAE 11	4009	5	15	30	60	100	150	200
Babbitt-Soft	4010	5	15	30	60	100	150	200
Babbitt-Navy #2	4011	5	15	30	60	100	150	200

tool meets the part. Rapid, proper, and uniform processing of a part largely depends on:

1. Whether the tool you select covers a sufficient or optimum contact area on the part.
2. Whether the tool covers the full length of an inside diameter, outside diameter, or flat area.
3. How you pump the solution through the plating tool when plating higher thicknesses on larger areas.

PREPARATORY TOOLS

The preparatory steps (cleaning, deoxidizing, etching, etc.) are relatively short steps as compared to the plating operation. Selection of the preparatory tools, therefore, is not as critical as for the plating tool. The preparatory tools, however, should contact approximately 10% or more of the area to be plated, and, if easily practicable, cover the full length of the area to be plated to assure uniform preparation.

Sufficient solution is supplied by dipping for solution. In most cases, a standard plating tool will meet the above requirements and special preparatory tools need not be made.

Proper Plating Tools

The plating step generally represents the major part of a complete plating operation and selection of the proper plating tool is, therefore, more critical than for the preparatory tools. The higher the thickness of plating to be applied, the larger the area to be plated and the larger the number of parts to be plated, the more important it is to have the proper tool. It is also important to have the proper tool when uniformity of deposit thickness is necessary.

Optimum Contact Area for Plating Tool

A tool that gives optimum contact area on the area to be plated lets you plate a good deposit as fast as possible. Optimum contact area depends on the power pack to be used, the solution to be used, the size of the area to be plated, and the shape of the area to be plated.

In determining the optimum contact area, refer to table 14-3 which gives the maximum contact area required for a given solution to be plated and the power pack to be used.

If, for example, Code 2080 solution is to be used with a 60-amp power pack, the maximum contact area required is 10 square inches.

NOTE: FORMULAS 1 THROUGH 7 ARE DISCUSSED AT THE END OF THIS CHAPTER, BEGINNING ON PAGE 14-65

Formula 3 can be used alternately to obtain this value. The maximum contact area is required on very large areas only. On very small areas optimum contact area is maximum contact that can be practicably obtained; that is, full contact for flat areas and 50% of the total area for outside diameters (O.D.) and inside diameters (I.D.). In other words, optimum contact area for a flat surface is full contact up to an area the size given in table 14-7 and for larger areas it remains that size.

On O.D.'s and I.D.'s where it is usually difficult to get a tool that contacts more than 50% of the total area, optimum contact area is 50% contact area up to a contact area of the size given in table 14-3; for larger O.D.'s and I.D.'s it remains that size.

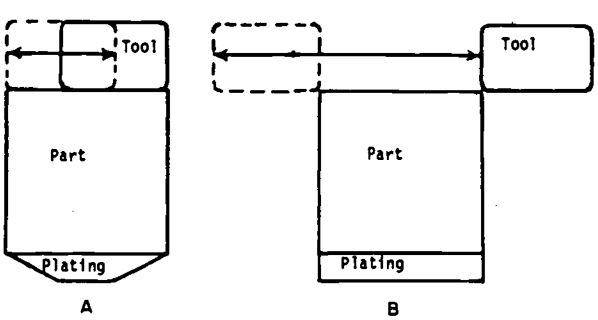
Covering Full Length

Covering the full length of an O.D., I.D., or flat surface with a tool makes it relatively easy to get a uniform thickness. When the tool does not cover the full length, problems arise. For example, take the case of trying to plate an O.D.

3 inches long with a tool that will cover only 2 inches. If you move the tool as shown in figure 14-12A, the center 1 inch is always covered, but in moving the tool to the ends there is less coverage time. The plate distribution you would get is shown at the bottom (plating). The alternative to this is to move the tool as shown in figure 14-12B. You get an even plate distribution, but now you waste some time with the tool off the part. This motion, also, may not be practical if there is a shoulder at one side. The same situation applies to I.D.'s and flat surfaces. Summarizing, always try to have the tool cover the full length of O.D. or I.D. or the full length or width of a flat surface.

Solution-Fed Tool

Solution-fed tools are used for plating high thicknesses on large areas of a large number of parts. It is, of course, not worthwhile to use a solution-fed tool when a small thickness of deposit is required on a small area of one part. Solution-fed tools are not used with precious metals, since a higher volume of a high cost solution is required. Solution-fed tools usually double plating speed and improve quality and reliability of deposit because the flowing solution (1) cools the anode, allowing higher currents to be passed; (2) ensures that sufficient fresh solution is maintained in the work area; and (3) eliminates time wasted in dipping for solution.



28.450
Figure 14-12.—Plating—covering the full length.

Use the following procedure to determine if it is worthwhile to use solution-fed tool.

1. Use formula 1 (page 14-65) to determine amp hours required for one part and then multiply by the number of parts if more than one.
2. Determine the type of tool to be used and also its contact area. Then use formula 4 (page 14-66) to determine total plating time if solution is pumped through the tool.

Since dipping for solution usually doubles plating time, the value arrived at in step 2 above also represents the extra time you will spend dipping for solution. This possible savings in time can help you determine if it is worthwhile to set up to pump solution.

Standard Tools

Standard tools (figures 14-13 and 14-14) are available for preparing and plating a wide variety of sizes and shapes of parts. These are described on the following pages. You can use standard tools if they meet the following requirements.

Preparatory Tools:

1. Cover approximately 10% or more of area to be plated.
2. Cover full length.

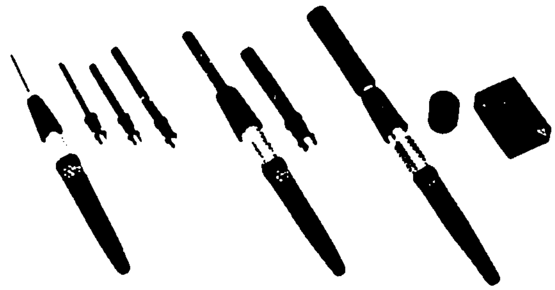
Plating Tools:

1. Provide optimum contact area.
2. Cover full length.
3. Allow for pumping solution when required.

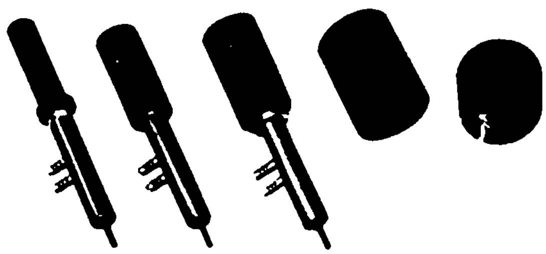
NOTE: You must allow 1/8 to 1/4 inch on the radius for the tool cover when considering standard tools for O.D.'s and I.D.'s.

Special Plating Tools

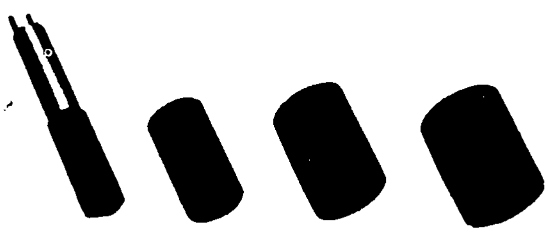
You should use special plating tools when standard plating tools will not effectively accommodate a particular area to be plated. The greater the thickness of plate desired and/or the larger the number of pieces to be plated, the



AC - SERIES



WC - SERIES



TOOL Cat. No.	Handle	COMPONENTS Anode	Adapter
AIR-COOLED (for small areas and I.D.'s) Solution Dip			
AC - 0	AC 1-3	AC-0* .09 ø x 2"	. . .
AC - 1	AC 1-3	AC-1 .180 ø x 2.25"	. . .
AC - 2	AC 1-3	AC-2 .25" ø x 2.25"	
AC - 3	AC 1-3	AC-3 .31" ø x 2.25"	. . .
AC - 4	AC 4-7	AC-4 .375" ø x 3"	. . .
AC - 5	AC 4-7	AC-5 .5" ø x 3"	. . .
AC - 6	AC 4-7	AC-6 .75" ø x 3.5"	AC
AC - 7	AC 4-7	AC-7 1.0" ø x 1"	AC
AC - 8	AC 4-7	AC-8 2.25" x 1.5" x .75"	AC
AC - 9	AC 4-7	AC-9 2" x 3" x .75"	AC
WATER-COOLED (for larger I.D.'s) Solution Dip			
WC - 25	WC - 25	WC-25 1.125" ø x 3.75"	FG
WC - 40	WC - 40	WC-40 1.625" ø x 3.75"	FG
WC - 55	WC - 75	WC-55 2.125" ø x 3.75"	FG
WC - 70	WC - 75	WC-70 3" ø x 3.75"	FG
WC - 75	WC - 75	WC-75 3.125" ø x 2.125"	FG
SOLUTION FED (for larger I.D.'s)			
RF - 15	F	RF-15 1.5" ø x 3.75"	FG
RF - 20	F	RF-20 2" ø x 3.75"	FG
RF - 25	F	RF-25 2.5" ø x 3.75"	FG
RF - 30	F	RF-30 3" ø x 3.75"	FG

*AC - 0 ANODE is platinum clad titanium

28.451

Figure 14-13.—Standard plating tools.

SCC - SERIES / SCG - SERIES

FG - SERIES / FF - SERIES

TOOL Cat. No.	Handle	COMPONENTS Anode	Adapter
SOLUTION FED (for O.D.'s)			
SCC-10	AC 4-7	SCC-10 1" I.D. x 1" wide	AC
SCC-15	AC 4-7	SCC-15 1.5" I.D. x 1" wide	AC
SCC-20	AC 4-7	SCC-20 2" I.D. x 1" wide	AC
SCC-25	AC 4-7	SCC-25 2.5" I.D. x 1" wide	AC
SCG-25	G	SCG-25 2.5" I.D. x 2" wide	FG
SCG-30	G	SCG-30 3" I.D. x 2" wide	FG
SCG-35	G	SCG-35 3.5" I.D. x 2" wide	FG
SCG-40	G	SCG-40 4" I.D. x 2" wide	FG
FLAT & MULTI-PURPOSE TOOLS Solution Dip			
FG - 1	G	FC-1 2.5" x 2.5" x 1"	FG
FG - 2	G	FG-2 3.5" x 3.5" x 1"	FG
FG - 3	G	FG-3 4.5" x 4.5" x 1"	FG
SOLUTION FED			
FF - 1	F	FF-1 2.5" x 2.5" x 1"	FG
FF - 2	F	FF-2 3.5" x 3.5" x 1"	FG
FF - 3	F	FF-3 4.5" x 4.5" x 1"	FG
FF - 4	F	FF-4 4" x 3" x 2"	FG
FF - 5	F	FF-5 6" x 4" x 2"	FG

NOTE: All anodes except AC-0 are made of special grades of graphite. Anodes of any size, shape or material can be made on short order, please inquire.

Figure 14-14.—Standard plating tools. 28.452

more desirable it is to use special tools, since there is more opportunity to offset the extra cost by savings in plating time.

1. Obtain required information on the job:

   a. Amperage output of the power pack to be used.
   b. Plating solution to be used.
   c. Shape and size of the area to be plated.

2. Determine the optimum contact area using either table 14-3 or formula 3 (see page 14-66).

3. Determine the maximum practical contact area:

   a. On flat surfaces it is the total area.
   b. On O.D.'s it is 50% of the total area since one can always cover the full length but only 50% of the circumference.
   c. On I.D.'s it is 50% of the total area since one can always cover the full length, but practically only 50% of the circumference. Attempts to get more than 50% contact on an I.D. are generally defeated by compression of the tool cover during plating.

4. When the maximum practical contact area (3 above) is less than optimum contact area (1 above) the special tools should be as follows:

   a. On flat areas the tool should be 1 or 2 inches wider than the area to be plated. This allows for moving the tool while plating.
   b. On I.D.'s and O.D.'s the tool should cover the full length and one-half of the circumference.

5. When the optimum contact area is less than the maximum practical contact area, the special tools should be designed to give the optimum contact area.

In the interest of getting a uniform thickness, the full length of an I.D. or O.D. and the smaller dimension of a rectangle is covered. This establishes one contact dimension. To get the second, divide the optimum contact area by the first dimension.

The height of the anode is not critical. It should be high enough to accommodate the handle hole and solution flow lines. If the anode is too high, it just adds to tool weight. Heights of 1 to 2 inches are generally used.

6. Select handles, solution inlet fittings, etc. based on design plating amperage. When dimensions of anodes are based on optimum contact area, the plating amperage should be the amperage output of the power pack. When the anode dimensions are based on maximum practical contact area, compute expected plating amperage using formula 4 (page 14-66).

At this point you will find a ruler and a compass helpful in sketching in the anode. Keep the following rules in mind:

- On radii for I.D. and O.D. tools, allow for the anode cover, usually 1/4 inch thick.

- Space the solution outlet holes coming out the working face of the anode at intervals of at least every 1 inch in the direction of the length of an I.D. or O.D. tool and perpendicular to the direction of tool movement on a flat surface tool. This eliminates the possibility of plating tapers through uneven solution distribution. In the other direction, they should be at least every 2 inches to ensure reasonably complete wetting of the cover and to permit passage of current throughout the cover. The outlet holes are usually 3/32 inches in diameter.

- Make the main distribution hole in the anode next to the inlet fitting at least 1/4 inch in diameter when using a small submersible pump and 1/2 inch in diameter when using a large submersible pump. This helps ensure that all outlet holes are reasonably well fed.

The following examples will help you understand how to make the special tools you may need in contact electroplating.

EXAMPLE #1:

Plate a 16-inch length on 13 inch O.D. tubing with .006 inch of nickel Code 2080. Use a 200-amp power pack. You can rotate the part in a lathe.

Optimum contact area is 34 sq in. which is less than 50% of the total area to be plated. Covering the full length of 16 inches gives one contact dimension. The contact width around the surface then is

$$CW = \frac{34}{16} = 2\ 1/8 \text{ inches}$$

Allowing for a cover thickness of 1/4 inch, a 6 3/4-inch radius will be put in the 16 inch X 2 1/8-inch face. To help keep the rather long tool squarely on the part, use two G-handles. Solution outlet holes will be slightly larger the farther away they are from the solution inlet port. (See figure 14-15.)

EXAMPLE #2:

A 12-inch I.D., 3 inches long requires 0.0035 inch of nickel, Code 2085. The part is very large and cannot be rotated. Therefore, you must move the tool by hand. Use a 100-amp power pack. The amp-hours required for the job are

$$\text{Amp-hr} = .015 \times 35 \times 113 = 59$$

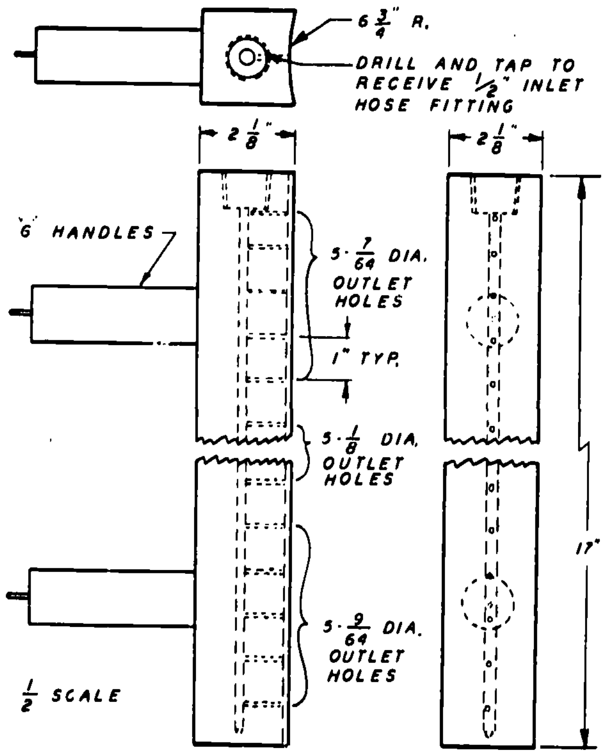
A tool such as the Rf30 will give a small contact area, draw only approximately 30 amps, and result in a plating time of 2 hours. It is elected to make a pie-shaped tool which has the

a. disadvantage of having to be rotated in addition to being moved around the I.D.

b. advantage of being able to draw 100 amps which reduces the plating time to 0.6 hours.

In view of the difficulty in moving the tool, make the tool 3 3/4 inches long to ensure full contact along the length. The bore being 3 inches long, the contact length remains 3 inches. (See fig. 14-16.) The optimum contact area is 15 sq in. The contact width then is

$$CW = \frac{15}{3} = 5 \text{ inches}$$



28.453

Figure 14-15.—Design of special tool.

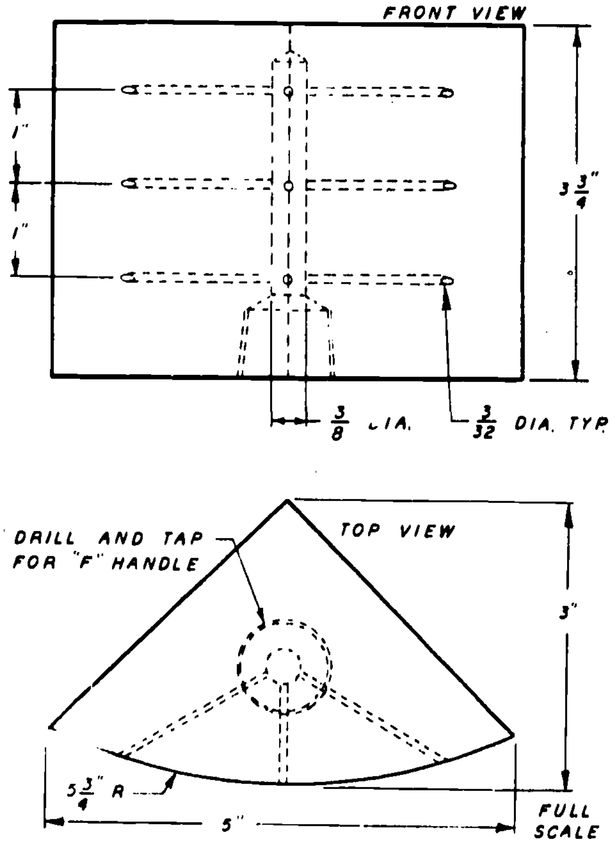
EXAMPLE #3:

Ten bearings must be plated on a 20 inch long, 26 inch I.D. with .002 inch of babbitt, Code 4009, per side. The part will be rotated in a barrel rotator, leaving the I.D. accessible from both ends. Use a 100-amp power pack.

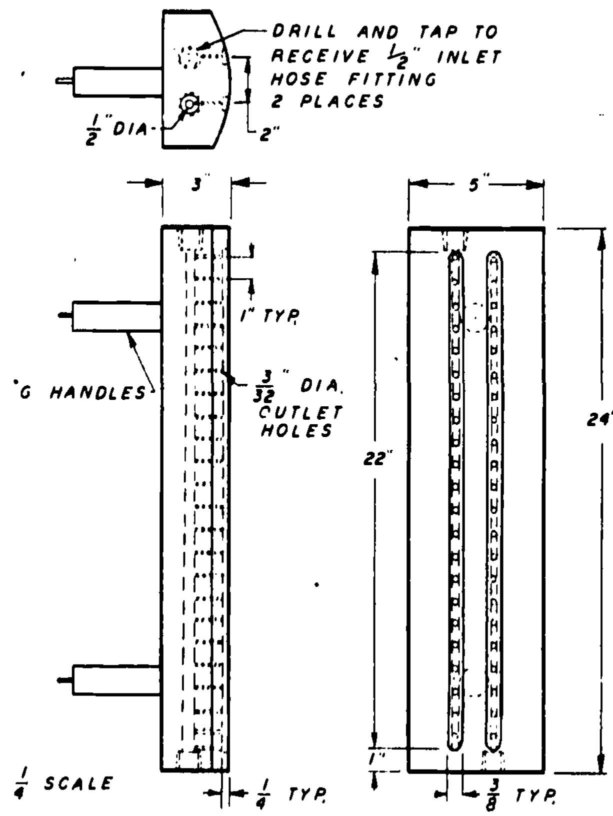
The optimum contact area is 100 sq in. Since the contact length is 20 inches, the contact width is 100/20 or 5 sq in. Solution will be pumped in from both ends to obtain more uniform solution distribution, since thickness control is critical. Use two G-handles to help keep the tool properly located on the part. Mill a channel into the anode face around the outlet hole to get better distribution along the length of the anode. (See fig. 14-17.)

PLATING TOOL ANODE MATERIALS

A grade of graphite with maximum resistance to breakage and anodic corrosion is



28.454
Figure 14-16.—Design of special tool.



28.455
Figure 14-17.—Design of special tool.

used on most standard tools and in the fabrication of special tools from block form. Other materials, however, have been used and are recommended. Check the manufacturer's instruction manual for particular applications.

Care of Anodes

Use sandpaper, Scotchbrite, etc. to remove loose graphite from the working area of graphite anodes used as part of the recovering operations. This helps keep subsequently used solutions clean. Then, thoroughly soak the anodes in clean water and wipe off the abraded area.

Thorough cleaning of the anode is particularly important when the tool will be used later with a different solution. Thorough cleaning of the anode (or use of one tool for one operation) is of maximum importance in forward "Cleaning and Deoxidizing" and "Activating" operations. Since thorough cleaning of the tool may not always be effectively accomplished, a given tool should be identified for and used with only one preparatory or plating solution.

PLATING TOOL COVERS

The plating tool cover performs several important functions:

1. It insulates the anode from the part and thereby (a) prevents damage to the part by direct shorting and (b) forces current to pass through the solution which allows electrocleaning, plating, etc. to occur.

2. It mechanically scrubs the surface being plated which permits sound deposits to be rapidly applied.

3. It holds and uniformly distributes the solution where it is needed.

TYPES OF COVERS

Several covering materials are used with the plating process and they may be categorized as follows:

INITIAL COVER: Holds and distributes solution, but requires a final cover since it is not wear resistant.

FINAL COVER: Overlay cover on an initial cover to provide wear resistance.

COMBINATION COVER: Can be used by itself since it holds and distributes solution uniformly and it has satisfactory wear resistance.

SPECIAL COVERS: Used for special effects such as described below.

The following list of plate coverings with their advantages and disadvantages will help you in selecting the proper covering for a particular plating job.

COVER	TYPE	ADVANTAGES, DISADVANTAGES, AND USES
Cotton Batting	Initial	Widely used because of its very low cost and excellent absorbency and purity. Cannot be used with chromium, Code 2031, and copper, Code 2055. Requires final cover for wear resistance.
Dacron Batting	Initial	Used very little because of its high cost relative to cotton batting. Used as a replacement for cotton batting with very corrosive solutions such as chromium 2031 and copper 2055.
Cotton Tubegauze	Final	Used to moderate degree as final cover. Very low cost and high purity and absorbency. Has less wear resistance than Dacron tubegauze. Used as final cover for preparatory tools and for rhodium plating.
Dacron Tubegauze	Final	Widely used as final cover especially for plating tools where its superior wear resistance to cotton tubegauze is important. Low cost, moderate purity, and absorbency.
White Scotch-brite	Combination	Used frequently for plating tools because of its moderate cost and high purity and wear resistance. Absorbency is poor and therefore satisfactory only when solution-fed tools are used with workpiece under tool.
Dacron Felt	Combination	Used frequently for plating tool because of its excellent wear resistance, absorbency, and moderate cost and purity.
Gray Scotch-brite	Special Purpose	Used occasionally when a higher than normal thickness, i.e., 0.005 to 0.015 inch is required in a certain deposit. Keeps deposit smoother than normal since it has an abrasive which polishes as plating is proceeding. One problem in using this material is that an effect called "Plating in Cover" usually starts in approximately 10 minutes. It is the actual plating of metal in the form of a fine powder in the cover at the expense of this material

COVER	TYPE	ADVANTAGES, DISADVANTAGES, AND USES
		being applied on the workpiece. This is indicated by brightening of the surface being plated and a considerable rise in amperage at a given voltage. This in turn requires that the voltage be decreased to maintain a constant amperage. As this continues, more and more plating occurs in the cover and less occurs on the part requiring at some point replacement of the cover, sometimes several times. Replacement of the cover is usually done when the voltage has been reduced to half of the starting voltage. Replacement of the cover is ordinarily done by quickly taking off the old Scotchbrite and applying new material, pre-soaked with plating solution. This eliminates the need to prepare (clean, etch, etc.) the surface for additional plating. Cost is moderate and wear resistance is good.
Bonnet Material	Combination	Used moderately for preparatory and plating tools. Moderate in cost, wearability, and purity. High in absorbency. Not recommended with certain preparatory and plating solutions. Refer to instruction manual.
Carbon Felt	Special Purpose	Applied directly on the anode and then covered with a thin final insulating cover. The carbon felt serves as an extension of the anode, i.e., the outside surface of the anode. The felt is conductive enough to carry plating current, but not conductive enough to damage the part by shorting if the thin final cover is worn through. Two important advantages are thereby gained using the combination carbon felt and thin final cover: (1) Better throwing power into internal corners such as in O-ring grooves. (2) Less tool overheating with solutions plated at high voltages and, therefore, higher possible plating times. Cost is high and absorbency and purity are excellent.
Orlon	Special Purpose Final Cover	Used for low thickness deposits (0.001 inch or less) where an as plated surface is desired that will be brighter than one started with. There is some sacrifice of quality of deposit and adhesion.
Pellon	Special Purpose Combination Cover	Very thin wear resistant cover useful for plating small I.D.'s, grooves, etc. where conventional covers cannot be used. Absorbency is poor.

Plating tools and their covers can often be used for a number of successive operations without any maintenance being required. When plating tool covers become noticeably dirty or worn, replace them.

Plating tools with clean and unworn covers, which will be used the next day, may be tightly wrapped in a clean plastic sheet or bag. Plating tools that will not be used for several days should be re-covered. Plating solution remaining in the covers can be squeezed out and filtered for reuse.

PREPARATION OF ANODES FOR THE ELECTROPLATING PROCESS

28.457

STEP-BY-STEP WRAPPING OF SCC AND SCG SERIES ANODES

USED FOR PLATING OUTSIDE DIAMETER SURFACES

PREPARATION OF COTTON BATTING

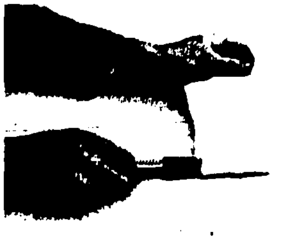
Cut a piece of cotton batting large enough to cover the concave side of the anode to be wrapped. It is important that the cotton fibers run along the longest dimension of the pad. This pad can be split into two layers for use on smaller anodes (picture 1). Thickness of the cotton used may vary according to the application. Experience has shown that 3/16" thickness works well for the average application.

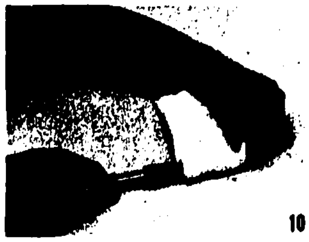
MOLD COTTON TO ANODE

Mold the cotton to the concave side of the anode (picture 2).

FASTEN TUBEGAUZE

Cut a suitable size of tubegauze (at least twice the length of the anode) and slip half the tubegauze over the anode and cover (picture 3). The remaining half of the tubegauze is then twisted (picture 4) and slipped over the anode. Two layers of tubegauze cover are thus provided, the ends of which are secured with rubber bands or tubegauze ties around the base of the Dalic Plating Solution flow tube (picture 5). Cut a hole in the tubegauze for the Dalic tool handle and insert the handle (pictures 6 and 7). The finished tool should have a smooth concave surface (picture 8).


9


10

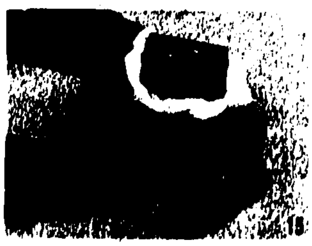

11


12


13

28.458

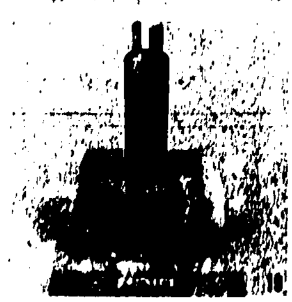

14


15


16


17


18


19

STEP-BY-STEP WRAPPING OF AC, WC AND RF SERIES ANODES

USED FOR PLATING INSIDE DIAMETER AND FLAT SURFACES

PREPARATION OF COTTON BATTING

Cut a piece of long-fiber cotton batting about one inch wider than the length of the Dalic anode and six to eight times longer than the diameter. Split the cotton to about 3/32" thickness so that the final cover thickness after rolling will be 3/16". Lay the cotton on a table and wet the anode with water so that it will adhere to the end of the cotton (picture 9).

FOLD ENDS EVENLY

Fold the protruding end of the cotton evenly over the tip of the anode (picture 10).

WRAP TIGHTLY

Wrap the cotton around the anode tightly by rolling from one end to the other. Feather the ends of the cotton so that the long fibers can be intertwined (picture 11).

TUBEGAUZE SECURES COTTON WRAP

The application of tubegauze provides maximum wear resistance and prevents cutting through on sharp edges. Apply the tubegauze as on a finger (picture 12). The finished wrapping should be neat and compact with no bulges or thin spots (picture 13).

STEP-BY-STEP WRAPPING OF FG AND FF SERIES ANODES

USED FOR PLATING FLAT AND OTHER SURFACES

FOLD COTTON AROUND ANODE

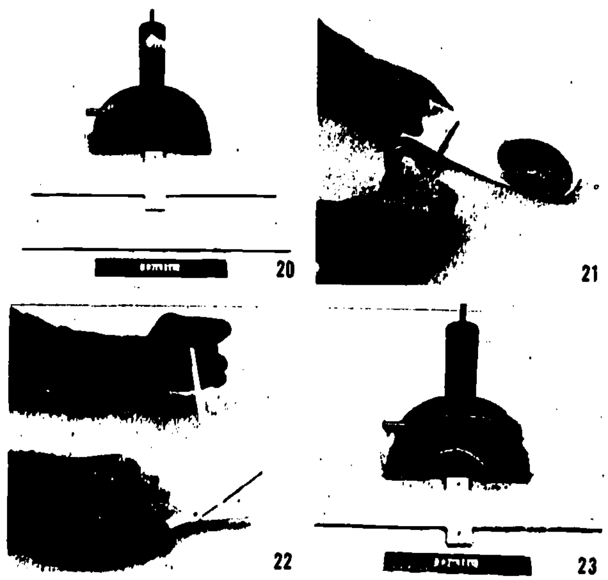
The long-fiber cotton pad for the FG and FF anodes should be cut to provide a 1/2 inch overlap around the anode. Place the anode on the cotton, making sure that the length of the cotton fibers run in the direction of the long side of the anode (picture 14). Fold the cotton evenly around the anode and keep the bottom surface smooth (picture 15).

INSERT INTO TUBEGAUZE

Holding the wrapped anode by the bottom to keep the cotton smooth, insert it into a piece of tubegauze of appropriate size (picture 16). Secure the ends of the tubegauze tightly by twisting them and binding them with rubber bands or tubegauze ties (picture 17).

CUT HOLE FOR HANDLE

Cut a hole in the tubegauze large enough to screw the Dalic tool handle into the anode (picture 18). The fully wrapped FG or FF anode should have a smooth even pad of cotton on the bottom, secured tightly by the tubegauze (picture 19).



20 21 22 23
28.459

STEP-BY-STEP WRAPPING OF SCC AND SCG ANODES

WITH SCOTCHBRITE, DACRON FELT AND SIMILAR MATERIALS

USED FOR PLATING OUTSIDE DIAMETER SURFACES

PREPARATION OF SCOTCHBRITE

Cut a piece of Scotchbrite 1/4-1/2" wider than the anode and long enough to cover the concave side of the anode and partway up the end. Punch holes for tubegauze ties (Picture 20).

MAKE TIES

Cut tubegauze ties (#.5 Dacron is best) as shown (Pictures 21 - 22).

TIE COVER TO TOOL

Secure cover to tool with ties (Picture 23). It may be necessary to make a cover with "ears" in some applications where a more secure cover is required.

24 25
28.460

STEP-BY-STEP WRAPPING OF FG, FF AND SOME SPECIAL ANODES WITH SCOTCHBRITE, DACRON FELT AND SIMILAR MATERIALS

USED FOR PLATING FLAT AND OTHER SURFACES

PREPARE SCOTCHBRITE AND TIES

Cut a piece of Scotchbrite 1/4-1/2" wider than the anode and long enough to cover the working surface and extend onto the top. Punch holes and make tubegauze ties (Picture 24).

TIE COVER TO TOOL

Secure cover to tool with ties (Picture 25).

STORAGE AND SHELF LIFE OF SOLUTIONS

The shelf life of the majority of solutions is unlimited. Each manufacturer can be consulted for specific information. As a general rule, solutions should be stored at room temperature away from light. Excess cold, in storage or in transit, may lead to "salting out," that is, formation of solid crystals at the bottom of the container. These solutions may be restored to full effectiveness by heating to approximately 140°F and stirring until all salted out material is redissolved.

Used plating solution should be returned to used plating solution bottles along with a log of the ampere-hours passed through the solution. This will provide some idea of how heavily the solution has been used. The used solution is best used on less critical applications requiring lower thicknesses of deposits.

A small percentage of dilution of solutions (from rinse water on the part or evaporation of water while in use) will have no effect on the solutions. When dilution reaches 25% the solution should be discarded.

MASKING

Masking performs several functions in the electroplating process: It prevents plating from being applied on areas where it is not wanted. It gives a definite size area that is being plated which permits more accurate thickness control. It reduces waste of metal from the plating solution. It reduces possibility of contaminating the solution.

Masking tapes are generally used to mask off areas immediately adjacent to the area being plated. The materials used include vinyl tape, polyester tape, aluminum tape and copper tape. Absorbent tapes, such as painter's masking tape, should not be used since they can lead to cross contamination of the solution.

Masking must be done more carefully when a corrosive solution is used on reactive base material or when considerable heat will be developed in plating.

Careful masking includes:

1. Careful cleaning of the surface before tape is applied.

2. Pressing in tape where a second layer of tape rises to cover a preceding layer of tape.

3. Applying vinyl tape on surfaces such as I.D.'s with no tension, since vinyl tape tends to spring back.

Vinyl tapes are ordinarily used for most solutions. However, there are exceptions listed by individual vendors. Consult your instruction manual for particular solutions where it cannot be used.

Use aluminum tape on demanding masking jobs such as when using corrosive solutions, when plating with solutions that develop heat, and when masking difficult areas such as I.D.'s. Aluminum tape has an excellent adhesive and is strong and ductile. It will stay when carefully pressed down. Vinyl tape may then be used on top, and it will stay better since it is on a fresh, clean surface.

A masking technique that offers a number of advantages is to mask with aluminum tape and then mask off a larger area with a nonconductive tape such as vinyl. A 1/8 to 1/4-inch band of aluminum tape is thus left exposed. The aluminum tape, being conductive, will in a minute or so start taking plating. The first traces of burning and high buildup will then occur at the nonconductive masked edge on the aluminum tape. The area of interest, therefore, will have no buildup and is less likely to be burned at the edges.

Large areas occasionally must be masked off to prevent corrosive solutions from attacking the part and to prevent solution contamination. In these cases, tapes are used on immediately adjacent areas; areas farther away are masked with (1) "Contact Paper" which comes in 18-inch wide rolls, (2) quick drying acrylic spray paints, (3) vinyl drop cloth, or (4) "Orange Paint" which is a tough adherent, heat resistant brush-on type paint.

SETTING UP JOB–LONGER RANGE PREPARATIONS

This section deals with how to properly make the longer range preparations to carry out a job. It includes recommendations on selecting and assuring that the proper solutions, power pack, preparatory tools, plating tools, etc. are available. The material is arranged in a step by step manner developed by past practical experience.

It is assumed that a basic installation is available including a power pack. Steps 1 through 7, however, can be used to select an appropriate installation including a power pack or to assure that an appropriate installation has been purchased.

Step #1 Obtain necessary information on job including:

a. Number of parts to be done.
b. Material on which deposit will be applied. In most cases, it will be the material from which the part is made. If the part, however, has had a surface treatment such as an electroplate, carburizing, etc., plating is to be applied on the surface material and not what is underneath.
c. Area to be plated; that is, have a concrete idea of size and shape of area to be plated.
d. Purpose and requirements of deposit; that is, why coating is being applied and what it is expected to do.
e. General idea of what is adjacent.
f. Thickness of deposit required.

Step #2 Selecting plating solution to be used.

This is, in most cases, an extremely important step. Proper selection assures that the desired results will be obtained with maximum ease and minimum cost. In many cases, the pure metal or alloy will have already been chosen either by a specification or blueprint; in other cases, the metal or alloy will be obvious, such as cadmium would be used to touch up a defective cadmium deposit. In these cases, if there is a choice of solutions, only the selection of the proper specific solution remains. There are other cases where a particular metal or alloy is not specified or obvious such as in salvage or repair. Tables 14-4, 14-5, 14-6, 14-7 and 14-8 have been prepared to assist in both instances. Review these tables carefully before making a selection.

Step #3 Calculate amp-hours using formula 1, page 14-65.

Step #4 Decide on the general approach to the plating job:

a. Whether the part will be rotated, or whether the tool will be moved by hand.
b. Whether the solution will be pumped or dipped.

Step #5 Decide on what type of plating tool you will use, whether a standard tool or special tool. If you will use a special tool, determine the design. (See figures 14-15, 14-16, and 14-17.)

Step #6 Based on plating tool to be used, determine contact area if not determined in Step #5.

Step #7 Based on contact area, determine the plating current if not established in Step #5. Use formula 4, page 14-66.

Step #8 Determine the plating time using formula 5, page 14-66. If the solution will be dipped for, double the plating time.

Step #9 Determine the amount of plating solution necessary, using formula 6,

Table 14-4.—Characteristics of Solutions and Deposits

Plating Solution	Code	Normal Maximum Thickness In One Layer (In.)	Ease in Using Solution	Ease in Reactivating Deposit	Corrosion Tendency in Base Materials	Special Toxicity Problems
Antimony	2000	-------	Very Difficult	----------	Low	Toxic Metal
Bismuth	2010	-------	Very Difficult	----------	Low	None
Cadmium	2020	.007	Easy	Very Easy	Some	Toxic Metal
Cadmium	2021	.005	Easy	Very Easy	Low	Toxic Metal
Cadmium	2022	.007	Very Easy	Very Easy	Low	Toxic Metal
Cadmium	2023	.005	Easy	Very Easy	Low	Toxic Metal
Chromium	2030	.002	Difficult	Very Difficult	Low	Toxic Metal
Chromium	2031	.0005	Very Difficult	Very Difficult	Some	Toxic Metal
Cobalt	2043	.008	Easy	Average	Low	None
Copper	2050	.015	Very Easy	Very Easy	High	Very Acidic
Copper	2051	.006	Average	Difficult	Low	None
Copper	2052	.004	Easy	Difficult	Low	None
Copper	2054	.015	Easy	Easy	High	Very Acidic
Copper	2055	.012	Easy	Easy	High	Very Acidic
Iron	2061	.007	Average	Average	Low	None
Lead	2070	.007	Very Easy	Easy	Low	Toxic Metal
Lead	2071	.007	Very Easy	Easy	Low	Toxic Metal
Nickel	2080	.007	Average	Average	Low	None
Nickel	2085	.015	Easy	Easy	Low	None
Nickel	2086	.007	Average	Average	Low	None
Nickel	2088	.007	Average	Average	Low	None
Tin	2090	.007	Very Easy	Easy	Low	None
Tin	2092	.007	Very Easy	Easy	Low	None
Zinc	2100	.003	Easy	Easy	Low	Toxic Metal
Zinc	2101	.008	Very Easy	Very Easy	Low	Toxic Metal
Zinc	2102	.006	Very Easy	Very Easy	Low	Toxic Metal
Zinc	2103	.012	Very Easy	Very Easy	Low	Toxic Metal
Gallium	3011	-------	Average	----------	Low	None
Gold	3020	.007	Easy	Very Easy	Low	Has Cyanide
Gold	3021	.007	Easy	Very Easy	Low	Has Cyanide
Gold	3022	.007	Easy	Very Easy	Low	Has Cyanide
Gold	3023	.001	Easy	Very Easy	Low	Has Cyanide
Indium	3030	.010	Very Easy	Very Easy	Low	None
Palladium	3040	.005	Easy	Average	Low	None
Platinum	3052	.005	Average	Easy	Low	Very Acidic
Rhenium	3060	.0001	Very Difficult	----------	Low	Very Acidic
Rhodium	3072	.002	Difficult	Average	High	Very Acidic
Rhodium	3074	.001	Difficult	Average	High	Very Acidic
Silver	3080	.005	Average	Easy	Some	Has Cyanide
Silver	3081	.007	Average	Easy	Some	Has Cyanide
Silver	3082	.010	Very Easy	Easy	Low	Has Cyanide
Silver	3083	.010	Very Easy	Easy	Low	Has Cyanide
Nickel-Cobalt	4002	.007	Average	Average	Low	Toxic Metal
Tin-Indium	4003	.007	Very Easy	Very Easy	Low	Toxic Metal
Tin-Lead-Nickel	4005	.015	Very Easy	Very Easy	Low	Toxic Metal
Cobalt-Tungsten	4007	.005	Difficult	Difficult	Low	Toxic Metal
Nickel-Tungsten	4008	.005	Difficult	Difficult	Low	Toxic Metal
Babbitt SAE 11	4009	.010	Very Easy	Very Easy	Low	Toxic Metal
Babbitt-Soft	4010	.010	Very Easy	Very Easy	Low	Toxic Metal
Babbitt-Navy #2	4011	.010	Very Easy	Very Easy	Low	Toxic Metal

Table 14-5.—Data on Solutions

Plating Solution	Code	Metal Content G/L	Factor	Current Density A/In² Nor. Max.	Current Density A/In² Avg.	Plating Rate In/Hr Nor. Max.	Plating Rate In/Hr Avg.	Yield %	Solution Required per Cu. In Lit.	Solution Required per Cu. In Gal.	Cost $ per Cu. In. Liter Price
Antimony	2000	80	.008	5	2.5	.062	.031	50	2.7	.72	128
Bismuth	2010	70	.008	3	1.5	.038	.019	50	4.6	1.21	287
Cadmium	2020	160	.007	12	6	.172	.086	40	2.2	.56	109
Cadmium	2021	70	.007	4	2	.057	.029	50	4.1	1.07	216
Cadmium	2022	110	.007	7	3.5	.100	.050	50	2.6	.68	139
Cadmium	2023	100	.007	8	4.0	.114	.057	50	2.8	.75	150
Chromium	2030	30	.137	12	6	.009	.005	7	56.0	14.8	2,974
Chromium	2031	150	.120	4	2	.003	.002	5	15.7	4.15	874
Cobalt	2043	80	.020	14	7	.070	.035	33	5.4	1.44	242
Copper	2050	60	.013	6	3	.046	.023	50	4.9	1.29	94
Copper	2051	60	.013	7	3.5	.054	.027	66	3.7	.97	98
Copper	2052	60	.013	7	3.5	.054	.027	66	3.7	.97	102
Copper	2054	60	.013	9	4.5	.069	.035	50	4.9	1.29	
Copper	2055	145	.013	25	12.5	.192	.096	25	4.1	1.07	116
Iron	2061	50	.018	12	6	.067	.033	12.5	20.6	5.45	864
Lead	2070	100	.006	4	2	.067	.033	50	3.7	.98	98
Lead	2071	100	.006	4	2	.067	.033	50	3.7	.98	106
Nickel	2080	110	.021	12	6	.057	.029	16.6	8.0	2.1	338
Nickel	2085	50	.015	14	7	.093	.047	50	5.8	1.54	160
Nickel	2086	40	.025	10	5	.040	.020	37.5	9.7	2.57	445
Nickel	2088	55	.021	12	6	.057	.029	30	8.8	2.34	
Tin	2090	80	.007	4	2	.057	.029	50	3.0	.79	172
Tin	2092	80	.007	4	2	.057	.029	50	3.0	.79	166
Zinc	2100	100	.011	6	3	.055	.027	50	2.3	.62	56
Zinc	2101	75	.011	14	7	.127	.064	50	3.1	.82	75
Zinc	2102	100	.011	14	7	.127	.064	40	2.9	.77	75
Zinc	2103	80	.011	14	7	.127	.064	50	2.9	.77	84
Gallium	3011	30	.015	3	1.5	.020	.010	50	6.5	1.71	
Gold	3020	100	.006	3	1.5	.050	.025	50	6.3	1.67	
Gold	3021	98	.006	3	1.5	.050	.025	50	6.5	1.71	
Gold	3022	90	.006	3	1.5	.050	.025	50	7.0	1.86	
Gold	3023	25	.007	.5	.25	.007	.004	50	25.3	6.7	
Indium	3030	60	.009	4	2	.044	.022	50	4.0	1.05	
Palladium	3040	30	.017	6	3	.035	.018	50	13.1	3.47	
Platinum	3052	50	.150	12	6	.008	.004	20	35.2	9.3	
Rhenium	3060	20	.750	6	3	.001	.0005	33	52.2	13.8	
Rhodium	3072	50	.030	6	3	.020	.010	60	6.8	1.80	
Rhodium	3074	20	.030	4	2	.013	.007	50	20.4	5.4	
Silver	3080	190	.005	8	4	.160	.080	33	2.7	.72	
Silver	3081	100	.005	2	1	.040	.020	50	3.4	.91	
Silver	3082	100	.005	5	2.5	.100	.050	50	3.4	.91	
Silver	3083	100	.005	5	2.5	.100	.050	50	3.4	.91	
Nickel-Cobalt	4002	84.2	.030	12	6	.040	.020	16.6	10.4	2.75	361
Tin-Indium	4003	73.4	.008	4	2	.050	.025	50	3.3	.86	265
Tin-Lead-Nickel	4005	84	.006	3	1.5	.050	.025	40	3.8	1.01	158
Cobalt-Tungsten	4007	80	.020	12	6	.060	.030	12.5	14.5	3.83	640
Nickel-Tungsten	4008	123	.025	12	6	.048	.024	10	11.9	3.13	487
Babbitt SAE 11	4009	[illegible]	.006	1	1	.017	.017	33	4.5	1.19	245
Babbitt-Soft	4010	[illegible]	.006	1	1	.017	.017	33	4.5	1.19	245
Babbitt-Navy #2	4011	[illegible]	.006	1	1	.017	.017	33	4.5	1.19	231

Table 14-6.—Properties of Deposits

Deposit	Code	Hardness Knoop	DPH	BHN	R_C	Structure Properly Plated	Ductility	Adhesion
Antimony	2000	47	40	38	--	----	Very Poor	Poor
Bismuth	2010	19	16	15	--	----	Very Poor	Poor
Cadmium	2020	25	21	20	--	No Defects	Good	Excellent
Cadmium	2021	23	20	19	--	Micro Porous	Fair	Fair
Cadmium	2022	30	26	25	--	No Defects	Good	Excellent
Cadmium	2023	27	23	22	--	Micro Porous	Fair	Fair
Chromium	2030	681	584	553	54	Micro Cracked	Not Coherent	Fair
Chromium	2031	908	778	709	63	Some Stress Cracks	Very Poor	Fair
Cobalt	2043	514	441	418	45	No Defects	Fair	Excellent
Copper	2050	165	141	134	--	No Defects	Excellent	Excellent
Copper	2051	249	213	202	(14)	No Defects	Poor	Fair
Copper	2052	244	209	198	(13)	No Defects	Poor	Fair
Copper	2054	206	177	168	(5)	No Defects	Good	Good
Copper	2055	260	223	211	(16)	No Defects	Fair	Good
Iron	2061	595	510	483	50	Some Stress Cracks	Very Poor	Excellent
Lead	2070	7	6	6	--	No Defects	Excellent	Good
Lead	2071	7	6	6	--	No Defects	Fair	Good
Nickel	2080	530	454	430	46	No Defects	Very Poor	Excellent
Nickel	2085	683	585	554	54	Micro Cracked	Very Poor	Fair
Nickel	2086	326	279	264	27	No Defects	Excellent	Excellent
Nickel	2088	400	343	325	35	No Defects	Fair	Excellent
Tin	2090	8	7	7	--	No Defects	Excellent	Good
Tin	2092	9	8	8	--	No Defects	Excellent	Good
Zinc	2100	48	41	39	--	Micro Porous	Fair	Good
Zinc	2101	61	52	49	--	No Defects	Good	Excellent
Zinc	2102	63	54	51	--	No Defects	Excellent	Excellent
Zinc	2103	55	47	45	--	No Defects	Excellent	Excellent
Gallium	3011	---	---	---	--	----	----	----
Gold	3020	148	127	120	--	No Defects	Fair	Excellent
Gold	3021	140	120	114	--	No Defects	Fair	Excellent
Gold	3022	143	123	117	--	No Defects	Fair	Excellent
Gold	3023	140	120	114	--	No Defects	Fair	Excellent
Indium	3030	2	2	2	--	No Defects	Excellent	Excellent
Palladium	3040	436	374	354	38	Micro Cracked	Not Coherent	Fair
Platinum	3052	550	471	446	47	No Defects	Fair	Good
Rhenium	3060	---	---	---	--	----	----	----
Rhodium	3072	927	795	718	64	Some Stress Cracks	Very Poor	Fair
Rhodium	3074	950	814	729	64	Some Stress Cracks	Very Poor	Fair
Silver	3080	110	94	89	--	No Defects	Very Poor	Fair
Silver	3081	163	140	133	--	No Defects	Poor	Good
Silver	3082	80	69	65	--	No Defects	Poor	Excellent
Silver	3083	142	122	116	--	No Defects	Poor	Excellent
Nickel-Cobalt	4002	543	465	441	47	No Defects	Very Poor	Excellent
Tin-Indium	4003	11	10	9	--	No Defects	Excellent	Good
Tin-Lead-Nickel	4005	9	8	8	--	No Defects	Excellent	Excellent
Cobalt-Tungsten	4007	630	540	512	52	Micro Cracked	Very Poor	Good
Nickel-Tungsten	4008	620	531	503	51	Some Stress Cracks	Very Poor	Good
Babbitt SAE 11	4009	25	21	20	--	No Defects	Fair	Good
Babbitt-Soft	4010	22	19	18	--	No Defects	Fair	Good
Babbitt-Navy #2	4011	23	20	19	--	No Defects	Fair	Good

Table 14-7.—Commonly Used Data

Plating Solution	Code	Factor	Plating Voltages: Minimum Small Tools and Areas	Plating Voltages: Maximum Large Tools and Areas	Plating Voltages: Procedures on New Jobs	Current Density A/In² Nor. Max	Current Density A/In² Avg	Solution Required per Cu. In. Lit.	Solution Required per Cu. In. Gal.	Maximum Recommended Use Amp Hrs. Per Liter	Maximum Recommended Use Amp Hrs. Per Gal.
Antimony	2000	.008	7	18	-	5	2.5	2.7	.72	29.5	111.6
Bismuth	2010	.008	2	10	-	3	1.5	4.6	1.21	17.4	66.0
Cadmium	2020	.007	6	20	6+1-1	12	6	2.2	.56	31.6	119.6
Cadmium	2021	.007	5	20	5+1-1	4	2	4.1	1.07	17.3	65.4
Cadmium	2022	.007	7	22	7+1-1	7	3.5	2.6	.68	27.1	102.8
Cadmium	2023	.007	6	20	6+1-1	8	4.0	2.3	.75	24.7	93.4
Chromium	2030	.137	8	25	8+1-1	12	6	[illegible]	14.8	24.4	92.4
Chromium	2031	.120	4	10	See 6.3.7	4	2	[illegible]	4.15	76.4	289.0
Cobalt	2043	.020	10	25	See 6.3.8	14	7	5.4	1.44	36.8	139.2
Copper	2050	.013	3	12	3+1-1	6	3	4.9	1.29	26.6	100.5
Copper	2051	.013	7	20	7+1-1	7	3.5	3.7	.97	35.4	134.0
Copper	2052	.013	4	18	4+1-1	7	3.5	3.7	.97	35.4	134.00
Copper	2054	.013	5	15	5+1-1	9	4.5	4.9	1.29	26.6	100.5
Copper	2055	.013	6	18	6+1-1	25	12.5	4.1	1.07	32.1	121.4
Iron	2061	.018	6	20	6+1-1	12	6	20.6	5.45	8.7	33.
Lead	2070	.006	6	20	6+1-1	4	2	3.7	.98	16.1	61.1
Lead	2071	.006	8	20	8+1-1	4	2	3.7	.98	16.1	61.1
Nickel	2080	.021	8	25	See 6.3.17	12	6	8.0	2.1	26.4	99.9
Nickel	2085	.015	6	20	6+1-1	14	7	5.8	1.54	25.7	97.3
Nickel	2086	.025	7	25	See 6.3.19	10	5	9.7	2.57	25.7	97.3
Nickel	2088	.021	8	25	See 6.3.20	12	6	8.8	2.34	23.7	89.9
Tin	2090	.007	5	20	5+1-1	4	2	3.0	.79	23.4	88.6
Tin	2092	.07	6	25	6+1-1	4	2	3.0	.79	23.4	88.6

Table 14-7.—Commonly Used Data—Continued

Plating Solution	Code	Factor	Plating Voltages: Minimum Small Tools and Areas	Plating Voltages: Maximum Large Tools and Areas	Plating Voltages: Procedures on New Jobs	Current Density A/In² Nor. Max	Current Density A/In² Avg.	Solution Required per Cu. In. Lit.	Solution Required per Cu. In. Gal.	Maximum Recommended Use Amp Hrs. Per Liter	Maximum Recommended Use Amp Hrs. Per Gal.
Zinc	2100	.011	8	20	8+1-2	6	3	2.3	.62	47.1	178.1
Zinc	2101	.011	9	20	9+1-2	14	7	3.1	.82	35.3	133.6
Zinc	2102	.011	9	20	9+1-2	14	7	2.9	.77	37.6	142.5
Zinc	2103	.011	7	17	7+1-2	14	7	2.9	.77	37.6	142.5
Gallium	3011	.015	4	12	-	3	1.5	6.5	1.71	23.2	87.9
Gold	3020	.006	4	20	See 6.3.27	3	1.5	6.3	1.67	9.5	35.9
Gold	3021	.006	4	20	See 6.3.27	3	1.5	6.5	1.71	9.3	35.1
Gold	3022	.006	4	15	See 6.3.28	3	1.5	7.0	1.86	8.5	32.3
Gold	3023	.007	4	10	See 6.3.29	.5	.25	25.3	6.7	2.8	10.5
Indium	3030	.009	8	25	See 6.3.30	4	2	4.0	1.05	22.5	85.3
Palladium	3040	.017	4	14	4+1-1	6	3	13.1	3.47	12.9	49.0
Platinum	3052	.150	3	10	See 6.3.32	12	6	35.2	9.33	42.7	161.5
Rhenium	3060	.750				6	3	52.2	13.84	152.5	577.3
Rhodium	3072	.030	4	15	4+1-1	6	3	6.8	1.80	44.1	167.1
Rhodium	3074	.030	3	12	3+1-1	4	2	20.4	5.4	14.7	55.7
Silver	3080	.005	8	20	8+1-1	8	4	2.7	.72	18.4	69.7
Silver	3081	.005	4	10	4+.5-1	2	1	3.4	.91	14.5	55.0
Silver	3082	.005	5	16	5+1-1	5	2.5	3.4	.91	14.5	55.0
Silver	3083	.005	5	16	5+1-1	5	2.5	3.4	.91	14.5	55.0
Nickel-Cobalt	4002	.030	8	25	See 6.3.39	12	6	10.4	2.75	28.9	109.2
Tin-Indium	4003	.008	8	25	-	4	2	3.3	.86	24.5	92.9
Tin-Lead-Nickel	4005	.006	5	15	5+1-1	3	1.5	3.8	1.01	15.6	59.2
Cobalt-Tungsten	4007	.020	10	25	See 6.3.41	12	6	14.5	3.83	13.7	52.2
Nickel-Tungsten	4008	.025	8	25	See 6.3.42	12	6	11.9	3.13	21.1	79.8
Babbitt SAE 11	4009	.006	3	15	See 6.3.43	1	1	4.5	1.19	13.2	50.0
Babbitt-Soft	4010	.006	3	15	See 6.3.43	1	1	4.5	1.19	13.2	50.0
Babbitt-Navy #2	4011	.006	3	15	See 6.3.43	1	1	4.5	1.19	13.2	50.0

Table 14-8.—Solutions Used for Salvage

Solution	Code	BHN	Ductility	Normal Maximum Build Up One Layer-Inches	Ease in Plating	Maximum Plating Speed In./Hr	Solution Cost $ Per In.^{3}
Chromium	2031	709	Very Poor	.0005	Very Difficult	.003	874.00
Nickel	2085	554	Very Poor	.015	Easy	.093	160.00
Chromium	2030	553	Not Coherent	.002	Difficult	.009	2,974.00
Cobalt-Tungsten	4007	512	Very Poor	.005	Difficult	.060	640.00
Nickel-Tungsten	4008	503	Very Poor	.005	Difficult	.048	487.00
Nickel	2080	430	Very Poor	.007	Average	.057	338.00
Cobalt	2043	418	Fair	.008	Easy	.070	242.00
Nickel	2088	325	Fair	.007	Average	.057	-
Nickel	2086	264	Excellent	.007	Average	.040	445.00
Copper	2055	211	Fair	.012	Easy	.192	116.00
Copper	2052	198	Poor	.004	Easy	.054	102.00
Copper	2050	134	Excellent	.015	Very Easy	.046	94.00
Silver	3083	115	Poor	.010	Very Easy	.100	391.00
Zinc	2102	51	Excellent	.006	Very Easy	.127	75.00

NOTE: Code 2085, 4007, and 4008 deposits should be ground if machining is required after plating. Code 2080, 2043, 2088, and 2086 deposits should be ground, but can be machined but with difficulty and high tool wear. Code 2055, 2052, 2050, 2102 and 3083 deposits are easily machined.

page 14-66. Multiply by some factor if some solution will be inadvertently lost. Factors vary from 1.5 to as much as 4. Factors will be found on solution bottles. (See figure 14-18.)

Step #10 Determine preparatory and preplate solutions required using table 14-9. Determine type of tools to be used with these solutions using figures 14-13 and 14-14.

Step #11 Determine covers to be used on all preparatory and plating tools.

Step #12 Determine masking required.

Two examples of the planning procedure used on actual jobs follow below. This information is furnished from the *Dalic Selective Plating Instruction Manual* for you to use as a guideline. Consult your particular instruction manual before any plating operation.

28.456

Figure 14-18.—Plating solutions.

EXAMPLE #1:

Step #1 Information on the job.

a. No. of parts–1

b. Base Material–Steel

c. Area to be plated–1" long x 3.500 + 0.000 bore in turbine wheel.

d. Purpose of deposit–Repair worn I.D. Color match important. Good hardness, adhesion and cohesion required.

e. General idea of what is adjacent–Part overall has a 3-ft O.D. and a maximum thickness of about 1 inch. Numerous turbine blades are at the O.D.

f. Thickness of deposit required–Diameter after truing up I.D. by grinding is 3.5015. A plating thickness of 0.001 inch will bring bore to middle of desired tolerance.

Step #2 Select plating solution to be used. Cobalt 2043 meets all requirements.

Table 14-9.—Solution and Deposit Properties

Solution	Code	Applications
Chromium	2031	Used occasionally as an overlay a few ten-thousandths inches thick on nickel or cobalt where a little more wear resistance is desired, such as on hydraulic piston rods. Never used alone for salvage.
Nickel	2085	Used extensively for salvage and repair of aluminum, cast iron, and steel parts. Works well under roller bearings, riding against babbitt bearings, etc. Not used in cases where there is extreme shock such as on cutting ends of punches, etc.
Chromium	2030	Very seldom used for salvage.
Cobalt-Tungsten	4007	Used occasionally for high wear applications, particularly at high temperature, i.e., up to approximately 1000°F. Maximum thickness approximately .005 inches.
Nickel	2080	Used often where a good combination of wear resistance, corrosion resistance, and toughness is desired. Used primarily on steel, stainless steel, nickel, etc.
Cobalt	2043	Used often where a good combination of wear resistance, and toughness is desired. Used primarily on steel, stainless steel, nickel, etc. Excellent color match with steel and stainless steel.
Nickel	2088 & 2086	Used often where maximum ductility and corrosion protection are desired along with some hardness.
Copper	2055	Used occasionally for high-buildups on smaller areas where maximum plating speed is important. Adhesion and coherence not quite as good as Code 2050.
Copper	2052	Used occasionally for buildups up to .004 inches on aluminum, steel, cast iron, and zinc, particularly where it is difficult to mask and prevent attack by other solutions.
Copper	2050	Used extensively on steel, copper, cast iron, nickel, and stainless steel particularly in high buildups. Often overlaid with nickel or cobalt for extra wear or corrosion resistance.
Silver	3083	Used occasionally on worn surfaces where the plating must be hand-worked to meet final dimensional requirements. Is hard enough for most applications, but is soft enough to be easily scraped or sanded.
Zinc	2102	Used extensively on aluminum and zinc particularly in high buildups.

Step #3 Amp-hr required.

A = 3.14 DL = 3.14 x 3.50 x 1.00 = 11.0
Amp hr = F x A x T = 0.020 x 11 x 10 = 2.2

Step #4 General approach.

The small area, amp-hr, and thickness involved suggest that (1) a special tool is not required and (2) that solution need not be pumped. This will be justified in the following steps. The part will be cleaned, etched, rinsed, etc. over a drain and then, being light enough, will be placed over a 14" x 17" collecting pan. A hole in the collecting pan will direct the solution back to the sclution container. The solution container is large enough to hold all the solution, but small enough to have enough depth of solution to thoroughly wet all of the plating tool.

Step #5 Plating tool to be used.

A RF-30 tool with a 1/4" thick cover will just match the internal diameter.

Step #6 Plating tool contact area.

Although tool with cover just matches the I.D., pressure on the tool cover will compact it and lead to perhaps 50% contact area, i.e., 5.5 square inches.

Step #7 Plating amperage.

Plating Amps = CA x ACD = 5.5 x 7 = 38.5

Step #8 Plating time.

$$PT(hr) = \frac{\text{Amp-hr}}{\text{Plating Amps}} = \frac{2.2}{38.5} = 0.057 \text{ hr}$$

Double the plating time because solution will be dipped for. The total plating time, therefore, is 0.114 hr or 6.84 minutes. The plating time is so short that turning the part in a lathe or pumping the solution is not necessary and the tool will be moved by hand.

Step #9 Plating solution required.

Liters = Q(L) x T(I) x A = 5.4 x 0.0010 x 11 = 0.059

This obviously is not enough to thoroughly wet the cover. It is estimated that 1 liter will be sufficient for the purpose.

Step #10 Preparatory and preplate solutions and tools.

a. Code 1010, 1022, 1023, and 2080.

b. Tools: AC-5. These, although relatively small, give a 1/2" x 1" contact area and should be satisfactory.

c. Quantity of solution required: Approximately 0.1 liter for each. This amount, when a small beaker is used, should thoroughly wet the cover.

Step #11 Covers to be used.

Preparatory tools: Cotton batting and cotton tubegauze.
Plating tool: Cotton batting and cotton tubegauze, since the cover is pure and inexpensive. Although cotton tubegauze is not wear resistant, it should easily hold up in the 15-minute plating time.

Step #12 Masking.

Aluminum tape and contact paper to prevent the part from contaminating the solution.

EXAMPLE #2:

Step #1 Information on job.

a. Number of parts–1
b. Base material–Steel with loose metal spray from previous repair.
c. Area to be plated–7" long area on 2.436 + 0.001 O.D. - 0.000
d. Purpose and requirements of deposit–Repair loose fit on inner race of roller bearing.
e. Although the part is a large recirculating fan about 5 feet long and with a maximum O.D. of 3 feet, the area being plated is a simple O.D. on a shaft.
f. Thickness required–It was decided to machine off the metal spray coating which was obviously very loose, leaving a gentle taper at edges. After machining, the diameter was 2.285". Thickness required therefore, is 0.152" in diameter or 0.076" on radius. Since plating will have to be stopped one or two times for machining to remove buildup at edges and improve the surface, a total of approximately 0.100 inch plating should be planned on.

Step #2 Select plating solution to be used.

Copper 2050 will be used because of the high thickness required. Copper 2050 stays smooth to high thicknesses and is easy to reactivate for more plating. Machining will be required because of the high thickness of deposit to be applied. The deposit, therefore, after copper plating will be machined 0.0005 inch undersize on the diameter and then be plated with 0.0005 inch of nickel 2085 for color match.

Step #3 Amp-hr required.

A = 3.14 DL = 3.14 x 2.436 x 7 = 53.5
Amp-hr (Cu) = F x A x T = 0.013 x 53.5 x 1000 = 696
Amp-hr (Ni) = F x A x T = 0.015 x 53.5 x 5 = 4.01

Step #4 General approach.

The part will be rotated in a lathe because a lathe is available. Solution will also be pumped through a special tool.

Step #5 Plating tool to be used.

A special tool will be prepared for copper plating since no standard tool is available to cover the full 7" length. The largest power pack available is a 60-35. Planning on drawing 55 amperes, the Optimum Contact Area was determined:

$$OCA = \frac{MA}{ACD} = \frac{55}{3} = 18.3$$

Since the length of the O.D. is 7", the contact length around the circumference should be $\frac{18.3}{7}$ or approximately 2.6 inches. A special anode, therefore, will be prepared about 7 1/2" long x 2 3/8" wide x 1 7/8" high. It will have a 1 1/2" radius (1/4" allowance for tool cover) placed in the 2 3/8" x 7 1/2" face. Solution will be fed through an F-handle to a 1/2" hole in the anode, running in the 7" direction (capped off at ends) and then through six 1/8" holes distributed along the 7" direction to the face having the radius.

Step #6 Plating tool contact area.

Copper–Not required (determined in Step #5).
Nickel–If an F-3 plating tool is used for nickel plating, the contact area will be 3 1/2" x 1" along the circumference with a soft pad. CA = 3.5 x 1 = 3.5 sq in.

Step #7 Plating current.

Copper–Not required (determined in Step #5).

Nickel–Plating amperage

Plating Amps = CA x ACD = 3.5 x 7 = 24.5

Step #8 Plating time.

$$\text{Copper PT (hr)} = \frac{\text{Amp-hr}}{\text{Plating Amps}} = \frac{696}{55} = 12.7$$

$$\text{Nickel PT (hr)} = \frac{\text{Amp-hr}}{\text{Plating Amps}} = \frac{4.01}{24.5} = 0.164$$

If solution is dipped for total nickel plating, time would double, i.e., 0.328 hour. The use of an F-3 tool, therefore, is justified and the solution need not be pumped through the anode.

Step #9 Plating solution required.

Copper 2050 (gal) = Q(G) x T(I) x A = 1.29 x 0.100 x 53.5 = 6.90

Since almost all solution can be caught for reuse, 7 gallons of copper 2050 should be sufficient.

Nickel 2085 (gal) = Q(G) x T(I) x A = 1.54 x .0005 x 53.5 = 0.041
Nickel 2085 (liter) = Q(L) x T(I) x A = 5.8 x .0005 x 53.5 = 0.155

Since an F-3 tool will be used to apply nickel, 0.155 liter would not be sufficient to wet the tool and the area to be plated. Approximately 1/2 liter is required.

Step #10 Preparatory and preplate solutions and tools.

a. To activate base material–1010, 1022, 1023, and 2080. To activate copper for more copper and the final nickel coating 1010,1023.

b. Tools required.

4 (F-2 or F-3)

c. Amount of solution required.

1010–1 liter (will be used several times)

1022–1/2 liter (will be used once)

1023–1 (will be used several times)

2080–1/2 liter (will be used once)

Step #11 Covers to be used.

Preparatory tools–Cotton batting and cotton tubegauze.
Copper plating tool–White Scotchbrite

Step #12 Masking.

Aluminum tape 2" and vinyl tape 2".

FINAL PREPARATION

Longer range planning should have assured that appropriate equipment, materials, and supplies are available to carry out the job. This section deals with the final preparations you should make just prior to plating.

FAMILIARIZATION WITH EQUIPMENT AND PROCEDURES

Success in carrying out plating operations is assured by quickly and knowledgeably carrying out the various steps. As the operator you should be familiar with the following:

1. The power pack and the position and purpose of the various controls and meters.

2. How the base material should look at various stages of preparation.

3. What a good and bad deposit look like as the plating is being applied.

Some practice is recommended when the equipment is new, a new base material is encountered, or a new plating solution is to be used. In practicing on a new base material, shorter and longer operations should be tried until the operator is certain when he is obtaining the desired appearance. In practicing with a new solution, very high and very low voltages should be used until the operator is certain that he knows what a good deposit and bad deposit (burned or otherwise) looks like. If possible, the operator should run a plating test on a 1" X 1" area using an AC-5 or similar size tool; the operator should be able to plate a good deposit at the volts and amps given in table 14-3 and the "Plating Example."

DRAFT A FLOW CHART

A very valuable tool for any operation is a good plan. Figure 14-19 shows a recommended plan or flow chart which will help you conduct the operation smoothly, and remind you of all the important elements of the operation.

PREPARE PART FOR PLATING

1. Inspect the area to be plated for any signs of a foreign surface being present such as an electr⁻ ⁿate, paint, scale, anodized coating, etc. Remove by suitable means such as vapor or dry blast, sandpaper, wire brush, etc. In pit-filling applications pay particular attention to ensure that the bottom of the pit is clean.

2. Preclean, if necessary, the area to be plated and the surrounding areas with a quick-drying solvent that leaves no residue (such as trichlorethylene, perchlorethylene, etc.). This should assure that masking materials will still stick and that solutions and tools will not come in contact with dirty, oily surfaces. The area to be plated should look clean.

3. Mask off the area to be plated.

4. If the part is to be rotated in a lathe or turning head, set the rpm to obtain optimum anode-to-cathode speed as given in table 14-10. If plating tool will be moved by hand, visualize the proper tool movement speed.

5. When the solution plates better at temperatures higher than room temperature, preheat the part and the solution, as required, by a suitable means. Methods used to preheat solutions include:

   a. Placing tightly capped bottles in a basin or tank of hot water.
   b. Pouring solutions into pyrex or stainless steel containers and heating over a range.
   c. Putting immersion heaters into the solution.

SETTING UP THE EQUIPMENT

Set up the power pack near the work so it is easily accessible and you can view the instruments. Connect appropriate size output leads to the power pack and connect the alligator clamp lead to the part or the lathe.

Wrap the tools, making sure the covers do not get dirty.

Pour out sufficient solution in clean containers. Set up the solution pump and test operate it. Soak the covered tools as long as possible in their respective solutions (at least five minutes).

Arrange so everything you will use is handy.

GENERAL SETUP

As the operator, you should be as comfortable as possible, particularly on lengthy plating jobs. You can then concentrate your full attention on the job, you will not be diverted by

Base Material - Cast Iron	Area Plated - 3" I.D. x 2" Long	Deposit Applied - Nickel Code 2085
Thickness Required - .003"	Tool to Part Movement - Lathe 64 r.p.m.	Plating Tool - Special, Covers 1/2 of Area
Solution Supply - Pump	Amp.-Hrs. - 8.5	Expected Amps - 66

Step	Operation	Material	Volts	Polarity	Visual Test - What you are looking for
1	Preclean	As required	--	--	Clean until surface free of visible films of oil, grease, dirt and oxide films.
2	Electroclean	Cleaning & Deoxidizing Code 1010	18	Forward	No water breaks after following rinse.
3	Rinse	Clean tap water	--	--	Thorough rinse of entire area. No water breaks.
4	Etch	No. 2 Etch Code 1022	12	Reverse	Uniform etch of entire area. Dark gray color. Cast iron grain structure visible.
5	Rinse	Clean tap water	--	--	Thorough rinse of entire area. No water breaks.
6	Desmut	No. 3 Etch Code 1023	20	Reverse	Uniform lightening to a light gray color. Will not become any lighter. Some cast irons change their grain structure.
7	Rinse	Clean tap water.	--	--	Thorough rinse of entire area. No water breaks.
8	Nickel Preplate	Nickel Code 2080	8 then +1 to approx. 13	Forward	Change of surface to a more nickel color. No darker burned areas.
9	Rinse	Clean tap water	--	--	Thorough rinse of entire area. No water breaks.
10	Prewet	Nickel Code 2085	--	--	Replacement of water on entire area with Nickel Code 2085.
11	Nickel Plate	Nickel Code 2085	8 then +1 to approx. 13	Forward	Medium gray, matte surface.
12	Rinse	Clean tap water	--	--	Thorough rinse of entire area. No water breaks.
13	Neutralize	Cleaning & Deoxidizing Code 1010	--	--	Replacement of water on entire area with Code 1010.
14	Dry	As required	--	--	Complete drying of surface.

Figure 14-19.—Sample flow chart for setting up plating.

Table 14-10.—Plating Solution Data for Operators

Plating Solution	Code	Factor Amp-Hrs. for .0001" on 1 in.^{2}	Plating Example Using AC-5 Tool Volts	Amps	Maximum Recommended Use Amp-Hrs. Per Liter	Gal.	Anode to Cathode Speed ft./min.	Temperature °F Optimum	O.K. at Room Temp.
Antimony	2000	.008	8	1.3	29.5	111.6	50	60-120	Yes
Bismuth	2010	.008	4	1	17.4	66.0	50	60-120	Yes
Cadmium	2020	.007	8	5	31.6	119.6	75	60-120	Yes
Cadmium	2021	.007	8	1	17.3	65.4	50	60-120	Yes
Cadmium	2022	.007	16	2	27.1	102.8	50	60-120	Yes
Cadmium	2023	.007	8	1.2	24.7	93.4	50	60-120	Yes
Chromium	2030	.137	12	6	24.4	92.4	20	60-120	Yes
Chromium	2031	.120	6	4	76.4	289.0	50	105	Yes
Cobalt	2043	.020	13	5	36.8	139.2	25	60-150	Yes
Copper	2050	.013	4.5	2.5	26.6	100.5	50	60-120	Yes
Copper	2051	.013	10	3	35.4	134.0	50	60-120	Yes
Copper	2052	.013	8	3	35.4	134.0	50	60-120	Yes
Copper	2054	.013	8	3	26.6	100.5	50	60-120	Yes
Copper	2055	.013	10	10	32.1	121.4	50	60-120	Yes
Iron	2061	.018	14	4	8.7	33.	50	60-150	Yes
Lead	2070	.006	10	1.5	16.1	61.1	50	60-120	Yes
Lead	2071	.006	12	1.2	16.1	61.1	50	60-120	Yes
Nickel	2080	.021	14	4	26.4	99.9	50	110-170	No
Nickel	2085	.015	8	3	25.7	97.3	75	60-150	Yes
Nickel	2086	.025	14	4	25.7	97.3	50	110-170	No
Nickel	2088	.021	14	4	23.7	89.9	50	110-170	No
Tin	2090	.007	8	1.2	23.4	88.6	50	60-120	Yes
Tin	2092	.007	8	1.0	23.4	88.6	50	60-120	Yes
Zinc	2100	.011	8	2	47.1	178.1	50	60-120	Yes
Zinc	2101	.011	13	4	35.3	133.6	50	60-120	Yes
Zinc	2102	.011	13	4	37.6	142.5	50	60-120	Yes
Zinc	2103	.011	9	2.5	37.6	142.5	50	60-120	Yes
Gallium	3011	.015	8	1.5	23.2	87.9	50	72 max.	Yes
Gold	3020	.006	8	1.0	9.5	35.9	50	60-120	Yes
Gold	3021	.006	8	1.2	9.3	35.1	50	60-120	Yes
Gold	3022	.006	8	1.0	8.5	32.3	50	60-120	Yes
Gold	3023	.007	7	.25	2.8	10.5	50	60-120	Yes
Indium	3030	.009	10	3	22.5	85.3	50	60-120	Yes
Palladium	3040	.017	8	2.0	12.9	49.[illegible]	50	60-120	Yes
Platinum	3052	.150	5	2.5	42.7	161.5	50	60-120	Yes
Rhenium	3060	.750	12	2.5	152.5	577.3	50	60-120	Yes
Rhodium	3072	.030	10	3.5	44.1	167.1	50	60-120	Yes
Rhodium	3074	.030	7	2	14.7	55.7	50	60-120	Yes
Silver	3080	.005	13	2.5	18.4	69.7	50	60-120	Yes
Silver	3081	.005	7	.8	14.5	55.0	50	72 min.	Yes
Silver	3082	.005	13	2	14.5	55.0	50	60-120	Yes
Silver	3083	.005	12	2.5	14.5	55.0	50	60-120	Yes
Nickel-Cobalt	4002	.030	14	4	28.9	109.2	50	110-170	No
Tin-Indium	4003	.008	6	.75	24.5	92.9	50	60-120	Yes
Tin-Lead-Nickel	4005	.006	10	1.2	15.6	59.2	50	60-120	Yes
Cobalt-Tungsten	4007	.020	13	4	13.7	52.2	50	110-170	No
Nickel-Tungsten	4008	.025	16	4	21.1	79.8	50	110-170	No
Babbitt SAE 11	4009	.006	9	.5	13.2	50.0	50	60-120	Yes
Babbitt-Soft	4010	.006	9	.5	13.2	50.0	50	60-120	Yes
Babbitt-Navy #2	4011	.006	9	.5	13.2	50.0	50	60-120	Yes

unnecessary distractions, and your efficiency will not decrease from fatigue.

You should have adequate lighting so you can see that the preparation and plating is proceeding properly.

Refer to table 14-11 for special safety precautions such as necessity for ventilation, gloves, special clothing, etc.

Have sufficient clean tap water available for rinsing the part.

Review the setup procedure one last time to ensure that everything necessary is available and handy. This step is to avoid delays during plating and in turn will produce a finer finished product.

PREPARATION INSTRUCTION: GENERAL

Electroplates and tank electroplates depend on atomic attraction of the electroplate to the base material for adhesion. Extremely thin, invisible films of oil, grease, dirt, oxides, passive films, etc. are sufficient to prevent an atomic attraction and there is no adhesion of an electroplate. A preparation cycle is used, therefore, just prior to plating to remove step by step all the last traces of these obstacles to developing excellent adhesion.

A preparation cycle consists of a number of operations, each one performing a specific function. The number and types of operations and the solutions used depend on the base material, not on the plating solution to be used later. Each operation should be carried out properly to ensure maximum adhesion. The operations and cycle are properly carried out when:

1. The proper solutions are used in the proper sequence.

2. The solutions are used in the proper direction, i.e., forward or reverse.

3. The operations follow each other as rapidly as possible and with the surface not being allowed to dry between operations.

4. The desired results are obtained in each operation.

In most operations, you can tell by the appearance whether you have achieved the desired results. The visual tests are important and you should pay particular attention to those given in this chapter.

Each operation is usually carried out within a certain voltage range as is shown in the following pages on preparing specific base materials. When a small tool is used on a small area, a low voltage in the range is used. When a large tool is used on a large area, a high voltage in the range is used. The voltage used in a preparatory step, however, is not critical and can vary by several volts. Obtaining the desired results as determined by the visual test is again the important part of the operation.

The following sections discuss the various types of operations carried out on various base materials.

CLEANING AND DEOXIDIZING

A cleaning and deoxidizing operation is usually performed first on most base materials to remove the last traces of dirt, oil and grease. It also removes the light oxide films on some metals. Forward current (cathodic electrocleaning) is usually used. The cleaning and deoxidizing solution, however, must be used with reverse current (anodic electrocleaning) whenever hydrogen contamination and embrittlement of the base material must be avoided, such as, in the case of ultra high-strength steel. The cleaning and deoxidizing operation is performed at 8 to 20 volts, depending on the base material and the size of the tool. Higher voltages, longer cleaning times, and development of heat in the tool are helpful in cleaning stubborn areas. Areas surrounding the area to be plated should be cleaned, since oil

Table 14-11.—Safety Precautions for Operators

Plating Solution	Code	Ave. pH	Special Problems	Ventilation Required	Requirements For Precautions Against Skin Contact
Antimony	2000	7.3	Poisonous Metal	Seldom	Very Strong
Bismuth	2010	10.8	Poisonous Metal	Seldom	Moderate
Cadmium	2020	0.5	Poisonous Metal- Corrosive Solution	Usually	Very Strong
Cadmium	2021	9.0	Poisonous Metal	Seldom	Very Strong
Cadmium	2022	8.8	Poisonous Metal	Seldom	Very Strong
Cadmium	2023	11.0	Poisonous Metal	Seldom	Very Strong
Chromium	2030	6.3	Poisonous Metal	Frequently	Very Strong
Chromium	2031	0.5	Poisonous Metal- Corrosive Solution	Usually	Very Strong
Cobalt	2043	1.5	----	Seldom	Moderate
Copper	2050	0.5	Corrosive Solution	Seldom	Very Strong
Copper	2051	11.1	----	Seldom	Moderate
Copper	2052	6.4	----	Seldom	Moderate
Copper	2054	1.7	Corrosive Solution	Seldom	Very Strong
Copper	2055	1.0	Corrosive Solution	Frequently	Very Strong
Iron	2061	2.8	----	Seldom	Moderate
Lead	2070	8.0	Poisonous Metal	Seldom	Very Strong
Lead	2071	8.0	Poisonous Metal	Seldom	Very Strong
Nickel	2080	2.4	----	Frequently	Moderate
Nickel	2085	7.3	----	Seldom	Moderate
Nickel	2086	3.0	----	Seldom	Moderate
Nickel	2088	3.0	----	Frequently	Moderate
Tin	2090	7.2	----	Seldom	Moderate
Tin	2092	7.3	----	Seldom	Moderate
Zinc	2100	7.7	Poisonous Metal	Seldom	Very Strong
Zinc	2101	5.8	Poisonous Metal	Seldom	Very Strong
Zinc	2102	4.9	Poisonous Metal	Seldom	Very Strong
Zinc	2103	2.7	Poisonous Metal	Seldom	Very Strong
Gallium	3011	11.0	----	Seldom	Very Strong
Gold	3020	9.9	Contains Cyanide- Cyanide In Fumes	Usually	Very Strong
Gold	3021	7.5	Contains Cyanide- Cyanide In Fumes	Usually	Very Strong
Gold	3022	5.1	Contains Cyanide- Cyanide In Fumes	Usually	Very Strong
Gold	3023	9.7	Contains Cyanide- Cyanide In Fumes	Usually	Very Strong
Indium	3030	9.3	----	Seldom	Moderate
Palladium	3040	8.3	----	Seldom	Moderate
Platinum	3052	0.5	Corrosive Solution	Seldom	Very Strong
Rhenium	3060	1.0	----	Seldom	Very Strong
Rhodium	3072	.6	Corrosive Solution	Seldom	Very Strong
Rhodium	3074	1.1	Corrosive Solution	Seldom	Very Strong
Silver	3080	10.6	Contains Cyanide	Seldom	Very Strong
Silver	3081	10.3	Contains Cyanide	Seldom	Very Strong
Silver	3082	9.6	Contains Cyanide- Cyanide In Fumes	Usually	Very Strong
Silver	3083	11.6	Contains Cyanide	Frequently	Very Strong
Nickel-Cobalt	4002	2.5	----	Frequently	Moderate
Tin-Indium	4003	8.7	----	Seldom	Moderate
Tin-Lead-Nickel	4005	7.3	Poisonous Metal	Seldom	Very Strong
Cobalt-Tungsten	4007	2.0	----	Frequently	Moderate
Nickel-Tungsten	4008	2.5	----	Frequently	Moderate
Babbitt SAE 11	4009	7.5	Poisonous Metal	Seldom	Very Strong
Babbitt-Soft	4010	7.5	Poisonous Metal	Seldom	Very Strong
Babbitt-Navy #2	4011	7.5	Poisonous Metal	Seldom	Very Strong

and grease travel on the surface of water. A thorough water rinse should follow. If water "breaks" on the surface, the cleaning and deoxidizing time was too short and the operation should be repeated.

ETCHING

An etching operation using an etching solution and reverse current usually follows the cleaning and deoxidizing operation. The operation electrochemically removes oxides, corrosion products and smeared and contaminated surface material, all of which impair adhesion. When the unwanted surface material is removed, a uniform, dull, grainy appearance develops and the etching operation is discontinued. Normally, 0.000050 to 0.0002 inch of material is removed and this requires 0.006 to 0.026 amp-hr per square inch of area.

DESMUTTING

The etching operation on some materials results in the formation of a loose layer of insoluble material on the surface. An example of this is carbon films on the surface after etching carbon steels. These layers can interfere with the development of maximum adhesion. These films are removed by an appropriate desmutting operation. The operation is completed when the surface is uniform in appearance and will not become any lighter in color.

ACTIVATING

An activating operation is used on some base materials, such as chromium, nickel, stainless steel, etc. to remove a "passive" film which quickly forms on these materials. A cleaning and deoxidizing operation on these materials does not remove the passive film. An etching operation on these removes material from the surface, but simultaneously forms the passive film. Passive films prevent maximum adhesion. An activating operation is, therefore, conducted on these materials just prior to plating, using forward current and an appropriate solution.

Cleanliness is of extreme importance in the activating operation since it is the last operation before plating. Contamination of the solution from any source must be avoided since this step is in the forward direction and contaminants may be plated out as a nonadherent film.

With the exception of chromium, there are no visual keys as to whether the operation has been conducted properly. The passive film is invisible and on most materials such as nickel and stainless steel no change can be detected when it is removed. Any change on these materials may indicate contamination from the activating solution, anode, or plating tool. The operation, therefore, must be carried out on a time basis with about 3 seconds spent on each part of the total area. With an activating tool covering all the area to be plated spend, therefore, about 3 seconds in the operation. With a tool covering 1/5 of the area, conduct the operation for 15 seconds, spending an equal amount of time on all parts of the area.

PLATING

The final preparatory operation should be followed as quickly as possible with the subsequent plating operation whether it is a preplate or the final desired plating. This is of particular importance when an activating operation has been the last step. THE PART SHOULD NOT BE ALLOWED TO DRY BETWEEN THE ACTIVATING AND PLATING OPERATIONS.

VERIFYING IDENTITY OF BASE MATERIAL

Obtaining good adhesion of a deposit begins with proper identification of the surface being plated. Frequently, the operator is misinformed as to the identity of the base material and is not informed that a coating is present. This, of course, can lead to adhesion problems. An alert operator, by carefully watching the etching operation, however, can frequently detect incorrect identification or the presence of coatings. The following may be helpful in this matter. (Also refer to table 14-3.)

Material	Result of No. 2 or No. 4 Etch Reverse Operation — Appearance of Etched Surface	Color of Solution in Cover	Surface rusts after etching when kept wet	Magnetic
Low Carbon Steel	Light Gray	No Color	Yes	Yes
Medium or High Carbon Steel	Medium Gray to Black	Black smut in cover	Yes	Yes
300 Stainless	Light Gray	Yellow at first green later	No	No
400 Stainless Soft	Light Gray	Blue-green	No	Yes
400 Stainless Hard	Black	Blue-green with Black smut	No	Yes
Monel	Light Gray	Pale Orange	No	Yes
Chromium	Shiny White	Yellow	No	No

PREPLATE INSTRUCTIONS

It may be necessary, in some cases, to apply a preplate with an appropriate plating solution. The preplate is applied immediately after the preparation of the surface and is followed immediately by a water rinse and plating with the final desired solution. The preplate ensures maximum adhesion of the final deposit. The base material and the final desired plating solution determine whether a preplate is required and, if so, what preplate is required. Table 14-12 lists the preplates required for commonly used solutions on commonly plated base materials. A Code 2080 preplate and then a Code 2050 preplate, for example, are required on stainless steel before plating copper 2055. A preplate is not required for plating Code 2103 on low carbon steel.

The preplate thickness applied varies from 0.000010 inch on smooth surfaces to 0.000050 inch on rough surfaces. Normally, when a uniform color change is noted as a result of plating the preplate on the base material, a satisfactory thickness has been applied. Since new operators often do not apply a sufficient thickness of preplate, they should calculate and pass the ampere-hours necessary for a thickness of at least 0.000025 inch. Examples for solutions are from the Dalic Selective Plating Manual. Each manufacturer has its own instruction manual and solution guide.

The preplate voltages used are as follows:

Code	Very Small Tool	Very Large Tool
2080	8	12
2050	4	6
2051	5	8
2085	7	10
3023	5	8
3040	5	8

Table 14-12.—Preplates for Base Materials for Various Solutions

Plating Solution	Code	Aluminum and Aluminum Alloys	Copper and Copper Base Alloys	Iron, Steel and Cast Iron	Nickel and Nickel Base Alloys	Stainless Steel	Zinc and Zinc Base Alloys
Antimony	2000	2080			2080	2080	2051 or 2085
Bismuth	2010	2080			2080	2080	2051 or 2085
Cadmium	2020	2080			2080	2080	2051 or 2085
Cadmium	2021	2080			2080	2080	2051 or 2085
Cadmium	2022	2080		1032	2080	2080	2051 or 2085
Cadmium	2023	2080			2080	2080	2051 or 2085
Chromium	2030	2080	2080	2080	2080	2080	
Chromium	2031	2080	2080	2080	2080	2080	
Cobalt	2043	2080		2080	2080	2080	2051
Copper	2050	2080		2080		2080	2051
Copper	2051	2080		2080	2080	2080	
Copper	2052	2080		2080	2080	2080	2051
Copper	2054	2080 + 2050		2080 + 2050	2080 + 2050	2080 + 2050	2051
Copper	2055	2080 + 2050		2080 + 2050	2080 + 2050	2080 + 2050	2051
Iron	2061	2080		2080	2080	2080	2051
Lead	2070	2080		2080	2080	2080	2051 or 2085
Lead	2071	2080		2080	2080	2080	2051 or 2085
Nickel	2080						2051
Nickel	2085	2080		2080	2080	2080	
Nickel	2086						2051
Nickel	2088						2051
Tin	2090	2080		2080	2080	2080	2051 or 2085
Tin	2092	2080		2080	2080	2080	2051 or 2085
Zinc	2100	2080			2080	2080	
Zinc	2101	2080				2080	
Zinc	2102	2080				2080	
Zinc	2103	2080				2080	
Gallium	3011	2080		2080	2080	2080	2051 or 2085
Gold	3020	2080		2080	2080	2080	
Gold	3021	2080		2080	2080	2080	
Gold	3022	2080		2080	2080	2080	
Gold	3023	2080		2080	2080	2080	
Indium	3030	2080			2080	2080	2051 or 2085
Palladium	3040	2080		2080	2080	2080	2051 or 2085
Platinum	3052	2080		2080	2080	2080	2051 or 2085
Rhenium	3060	2080		2080	[illegible]	2080	2051 or 2085
Rhodium	3072	2080		2080		2080	2051
Rhodium	3074	2080		2080		2080	2051
Silver	3080	2080 + 3040*	3040	2080 + 3040*	2080 + 3040*	2080 + 3040*	2051 + 3040*
Silver	3081	2080 + 3040*	3040	2080 + 3040*	2080 + 3040*	2080 + 3040*	2051 + 3040*
Silver	3082	2080 + 3040*	3040	2080 + 3040*	2080 + 3040*	2080 + 3040*	2051 + 3040*
Silver	3083	2080 + 3040*	3040	2080 + 3040*	2080 + 3040*	2080 + 3040*	2051 + 3040*
Nickel-Cobalt	4002						2051
Tin-Indium	4003	2080		2080	2080	2080	2051 or 2085
Tin-Lead-Nickel	4005	2080		2080	2080	2080	2051
Cobalt-Tungsten	4007	2080	2080	2080	2080	2080	2051
Nickel-Tungsten	4008	2080	2080	2080	2080	2080	2051
Babbitt SAE 11	4009	2080	2080	2080	2080	2080	2051
Babbitt-Soft	4010	2080	2080	2080	2080	2080	2051
Babbitt-Navy #2	4011	2080	2080	2080	2080	2080	2051

* Gold Code 3023 may be used in place of Palladium Code 3040

SUMMARY OF ELECTROPLATING

The ideal plating operation is carried out when simultaneously (1) the quality of deposit is the best possible; (2) the deposit is applied in a minimum amount of time; and (3) the deposit has a uniform desired thickness.

A number of steps should have been taken in initial and final preparations to ensure that the best possible quality deposit will be applied. Some of them include use of clean anodes, uncontaminated solution, proper cover material, etc. The operator, at the time of plating, however, must still carry out the operation properly.

GUIDELINES FOR THE OPERATOR

The guidelines for the operator discussed in detail throughout this chapter, are reviewed briefly in the following sections.

- Keep the area being plated clean.
- Keep the surface wet with plating solution.
- Keep the number and length of plating interruptions to a minimum.
- Prevent the solution from depleting in the work area.
- Maintain proper anode-to-cathode speed.
- Plate at the proper current density.
- Plate at approximately the proper temperature when plating temperature is important.

The first three guidelines assure obtaining good adhesion of the deposit to itself, the last four assure obtaining best deposit quality.

Keep the Area Being Plated Clean

Contamination of the area by oil, grease, dirt, etc. can result in adhesion problems and possibly poor deposit quality. Careful final preparations prior to plating should prevent contamination of the area being plated. Watch, however, for a possibly overlooked source of contamination such as the tool or solution moving over a dirty surface; correct as soon as possible.

Keep the Surface Wet with Plating Solution

Drying of the solution on the area being plated is obviously a significant change in the composition of the solution. This can affect the adhesion of the following deposit. Proper setups will largely ensure that the surface will not dry during plating. However, you should watch for signs of "overheating" of the part and the solution. If this occurs, supply more solution. If dipping for solution, you should dip often enough every 5 seconds or as required.

Keep Interruptions to a Minimum

Some metals, primarily nickel, cobalt, and chromium are subject to passivation which is the formation, in a short period of time, of a thin invisible oxide film. Good bond cannot be obtained without activation. Passivation can be prevented by avoiding all unnecessary interruptions of the plating operation, minimizing the length of time of unavoidable interruptions, and ensuring that all areas being plated are covered periodically (at least every 10 seconds) during plating.

Prevent Solution from Depleting in Work Area

Depletion of solution in the work area (where cover meets the part) has various effects depending on the solution. With most plating solutions, there is a greater tendency to get shiny, low thickness deposits. Other indications are a drop-off in plating current and a change in color of the solution in the cover. To prevent this, provide a sufficient amount of fresh plating solution and then pump fast enough or dip often enough to get it in the work area.

Maintain Proper Anode-to-Cathode Speed

Ensure that the tool is always moving relative to the part (fig. 14-12). If you set a plating tool on the part then move it straight back and forth instead of in a rotary motion on flat parts and move the tool in the direction of a rotating part, in some spots momentarily you have no relative movement. Burning of the deposit can result from this.

Visual Control

While plating, you can see what the deposit looks like as it is going down. Deposit appearance gives valuable information on deposit quality and overall plating efficiency. If you know the significance of variations in the deposit and what causes them, you can, when necessary, make appropriate corrections such as by changing the voltage, the anode-to-cathode speed, or the rate of solution supply. You should be aware of what good and bad deposits look like, pay attention to appearance while plating, and be able to make the appropriate corrections when they must be made.

EVALUATING DEPOSITS

The qualities to look for in plating deposits in all applications are good adhesion, proper thickness of coating, and high density of deposit. In corrosion protection applications where nonsacrificial coatings are used, also look for freedom from pores and surface-to-base metal cracks.

EVALUATING ADHESION

Some of the tests that can be used in sampling or prequalification testing, all of which are excellent, are (1) the chisel, knife, and scratch test or the grind and saw test.

Chisel, Knife, and Saw Test

If the deposit is sufficiently thick to permit the use of a chisel, test the adhesion by forcing the chisel between the coating and basic metal. Use a hammer to apply the force. Test thinner coatings by substituting a knife or scalpel for the chisel and lightly tap it with a hammer. Test very thin coatings by scratching through the coating to the basic metal. After these tests, closely examine the test area for lifting or peeling of the deposit at the base material to plating interface.

Grind and Saw Tests

Another good test for adhesion is to grind an edge of the plated specimen with a grinding wheel with the direction of cutting from the basic metal to the deposit. If adhesion is poor, the deposit is torn from the base. You can use a hacksaw instead of the grinder. It is important to saw in such a direction that a force is applied that tends to separate the coating from the basic metal. Grinding and sawing tests are especially effective on hard or brittle deposits.

TROUBLESHOOTING

POOR ADHESION

Carefully inspect the plated area to determine if the deposit is coming off the base material, the final deposit is coming off the preplate, or the final deposit is coming off itself. Examine the back side of the material coming off. Etch tests using Code 1022 or 1024 solution on the area where the material came off and a comparison of appearance with etched deposit and etched base material is often helpful.

In some cases, such as when plating on metal spray, tungsten carbide, electroless nickel, etc., the separation is in the base material, and, therefore, the DALIC plating cannot be faulted.

1. COMMON CAUSES FOR DEPOSIT COMING OFF BASE MATERIAL

   a. Base material not correctly identified.

      1. Determine for certain identity of base material.
      2. Check to determine if surface was etched as it should have been.

   b. Surface has a foreign coating such as metal spray, chrome plate, etc.

      1. Determine if plated area and other areas on workpiece etch the same. Use Code 1022 or 1024 and reverse current.

   c. Did not thoroughly, properly, and quickly carry out preparatory procedure.

   d. Contaminated preparatory solutions.

   e. Contaminated preparatory or preplate tools.

   f. Preparatory tools too small.

   g. Did or did not pre-wet surface prior to plating.

   h. Selected wrong plating solution.

   i. Did not use recommended preplate.

2. FINAL DEPOSIT COMING OFF PREPLATE

   a. Did not follow quickly with final plating solution.

   b. Did or did not pre-wet surface per instructions.

   c. Selected wrong plating solution.

3. FINAL DEPOSIT COMING OFF ITSELF

   a. Burned deposit.

   b. Plating operation interrupted too long.

   c. Solution contaminated.

   d. Contaminated anode.

   e. Wrong anode cover used.

POOR DEPOSIT QUALITY

Poor deposit quality includes roughness, stress-cracking, and stress-crack lifting. Common causes are as follows:

1. Selected wrong solution.
2. Plating solution contaminated.
3. Plating tool contaminated.
4. Used wrong cover or cover too thick or too thin.
5. Plating wrong.

LOW THICKNESS DEPOSIT

Low thickness deposit, i.e., did not achieve the desired thickness. Common causes are as follows:

1. Did not properly calculate area.
2. Did not properly calculate amp-hr.
3. Considerable plating went on aluminum tape or adjacent areas.
4. Overetched base material.
5. Plated wrong with "variable factor" solution.
6. Certain solutions overused.
7. Insufficient solution supply with certain solutions.
8. Plated in cover. Wash and examine cover to see if this actually occurred.
9. Nonuniform distribution of plating and measuring in low areas.

NONUNIFORM THICKNESS OF DEPOSIT

1. Tool not right.
2. Tool not used right.
3. Solution not distributed uniformly in cover.
4. Tool cover thickness varied.

TOOK TOO LONG TO FINISH JOB

1. Used wrong solution.
2. Plating tool too small.
3. Power pack too small.
4. Not plating as fast as possible with existing tool or solution.
5. Did not preheat properly certain variable factor solutions.

MACHINING AND GRINDING

Machining

The deposits softer than 250 Brinell are easily machined and specific recommendations are not necessary for these.

The cobalt, iron, and nickel deposits or their alloys are difficult to machine. If possible, grind rather than machine. When it is absolutely necessary to machine, good equipment and technique is required. Recommendations include:

1. Use new tight machine tools.
2. Use sharp carbide bits.
3. Use plenty of coolant.
4. Take light cuts of approximately 0.005 inch.
5. Use low cutting speeds, i.e., approximately 50 ft/min.

Grinding Nickel and Cobalt Deposits

The Norton Company, Worcester, Massachusetts, makes the following recommendations relative to grinding nickel or cobalt deposits:

1. Wet grinding recommended. Use plenty of coolant.

2. Wheel–Silicon Carbide Vitrified.

37060–KVK Recommended.

3. Wheel and Work Speeds–Wheel 6000 surface feet per minute.

4. Depth of Cut–0.0002 inch maximum to ensure against overheating of the deposit and deposit to base metal interface.

FORMULAS

There are a number of formulas that prove very useful with the DALIC Process. They, when used, assure fast, efficient, and trouble free DALIC plating operations.

Formula 1: Formula to control thickness of metal deposited

Amp-Hr = F X A X T

This formula should be used to determine the ampere-hours that should be passed while plating to obtain the desired thickness of deposit on the area to be plated.

In this formula, F is the factor obtained from plating solution bottle or table 14-5.

A = area of surface to be plated in square inches.

T = thickness of deposit desired in one ten-thousandths inches.

Deposit Thickness Desired–Inches	T equals
0.010	100.0
0.002	20.0
0.001	10.0
0.0005	5.0
0.000060	0.6
0.000008	0.08

Note: The proper value for T to put in the above formula may, also, be obtained by writing the thickness desired in inches and then moving the decimal point four (4) places to the right.

Example:

0.001 inch thickness desired
0.001 is the same as 0.0010
move the decimal point four places to the right,
i.e., 0010.
T, therefore, is 10

Formula 2: Formula to determine current density

$$CD = \frac{PA}{CA}$$

CD = current density in amps per square inch
PA = plating amperage
CA = contact area being made by the plating tool on the part in square inches

This formula allows one to compute the current density at which you are plating in a given operation. A comparison can then be made with values given in table 14-3 to determine if one is plating at a low current density, a normal current density, or an excessive current density etc. The operator with this information can then make appropriate adjustments while plating.

Formula 3: Formula to determine optimum plating tool contact area when designing special tools

$$OCA = \frac{MA}{ACD}$$

MA = maximum aperage output of power pack to be used
ACD = average current density for solution to be used

This formula should be used when designing special tools to develop the right size tool, neither too large or too small.

Formula 4: Formula to estimate plating amperage to be drawn with a given solution and plating tool

PA = CA X ACD
ACD = average current density for solution

This formula is used for two purposes, i.e.,

1. In conjunction with Formula 5 to estimate plating time.
2. By itself to determine if one is plating at the right amperage.

Formula 5: Formula to estimate plating time

$$PT\ (Hrs) = \frac{Amp\text{-}Hrs}{PA}$$

PA = value from Formula 4 for purpose 1 or average current while plating for purpose 2

This formula is used for two purposes:

1. To estimate plating time in setting up a job
2. To control thickness when no ampere-hour meter is available

Formula 6: Determining amount of solution required

Liters = Q(L) X T(I) X A

This formula is used (1) in estimating jobs and (2) to ensure that appropriate amounts of solution are available and used in a given job.

Q(L) is Solution Required Average per Inch^3 in liters from table 14-10.

T(I) = thickness of deposit desired in inches

A = area of surface to be plated

Formula 7: Formula to check ampere-hour meter accuracy

Amp-Hrs = Amps X Hrs

This formula is used periodically as a maintenance procedure to ensure that the amp-hr meter is accurate, or in cases where its accuracy is suspect.

The test is run by shorting the d.c. output leads and running the power pack for a set time (hr) at a set amperage. Assume a test is run for 3 minutes at 20 amps.

Amps = 20

$$\text{Hrs} = \frac{3}{60} \text{ or } 0.05$$

Placing these values in the above formula

Amp-Hr = 20 X 0.05

Amp-Hr = 1.00

The computed value (1.00) should be close (within a small percentage) to that passed on the amp-hr meter when shorting the d.c. output leads and running for 3 minutes at 20 amps.

CHAPTER 15

THE REPAIR DEPARTMENT AND REPAIR WORK

As a Machinery Repairman you may be assigned to almost any type ship. Aboard many ships, you will be a member of the engineering department; most Machinery Repairmen, however, are assigned to repair and tender type ships. On ships of this type, you will be part of the repair department and should know something about its functions, personnel, and shops. In this chapter, you will also find some examples of the type of repair work you are likely to encounter.

In effect, repair ships and tenders are floating bases, capable of performing a variety of maintenance and repair services that are beyond the capabilities of ships they serve. They are rather like small-scale Navy yards, with the same primary mission: to provide repair facilities and services to the forces afloat.

The most common type of repair ship, designated AR, provides general and special repairs to all types of ships. Special types of repair ships have been developed for special uses; for example, the ARG is designed for the repair of internal-combustion engines.

Each type of tender provides services for one type of ship, as indicated by the designation of the tender. The two best known types of tenders are the destroyer tender (AD) and the submarine tender (AS). Both conventional submarines and fleet ballistic missile submarines are tended by the AS; however, the organization of the repair department of an AS that tends fleet ballistic missile submarines differs somewhat from that of an AS that tends conventional submarines.

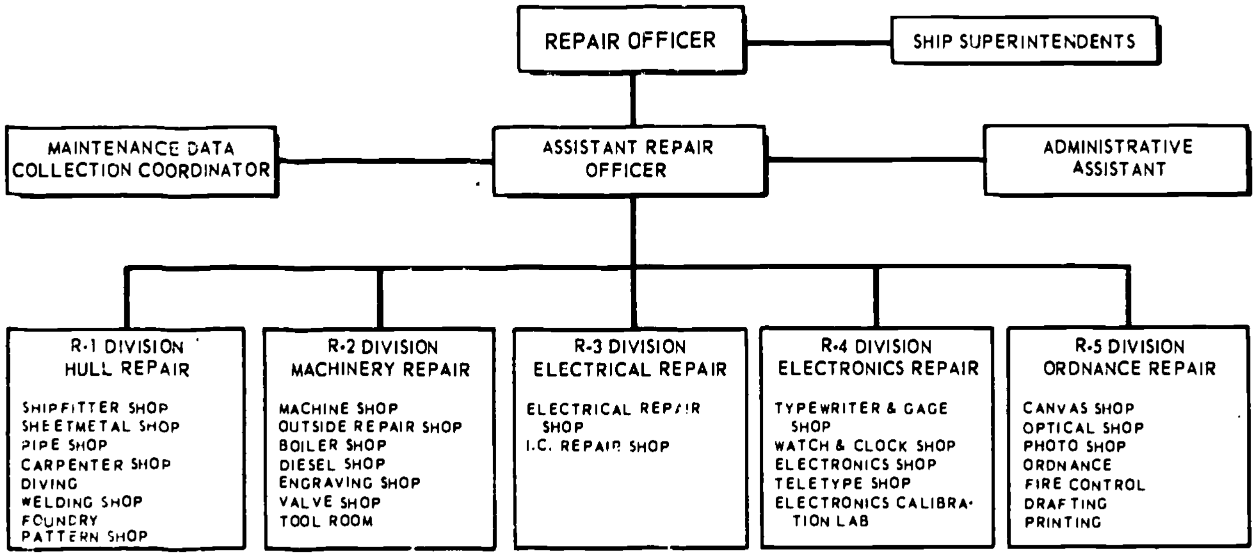
Since repairs and services to other ships are the primary functions of all repair ships and tenders, the repair department on a repair ship or tender makes a direct and vital contribution to fleet support. The operating forces of the fleet depend upon the services provided by all personnel of the repair department.

REPAIR DEPARTMENT ORGANIZATION AND PERSONNEL

The type of repair ship to which you will probably be assigned will be a destroyer tender (AD), a repair ship (AR), an internal-combustion engine repair ship (ARG), or a submarine tender (AS). When you are assigned to shore duty, you will almost certainly be assigned to a billet in the Repair Department of a shore installation. Since the shore-based installation has the same essential mission as the repair ship, the organization will be similar.

When you report aboard ship, you will need to learn the lines of authority and responsibility in the repair department. You will need to find out where your orders and assignments originate, exactly what is expected of you, and where to go for information, assistance, and advice. You can start acquiring this knowledge by studying the following material on repair department organization and personnel.

Repair department organization varies somewhat from one ship to another, as you can see by comparing figures 15-1 and 15-2. Figure 15-1 shows the organization of the repair department in a typical repair ship (AR); figure 15-2 shows the organization of the repair department in a fleet ballistic missile (FBM) submarine tender (AS).



63.10

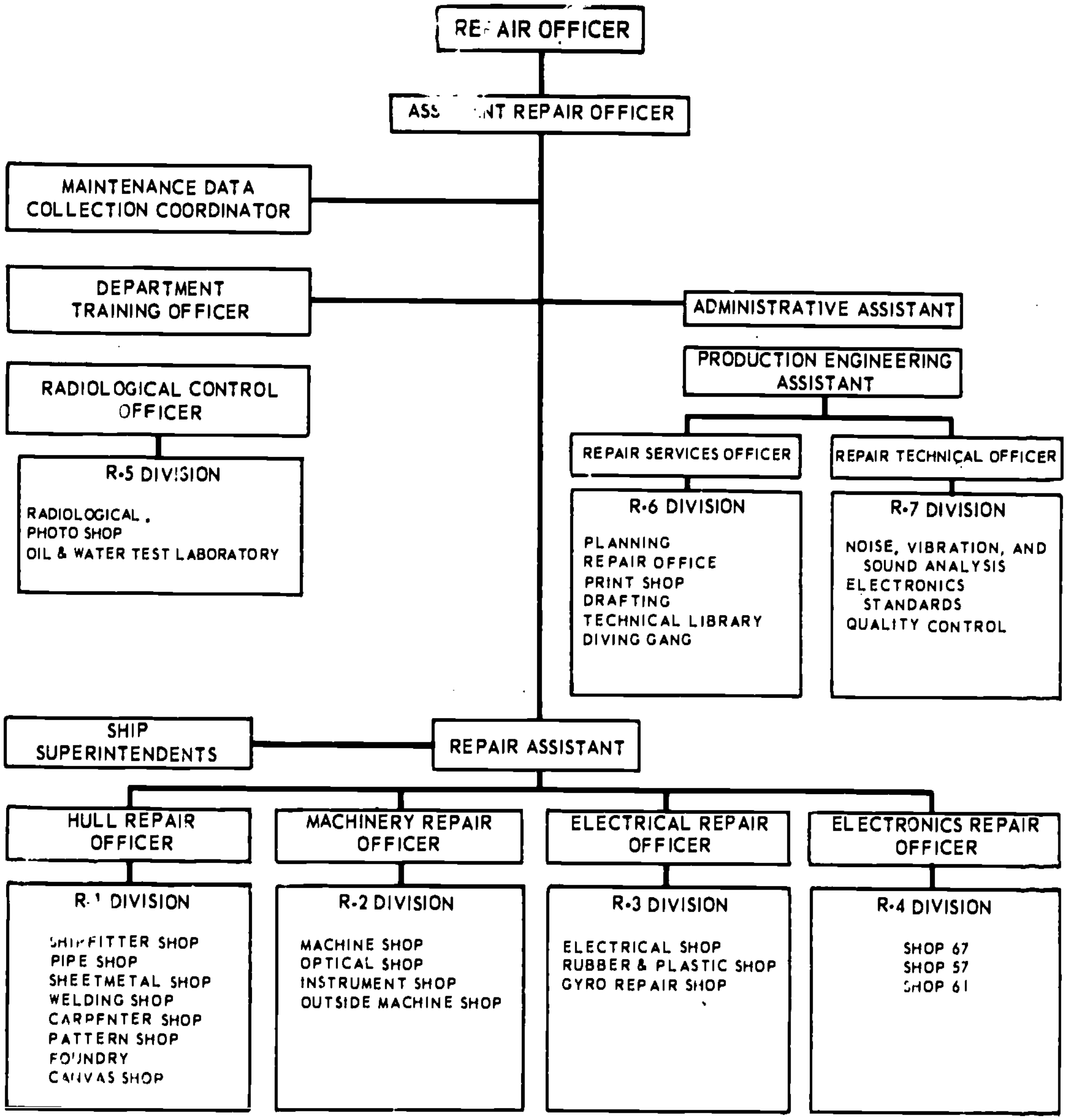
Figure 15-1.—Organization of repair department in typical repair ship (AR).

In comparing the two illustrations, you will notice several differences. For one thing, the repair department in the AR includes an ordnance repair division (R-5) which is not included in the repair department of the AS. Instead, the AS has a separate weapons repair department under a weapons repair officer. In all types of repair ships, you will probably be assigned to the R-2 division. The machine shop is normally within the R-2 division organization.

The duties of personnel in the repair department vary somewhat according to the type of ship. However, the following description of personnel functions will give you a general idea of the way things are in most repair departments.

REPAIR OFFICER

In a repair ship or tender, the repair officer is head of the repair department. As head of the repair department, the repair officer is responsible under the commanding officer for accomplishing repairs and alterations to the ships tended or granted availabilities. The repair officer is also responsible for accomplishing repairs and alterations to the ship itself (tender or repair ship) which are beyond the capacity of the engineering or other departments. The repair officer is responsible for maintaining a well-organized and efficiently operated department, or, in other words, for ensuring that subordinates perform as required. He issues and enforces repair department orders which govern department procedures. Like other department heads, he is also responsible for enforcing orders of higher authority. He must know the current workload and capacity of his crew and facilities, and keep the staff maintenance representative informed of the current status in order that the latter officer may properly schedule and assign ships. He is responsible for the review of work requests received via the staff maintenance representative from the ships assigned for repair and for acceptance or rejection of the individual jobs according to the capacity of his department. He is responsible for the review and acceptance of any work lists or work requests which develop after an availability period has started. He is also responsible for operating the department within the allotment granted and for



63.10A

Figure 15-2.—Organization of repair department in fleet ballistic missile submarine tender (AS).

the initiation of requests for further funds, if required. He must ensure the accuracy, correctness, and promptness of all correspondence, including messages, prepared for the commanding officer's signature. The repair officer is charged with the review of all personnel matters arising in all the divisions within the department such as training, advancement in rate, assignment to divisions, and leave. To acquire a thorough knowledge of conditions and ensure adequate standards, he must make frequent inspections of the department and require the division officers to make corrections as necessary.

Specific duties of the repair officer vary somewhat, depending upon the type of repair ship or tender. In general, however, a summary of the repair officer's duties include the following:

- Plan, prepare, and execute schedules covering alterations and repair work assigned to the repair department.

- Establish and operate the Planned Maintenance System of the 3-M System.

- Coordinate repair capabilities, work assignments, and available personnel to ensure maximum use of manpower.

- Supervise and inspect repairs and service to ensure timely and satisfactory completion of work; provide controls for quality control.

- Prepare records, reports, forms, and orders in connection with repair functions and duties.

- Ensure proper operation of all equipment and material assigned to the repair department.

- Ensure strict compliance with safety precautions and security measures.

- Report progress of major repairs and alterations to the commanding officer; keep the executive officer informed; report promptly any inability to meet scheduled completion dates.

ASSISTANT REPAIR OFFICER

In the absence of the repair officer, the assistant repair officer is charged with the responsibilities of the repair officer. As the assistant repair officer, he is the personnel administrator for the repair department. Under his cognizance are the assistant of personnel, the administrative control of the repair office, and the department control of training.

Specific duties of the assistant repair officer may vary somewhat, depending upon the type of repair ship or tender. In general, however, the duties of the assistant repair officer include the following:

- Assign personnel to divisions, schools, shore patrol, and beach guard.

- Have a basic knowledge of applicable courses, schools, and rating programs necessary to further education of personnel and their advancement in rate for the benefit of the personnel, the ship, and the Navy.

- Maintain the office stores and accounts.

- Assist the repair officer in all matters pertaining to general office routine, current availabilities of ships assigned to the repair ship or tender, and liaison between the repair office and the ship alongside and in shipyards.

- Review all work requests on receipt.

- Assign work and priority rating to the division and shops.

- Maintain liaison with the supply department for materials on order or to be ordered for the work requested.

- Procure the necessary blueprints, sketches, or samples for the shops.

- Schedule the services of tugs, cranes, and technical services, as available, for successful completion of an availability.

- Survey the report from the shop to ascertain the successful completion of all work during the allotted time.

- Analyze man-hour shop reports to determine an even balance of work versus personnel assigned. Regulate the coordination between the repair office and the shops toward the full productive capacity of the repair facilities.

In addition to the assistant repair officer, there are usually several other officers who assist the repair officer in the performance of repair department functions. These may include a production engineering assistant, a repair assistant, a radiological control officer, a department training officer, and an administrative assistant.

DIVISION OFFICERS

Each division within the repair department is under a division officer. The division officer may be a commissioned officer, a warrant officer, or a chief petty officer. The duties of the division officer vary, of course, according to the nature of the work done in the division.

ENLISTED PERSONNEL

As a Machinery Repairman assigned to the repair department of a repair shop or tender, you will work with people in a number of other ratings. It will be very much to your advantage to learn who these people are and what kind of work they do. Ratings that are often assigned to the repair department include (but are not limited to) Opticalmen, Electronics Technicians, Radiomen, Fire Control Technicians, Gunner's Mates, Draftsmen, Lithographers, Hull Maintenance Technicians, Patternmakers, Molders, Machinist's Mates, Boiler Technicians, Enginemen, Gas Turbine Systems Technicians, Electrician's Mates, and IC Electricians.

You can get some idea of the work done by people of these ratings by looking through the *Manual of Navy Enlisted Manpower and Personnel Classifications and Occupational Standards*, NAVPERS 18068 (revised). You can also learn about the work of these ratings by observing how the work is handled in the repair department. In handling repair work, it is often necessary for two or more shops (and two or more ratings) to cooperate to complete corrective maintenance actions.

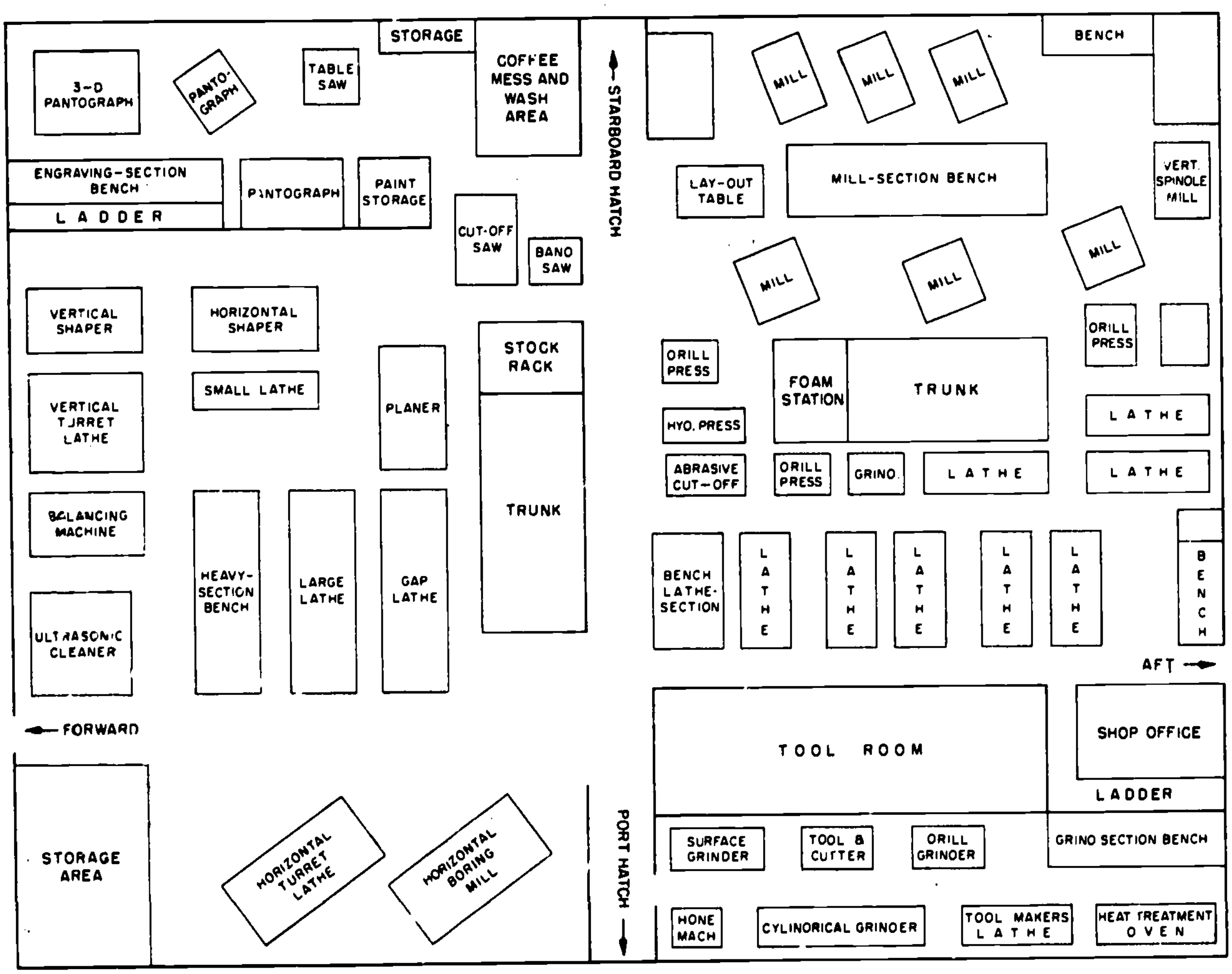
REPAIR DEPARTMENT SHOPS

Each shop in the repair department is assigned to one of the divisions. As a Machinery Repairman you will be concerned primarily with the machine shop. However, you will find it very useful to learn as much as you can about the other shops. After you have gotten acquainted with personnel in your own shop and have learned to find your way around your own working spaces, make an effort to find out something about the other shops in the division and department. Find out where each shop is located, what kind of work is done in each shop, and what administrative procedures are necessary when one shop must call on the services of another for assistance in completing corrective maintenance actions.

MACHINE SHOP LAYOUTS

Shop layout and arrangement vary somewhat from one ship to another depending upon space available, the nature and amount of equipment installed, and the services that must be provided by the ship. The following discussion is intended to give a general picture of a shop layout in AR, AS, and AD type ships. Figure 15-3 shows the layout of a Navy machine shop in a new submarine tender.

Most machine shops are broken down into sections as you can see in figure 15-3. These sections are lathe, milling, engraving, grinding, and heavy. Also included in the layout is a toolroom and shop office. The toolroom should be as centrally located as possible and be of adequate size to store all the tools needed for the work required of the shop.



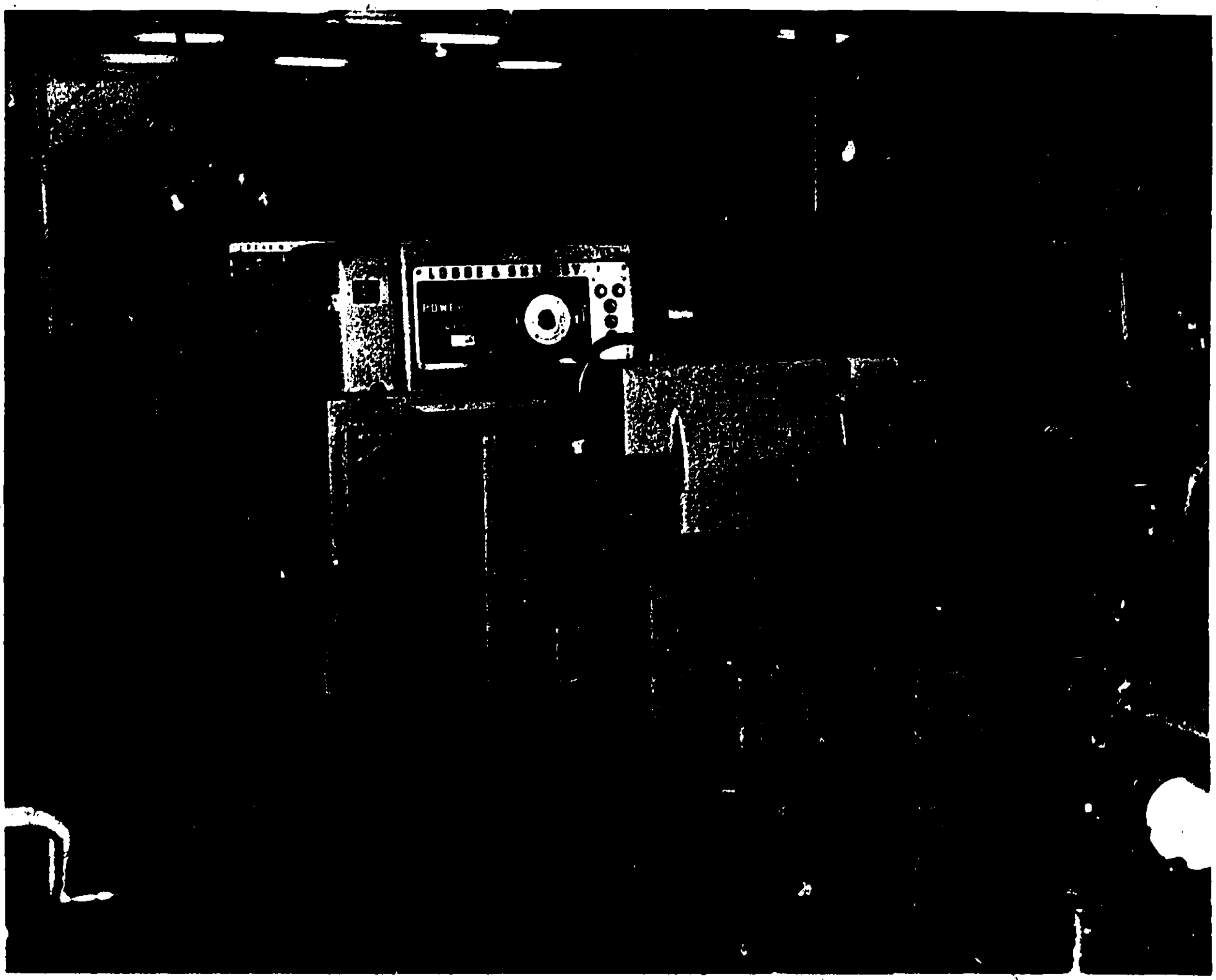
28.313

Figure 15-3.—Machine shop layout (new submarine tender).

The positioning of the machines is of great importance. In figure 15-4, you can see that the lathes are positioned headstock end to footstock end. This way the operators won't interfere with one another, and the chips from one machine will not fly in the direction of the next operator. Good lighting is of prime importance also. In figure 15-4 you can see good overhead lighting as well as work lights on the machines. The problem of one machine interfering with another is taken care of by angular placement as illustrated in figure 15-5. A good monorail system is another important asset to the machine shop. You can see in figure 15-5 that the monorail system covers all machines and work benches.

OTHER REPAIR SHOPS

As previously stated, you should get to know the other shops within the repair department. Machining is only a small portion of a Machinery Repairman's work. You can expect to work with every shop within the Repair Department. An example of this might be hatch dogs. The pattern shop makes the pattern, the



28.314

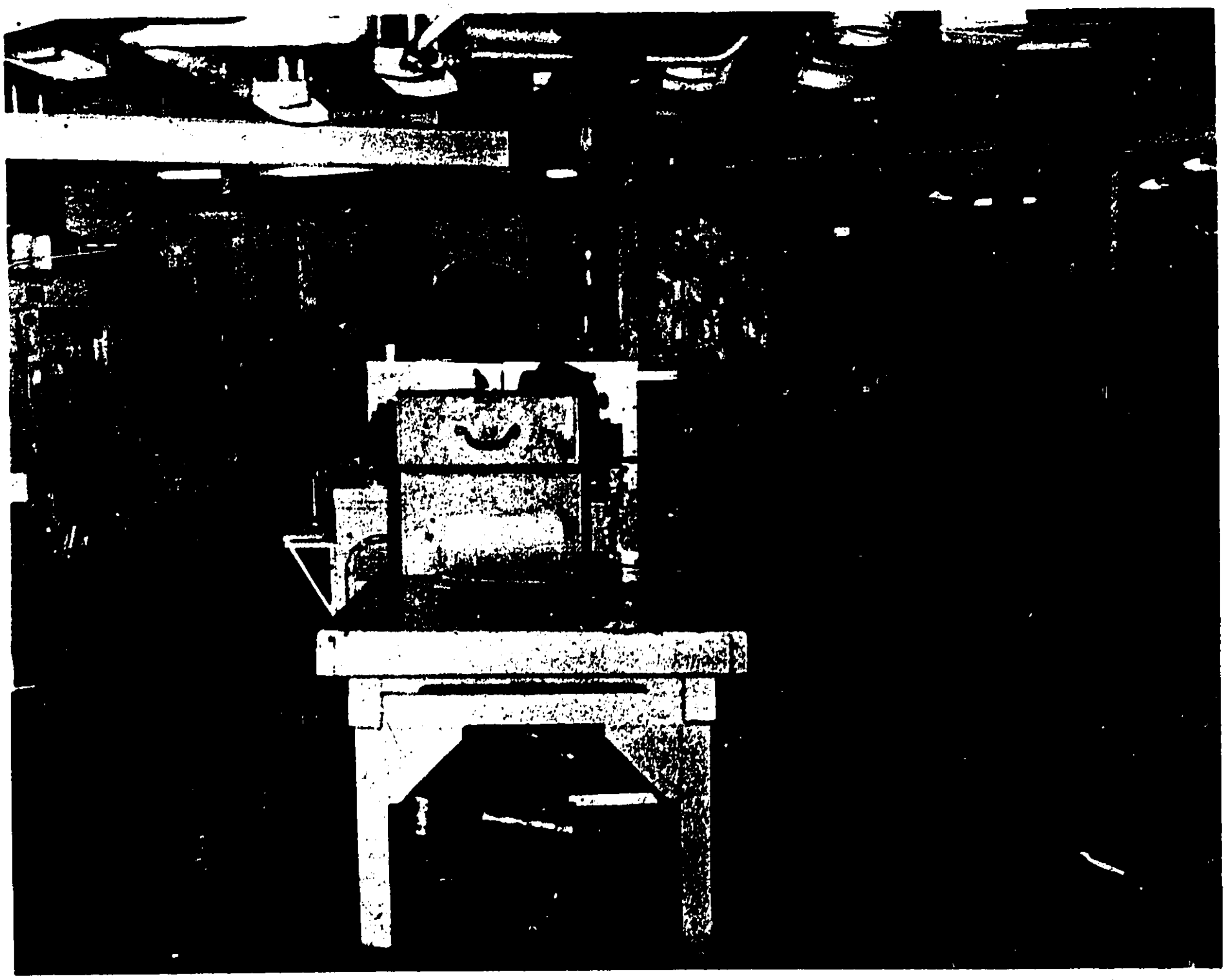
Figure 15-4.–Machine shop lathe section.

molders cast them in the foundry, the machine shop machines them, and possibly the outside repair shop installs them. You can see from this example that a smooth flow of work demands close cooperation between many shops.

REPAIR WORK

Replacement parts for most equipment are usually available through the Navy supply system. But occasionally, parts such as shafts and gears must be made in the machine shop (see fig. 15-6).

A major portion of the repair work done in shipboard machine shops involves machining worn or damaged parts so that they can be placed back in service. For example, the sealing surfaces of valves and pumps must be machined if leaks occur; broken studs must be removed, and bent or damaged shafts must be repaired. This repair work is usually more difficult than manufacturing work because of alignment problems in the machining operation.



28.315

Figure 15-5.—Machine shop milling section.

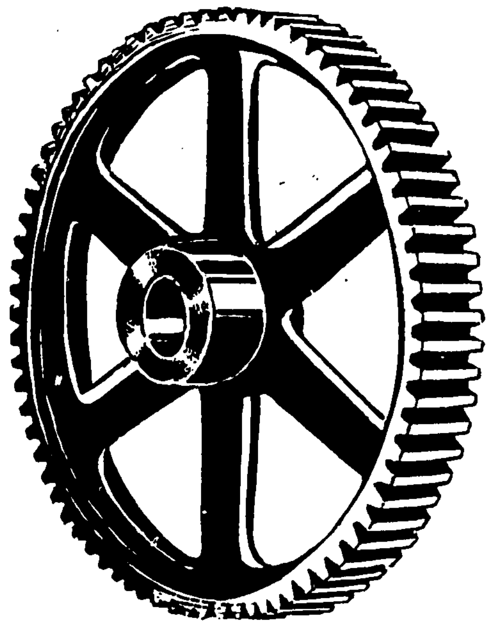
Many of the repair jobs that you will be assigned to do will require you to make certain mathematical calculations such as finding the areas of circles, rectangles, and triangles and calculating linear dimensions. You may also have to find the volume of cylinders and cubes. Information such as this may be obtained by using the formulas found in the various machinist's handbooks and in *Mathematics, Volume 1*, NAVPERS 10069-C.

When you are making a replacement part, the leading petty officer of the shop will usually give you a working drawing of the part or a sample part similar to the one required. Study the drawing or sample until you are familiar with the details and ensure that you have all pertinent information.

Decide which machines are required for making the part and calculate all necessary dimensions from the information provided. Choose the most logical sequence of machining operations so that the part is machined in a minimum number of setups.

GEARS

When manufacturing gears, you may need to calculate simple gear trains or gear trains using



28.360
Figure 15-6.—Part made in a machine shop.

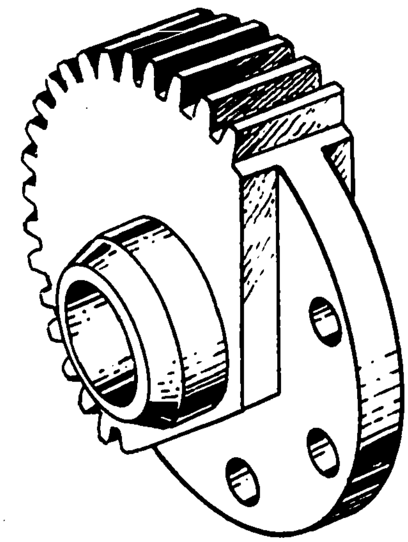
compound gearing. Information on this subject may be found in *Basic Machines*, NAVPERS 10624-A.

A gear is made by cutting a series of equally spaced, specially shaped grooves on the periphery of a wheel (see fig. 15-7). A rack is made by cutting similar grooves in a straight surface. The grooves and teeth of a spur gear are straight and parallel to the axis of the wheel.

To calculate the dimensions of a spur gear, you must know the terms used to designate the parts of the gear. In addition, you must know the formulas for finding the dimensions of the parts of a spur gear. To cut the gear you must know what cutter to use and how to index the blank so that the teeth are equally spaced and have the correct profile. In chapter 11 of this manual, you learned how to index.

Spur Gear Terminology

The following terms (see fig. 15-8) are used to describe gears and gear teeth (symbols in



28.361
Figure 15-7.—Cutting specially shaped grooves.

parantheses are standard gear nomenclature symbols used and taught at MR Schools):

OUTSIDE CIRCLE (OC): The circle formed by the tops of the gear teeth.

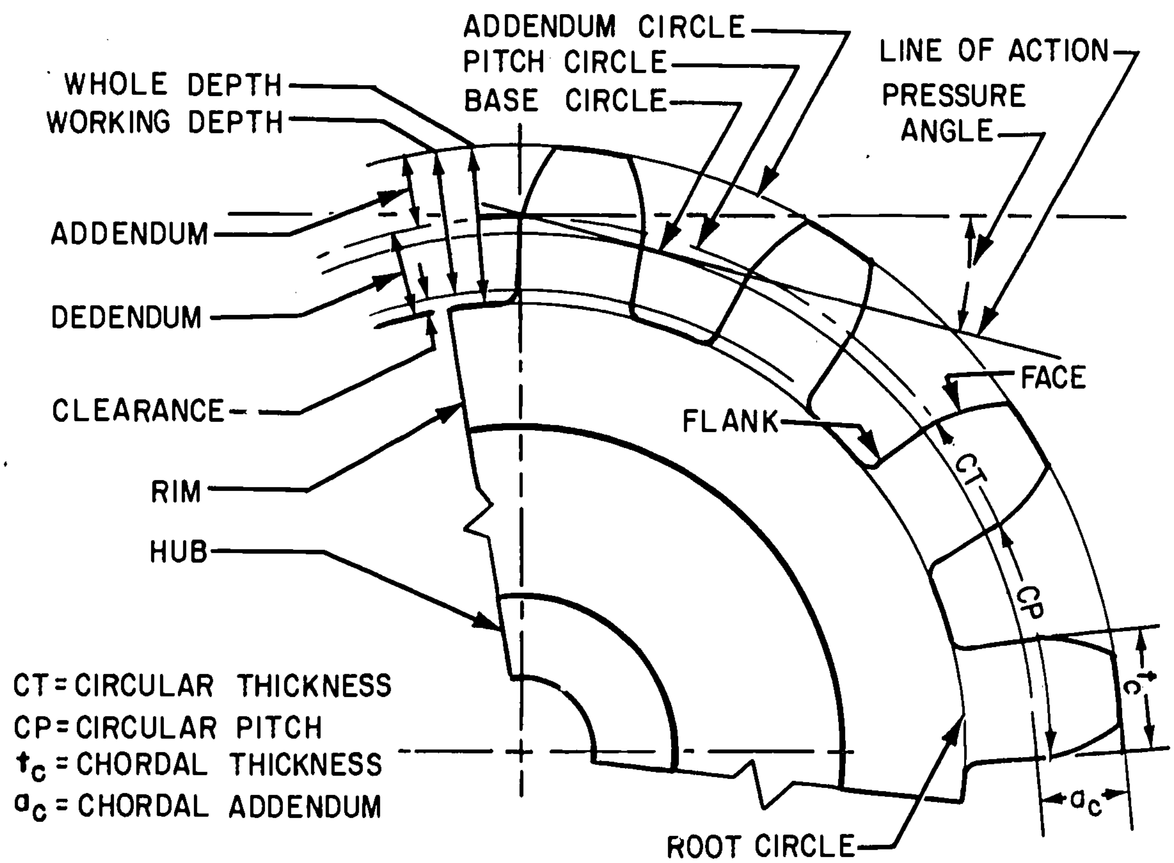
OUTSIDE DIAMETER (OD): The diameter to turn the blank to; the overall diameter of the gear.

PITCH CIRCLE (PC): (a) Contact point of mating gears, basis of all tooth dimensions. (b) Imaginary circle one addendum distance down the tooth.

PITCH DIAMETER (PD): (a) The diameter of the pitch circle. (b) In parallel shaft gears, the pitch diameter can be determined directly from the center to center distance and the number of teeth.

ROOT CIRCLE (RC): The circle formed by the bottoms of the gear teeth.

ROOT DIAMETER (RD): The distance from one side of the root circle to the opposite side passing through the center of the gear.



28.259.1

Figure 15-8.—Gear terminology.

ADDENDUM (ADD): The height of that part of the tooth extending outside the pitch circle.

CIRCULAR PITCH (CP): The distance from a point on one tooth to a corresponding point on the next tooth measured on the pitch circle.

CIRCULAR THICKNESS (CT): (a) One-half of the circular pitch. (b) The length of the arc between the two sides of a gear tooth, on the pitch circle.

CLEARANCE (CL): The space between the top of the tooth of one gear, and the bottom of the tooth of its mating gear.

DEDENDUM (DED): (a) The depth of the tooth inside of the pitch circle. (b) The radial distance between the root circle and the pitch circle.

WHOLE DEPTH (WD): The radial depth between the circle that bounds the top of the gear teeth and that which bounds the bottom.

WORKING DEPTH (WKD): (a) The whole depth minus the clearance. (b) The depth of engagement of two mating gears, the sum of their addendums.

CHORDAL THICKNESS (t_c): (a) The thickness of the tooth measured at the pitch circle. (b) The section of the tooth that is measured to see if the gear is cut correctly.

CHORDAL ADDENDUM (a_c): The distance from the top of a gear tooth to a chord subtending (to extend under) the intersections of the tooth thickness arc and the sides of the tooth (used for setting gear tooth vernier calipers for measuring tooth thickness).

DIAMETRAL PITCH (DP): (a) The most important calculation, it regulates the tooth size. (b) The ratio of the number of inches of the pitch diameter. (c) The number of teeth that will go into each inch of the pitch diameter evenly.

NUMBER OF TEETH (NT): The actual number of teeth of the gear.

BACKLASH (B): The difference between the tooth thickness and the tooth space of engaged gear teeth at the pitch circle.

The symbols used by the American Gear Manufacturers Association to describe gears and gear teeth are different from those used by the Navy. The following list will familiarize you with these symbols.

Spur Gear Terms	Machinery Repairman School Abbreviations	American Gear Manufacturers Association Abbreviations
Pitch Circle	PC	*
Pitch Diameter	PD	D
Center to Center Distance	C—C	C
Addendum	ADD	a
Dedendum	DED	d
Working Depth	WKD	h_k
Clearance	CL	c
Whole Depth	WD	h_t
Root Circle	RC	*
Outside Diameter	OD	D_o
Circular Thickness	CT	t_c
Circular Pitch	CP	P
Diametral Pitch	DP	P
Number of Teeth	NT	N
Root Diameter	RD	D_R
Chordal Thickness	t_c	*
Chordal Addendum	a_c	*

* None

Diametral Pitch System

The diametral pitch system was devised to simplify gear calculations and measurements. It is based on the diameter of the pitch circle, rather than on the circumference. Since the circumference of a circle is 3.1416 times its diameter, this constant must always be taken into consideration in calculating measurements based on the pitch circumference. In the diametral pitch system, however, the constant is in a sense "built into" the system, thus simplifying computation.

When using this system, there is no need to calculate circular pitch. Indexing devices based on the diametral pitch system will accurately space the teeth, and the formed cutter associated with the indexing device will form the teeth within the necessary accuracy. All calculations, such as center distance between gears and working depth of teeth, are simplified by the diametral pitch system.

Many formulas are used in calculating the dimensions of gears and the gear teeth. Only the formulas needed in this discussion are given here; a more complete list of formulas for calculating the dimensions of gears is provided in Appendix II of this manual. Appendix III contains explanations of how you determine the formulas to calculate the dimensions of gear teeth.

Usually the outside diameter (OD) of a gear and the number of teeth (NT) are available from a blueprint or a sample gear. Using these two known factors, you can calculate the necessary data.

For example, to make a gear 3.250 inches in diameter that has 24 teeth:

1. Find the pitch diameter (PD) using the formula:

$$PD = \frac{(NT)\ OD}{NT + 2}$$

$$PD = \frac{24 \times 3.250}{24 + 2} = \frac{78}{26} = 3.000 \text{ inches}$$

2. Find the diametral pitch (DP) using the formula:

$$DP = \frac{NT}{PD}$$

$$DP = \frac{24}{3} = 8$$

3. Find the whole depth of tooth (WD) by the formula:

$$WD = \frac{2.157}{DP}$$

$$WD = \frac{2.157}{8} = 0.2696 \text{ inch}$$

You can select the cutter for machining the gear teeth as soon as you have computed the diametral pitch. Formed gear cutters are made with eight different forms (numbered from 1 to 8) for each diametral pitch, depending upon the number of teeth for which the cutter is to be used. The following chart indicates the ranges of teeth for various cutters.

Number of cutter	Range of teeth
1	135 to a rack
2	55 to 134
3	35 to 54
4	26 to 34
5	21 to 25
6	17 to 20
7	14 to 16
8	12 to 13

If you need a cutter for a gear that has 24 teeth, use a number 5 cutter inasmuch as a number 5 cutter will cut all gears containing from 21 to 25 teeth. Most cutters are stamped, showing the number of the cutter, the diametral pitch, the range for the number of cutter, and the depth. The involute gear cutters usually (on-board a repair ship) run from 1 to 48 diametral pitch and 8 cutters to each pitch.

To check the dimensional accuracy of gear teeth, use a gear tooth vernier caliper (see fig. 15-9). The vertical scale is adjusted to the CHORDAL ADDENDUM (a_c) and the horizontal scale is used for finding the CHORDAL THICKNESS (t_c). Before calculating the chordal addendum, you must determine the arc tooth thickness (t) and addendum (a).

The formula for the addendum is

$$ADD = \frac{PD}{NT}$$

Using values from the preceding example,

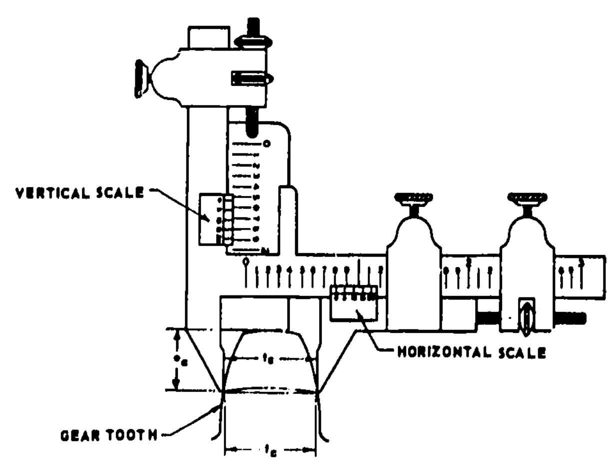
$$ADD = \frac{3.000}{24} = 0.125 \text{ inch}$$

The formula for circular thickness is

$$CT = \frac{1.5708}{DP}$$

Using values from the example,

$$CT = \frac{1.5708}{8} = 0.1964 \text{ inch}$$



28.260
Figure 15-9.—Measuring gear teeth with a vernier caliper.

The formula used for finding the chordal addendum is

$$a_c = ADD + \frac{(Ct)^2}{4(PD)}$$

$$a_c = 0.125 + \frac{(0.1964)^2}{4 X 3}$$

$$= 0.125 = \frac{0.0386}{12} = 0.128 \text{ inch}$$

The formula for the chordal tooth thickness is

$$t_c = PD \sin\left(\frac{90°}{NT}\right)$$

From the example,

$$t_c = 3 \text{ x } \sin\left(\frac{90°}{24}\right)$$

$$= 3 \text{ x } \sin 3°45''$$

$$= 3 \text{ x } 0.0654$$

$$= 0.1962 \text{ inch}$$

(Note: *Mathematics, Volume II,* NavPers 10071-B and various machinist's handbooks contain information on trigonometric functions.)

Now set the vertical scale of the gear tooth vernier caliper to 0.128 inch. Adjust the caliper so that the jaws touch each side of the tooth as shown in figure 15-9. If the reading on the horizontal scale is 0.1962 inch, the tooth has correct dimensions; if the dimension is greater, the tooth groove is too shallow; if the reading is less, the tooth groove is too deep.

Sometimes you cannot determine the outside diameter of a gear or the number of teeth from available information. However, if a gear dimension and a tooth dimension can be found, these dimensions can be keyed into one or more of the formulas in Appendix II of this training course so that the required dimensions can be calculated.

Machining the Gear

The procedures for making a gear of the dimensions given in the preceding example are as follows:

1. Select and cut a piece of stock to make the blank. Allow at least 1/8 inch on the diameter and thickness of the blank for cleanup cuts.

2. Check the stock in a chuck on a lathe, and at the center of the blank, face an area slightly larger than the diameter of the hole required.

3. Drill and bore to required size (within tolerance).

4. Remove the blank from the lathe and press the blank on a mandrel.

5. Set the mandrel and gear blank up between centers on the lathe and machine to proper dimensions.

6. Remove the gear blank and mandrel from the lathe and set up between center of the index head and footstock on the milling machine. Dial within tolerance.

7. Select a number 5 involute gear cutter (8-pitch) and mount and center as described in chapter 11 of this manual.

8. Set the index head for indexing 24 divisions.

9. Start the milling machine spindle and move the table up until the cutter just touches the gear blank. Set the micrometer collar on the vertical feed handwheel to zero, then feed the table up toward the cutter slightly less than the whole depth of tooth.

10. Cut one tooth groove, index the workpiece for one division and take another cut. Check the tooth dimensions with a vernier gear tooth caliper as described previously. Make the required adjustments to provide an accurately "sized" tooth.

11. Continue indexing and cutting until the teeth are cut around the circumference of the workpiece.

When machining a rack, space the teeth by moving the worktable an amount equal to the circular pitch of the gear for each tooth cut.

Calculate circular pitch by dividing 3.1416 by the diametral pitch or

$$CP = \frac{3.1416}{DP}$$

Calculations for corrected addendum and chordal pitch need not be made for checking rack teeth dimensions because on racks the addendum is a straight line dimension and the tooth thickness is one-half the linear pitch.

SHAFTS

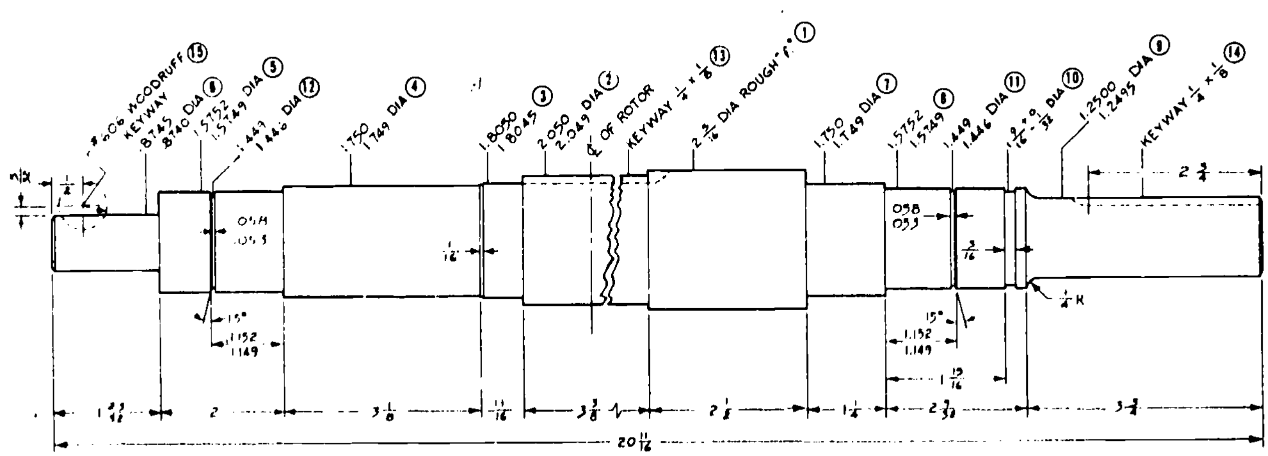
The repair or manufacture of components such as pump or rotor shafts is an important part of machine shop work. Information provided here will help you to see the proper method of manufacturing a new shaft and also the proper method of repairing a bent or damaged shaft.

Manufacturing a New Shaft

Figure 15-10 illustrates a shaft that might be made in the machine shop. The information given in the illustration is normally available in the manufacturer's technical manual for the machinery component for which the shaft is required. The encircled numbers indicate a sequence of operations for machining the various surfaces of that shaft.

Select and cut a piece of round stock which is at least 1/16 inch larger in diameter and 1/8 inch longer than the shaft. Face and centerdrill each end of the stock. In facing, ensure that the workpiece is faced to the correct length for the shaft, which in this example is 20 11/16 inches. Most of the linear dimensions in figure 15-10 are given in the form of mixed numbers or proper fractions which indicate that rule measurement of these dimensions will be sufficiently accurate. In manufacturing a new shaft, you must take all linear dimensions from the same reference point to ensure the correct lengths. However, the linear position of the grooves at numbers 11 and 12 are in the form of decimal fractions and require greater accuracy than is available by rule measurement.

The shaft shown in figure 15-10 may be machined in two lathe setups and two mill setups. Plain turning required on surfaces 1 through 6 (see encircled numbers in figure 15-10) is performed in the first lathe setup; surfaces 7 through 12 are machined in the second lathe setup. Keyways 13 and 14 are machined in the first milling setup and then the cutter is changed for machining the Woodruff keyway



28.261

Figure 15-10.—Steps in making a shaft.

(15). For machining the shaft, take the following steps:

1. Turn the workpiece to a 2 3/16 inch diameter. Check the diameter for taper and make corrections as necessary.

2. Set hermaphrodite calipers to 1 1 3/32 inches and lay out the shoulder between the 2 3/16 inch diameter and 2.050 inch diameter. Using the crossfeed handwheel with the micrometer collar set on zero, feed the tool in 0.068 inch (which is one-half of the difference you get by subtracting 2.050 from 2 3/16). Make a short length of cut at the end of the shaft and measure the diameter with micrometers. Adjust the crossfeed handwheel as required to provide the $2.050\,^{+.000}_{-.001}$ diameter and complete the cut to the layout line.

3. Use procedures similar to those described in step 2 for machining surfaces 3 through 6. Be extremely careful to accurately measure the diameter of the beginning of each cut to ensure that you hold the dimensions within the range provided in the illustrations.

4. Turn the workpiece end-for-end and machine surfaces 7, 8, and 9 as described in step 2.

5. Set a 3/16-inch parting tool in the toolholder and (by rule measurement) position the tool for making groove 10 and make the groove.

6. Set the compound rest parallel to the axis of the workpiece for laying out grooves 11 and 12. Place a sharp pointed tool in the toolholder and align the point of the tool with the shoulder between surfaces 7 and 8. Then use the compound rest to move the tool 1.152 inches longitudinally as indicated by the micrometer collar on the compound feed screw. Feed the tool toward the work with the crossfeed until a thin line is scribed on the surface of the workpiece. Now swivel the compound to the angle required for cutting the chamfer and cut the chamfer. (Calculate the angular depth from the given dimensions.) Then using a parting tool between 0.053 and 0.058 inch wide, make the groove.

7. With a fine cut file, remove all sharp edges from shoulders and grooves.

8. Remove the shaft from the lathe and mount in the milling machine and mill the keyways to the required dimensions.

Because relatively small depths of cuts are taken, each surface of this shaft is finished in one pass. Check the dead center frequently to see that it does not overheat and to prevent the workpiece from becoming loose on the center. Use a center rest as necessary, for supporting the work.

Repairing Shafts

Bent shafts 1 1/4 inches and less in diameter which are used for low-speed operations can be straightened so that they have less than 0.003- to 0.004-inch runout. Before attempting to straighten a shaft, however, always ensure that the leading petty officer of the shop is informed of the operation. To straighten a shaft take the following steps:

1. Mount the shaft between centers in a lathe. If the shaft is too long for mounting between centers, mount it in a 4-jaw chuck and center rest.

2. Clamp a dial indicator on the compound rest and locate the area of the bend and measure how much the shaft is bent (runout). To determine the area of the bend, run the dial indicator along the shaft longitudinally. The greatest variation of the pointer from zero indicates the bend area. With the dial indicator set at this point, rotate the shaft and note the amount of fluctuation of the pointer. This fluctuation is the amount of runout. Mark the longitudinal position of the bend and the high side of the bend with chalk or a grease pencil.

3. Remove the shaft from the lathe and place it on a hydraulic press. Place a V-block on each side of the bend area and turn the shaft so that the high side is up. Move the press ram downward until it touches the shaft. Set up a dial indicator so that the contact point contacts the high side of the shaft as near to the ram as possible.

4. Carefully apply pressure on the shaft with the ram. Watch the pointer of the dial indicator to determine how much the shaft is "sprung" in the direction opposite the bend.

When the indicator reading is 0.002 or 0.003 inch greater than the amount of runout. release the ram pressure.

5. Set up the shaft between centers and check again as explained in step 1. Repeat steps 2, 3, and 4 until the runout is decreased to the required limits.

If little or no change in runout results from the first straightening procedure, the shaft should be sprung farther in the second operation to overcome the elasticity of the shaft so that it bends in the required direction. However, it is better to use several steps for straightening the shaft a few thousandths of an inch at a time than to attempt to straighten the shaft in one or two steps with the possibility of bending the shaft too far in the direction opposite to that of the original bend.

Damaged ends of shafts can be repaired by removing the bad section and replacing it with a new stub end. Check with the type commander to ensure that stubbing of shafts is allowed.

Take the following steps when stubbing a shaft:

1. If a blueprint is not available, make a drawing of the shaft showing all dimensions.
2. Machine a piece of scrap stock (spud) in the lathe to the diameter of the shaft at the point where the center rest will be used. Carefully align the center rest on this spud.
3. Mount the undamaged end of the shaft in a 4-jaw chuck and "zero in" the shaft near the jaws of the chuck. Use soft jaws or aluminum shims to prevent damage to the shaft surface.
4. Position the previously set center rest under the shaft so that the center rest is between the chuck and the damaged end of the shaft.
5. Cut off the damaged portion of the shaft.
6. Face, centerdrill, and drill the end of the shaft. The diameter of the hole should be about 5/8 of the diameter of the shaft; the depth of the hole should be at least 1 1/2 times the hole diameter.
7. Chamfer the end of the shaft liberally to allow space for weld deposits.
8. Make a stub of the same material as the shaft. The stub should be 1/4 inch larger in diameter and 3/8 inch longer than the damaged portion of the shaft plus the depth of the hole drilled in the shaft. This provides ample machining allowance.
9. Machine one end of the stub to a press fit diameter of the hole in the shaft. The length of this portion should be slightly less than the depth of the hole in the shaft. (A screw fit between the shaft and stub can be used instead of the press fit.)
10. Chamfer the shoulder of the machined end of the stub the same amount as the shaft is chamfered.
11. Press (or screw for a threaded fitting) the stub into the shaft and have the chamfered joint welded and stress relieved.
12. Mount the shaft with welded stub back in the lathe and machine the stub to the original shaft dimensions provided by the drawing or blueprint.

VALVES

In repairing valves, you must have a knowledge of the materials from which they are made. Each material has its limitations of pressure and temperature; therefore, the materials used in each type of valve depend upon the temperatures and pressures of the fluids which they control.

Valves are usually made of bronze, brass, cast or malleable iron, or steel. Steel valves are either cast or forged and are made of either plain steel or alloy steel. Alloy steel valves are used in high-pressure, high-temperature systems; the disks and seats of these valves are usually surfaced with a chromium-cobalt alloy known as Stellite. This material is extremely hard.

Brass and bronze valves are never used for temperatures exceeding 550°F. Steel valves are used for all services above 550°F and for lower temperatures where conditions, either internal or external, such as high-pressure, vibrations, or shock, may be too severe for brass or iron. Bronze valves are used almost exclusively in systems carrying saltwater. The seats and disks of these valves are usually made of Monel, an excellent corrosion- and erosion-resistant metal.

Information on the commonly used types of valves and their construction is provided in *Fireman*, NAVEDTRA 10520-E. The information supplied here applies to gate and globe valves but the procedures discussed can usually be adapted for repairing any type of valve.

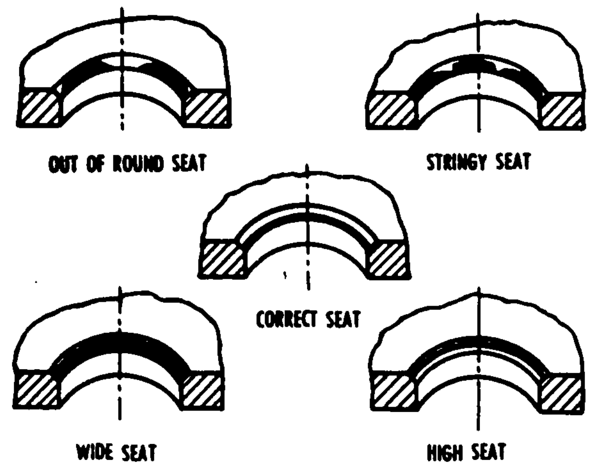
Glove Valves

Closely inspect the valve seat and disk for erosion, cuts on the seating area, and proper fit of disk to seat. Inspect all other parts of the valve for wear and alignment and, if found defective, repair or renew. Generally, valve repair is limited to overhaul of the seat and disk. Overhauling of the disk and seat is usually done by grinding-in the valve seat and disk or by lapping the seat and machining the disk in a lathe. Where the disk and seat surfaces cannot be reconditioned by grinding or lapping, you must machine both the valve disk and valve seat in a lathe.

If upon inspection, the disk and seat appear to be in good condition, spot them in to find out whether they actually are in good condition.

SPOTTING-IN.—The method used to visually determine whether or not the seat or disk make good contact with each other is called spotting-in. To spot-in a valve seat, first apply a thin coating of prussian blue evenly over the entire machined face surface of the disk, then insert the disk into the valve and rotate it a quarter turn, using a light downward force at the same time. The prussian blue will adhere to the valve seat at points where the disk makes contact. Figure 15-11 shows what a correct seat looks like upon spotting-in, and also shows what various kinds of imperfect seats look like upon spotting-in. After you have noted the condition of the seat surface, wipe all the prussian blue off the disk face surface and apply a thin, even coat of prussian blue on the contact face of the seat and again place the disk on the valve seat and rotate the disk a quarter turn. Examine the resulting blue ring on the valve disk. If the ring is unbroken and of uniform width, the disk is in good condition, if there are no cuts, scars, or irregularities on its face. If the ring is broken or wavy, the disk is not making proper contact with the seat and must be machined.

GRINDING.—Valve grinding is the method of removing small irregularities from the contacting surfaces of the seat and disk. This



8.71

Figure 15-11.—Examples of spotted-in valve seats.

process is also used to follow up all seat or disk machining work on a valve.

To grind-in a valve, apply a small amount of grinding compound to the face of the disk, insert the disk into the valve and rotate the disk back and forth about a quarter turn. Shift the disk-seat relation from time to time so that the disk will be rotated gradually in increments through several rotations. During the grinding process, the grinding compound will gradually be displaced from between the seat and disk surfaces, so you must stop every minute or so to replenish the compound. For best results when you do this, wipe both seat and disk clean and then apply the compound to the disk face.

When it appears that the irregularities have been removed, spot-in the disk to the seat as described previously.

When a machined valve seat and disk are initially spotted-in, the seat contact will be very narrow and located close to the edge of the bore. Grinding-in, using finer compounds as the work progresses, causes the seat contact to become broader until a seat contact is produced as illustrated in figure 15-11. The contact area should be a perfect ring, covering approximately one-third of the seating surface, as shown in the correct seat in figure 15-11.

Avoid overgrinding. It will produce a groove in the seating surface of the disk and also will tend to round off the straight angular surface of the seat. The effects of overgrinding can be corrected only by machining the surfaces.

LAPPING.—Lapping is the truing of a valve seat surface by means of a cast iron lapping tool, shaped like and of exactly the same size as the disk for that particular valve.

By use of such a tool, you can remove slightly larger irregularities from the seat than it is possible to remove by grinding the disk to the seat. (See fig. 15-12.) NEVER USE THE VALVE DISK AS A LAP.

Below is a summary of the essential points you must keep in mind while using the lapping tool.

1. Do not bear heavily on the handle of the lap.

2. Do not bear sideways on the handle of the lap.

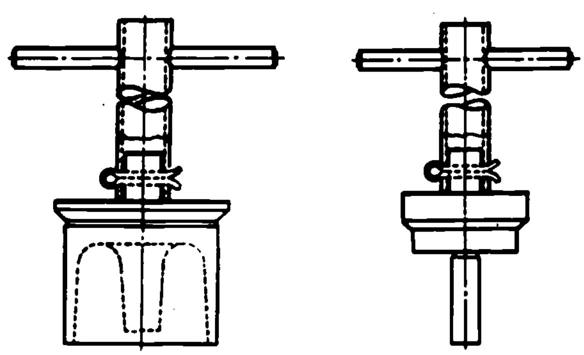
3. Shift the lap-valve seat relation so that the lap will gradually and slowly rotate around the entire seat circle.

4. Check the working surface of the lap; if a groove wears on it, have the lap refaced.

5. Use only clean compound.

6. Replace the compound often.

7. Spread the compound evenly and lightly.



11.353

Figure 15-12.—Lapping tools.

8. Do not lap more than is necessary to produce a smooth and even seat.

9. Always use a fine grinding compound to finish the lapping job.

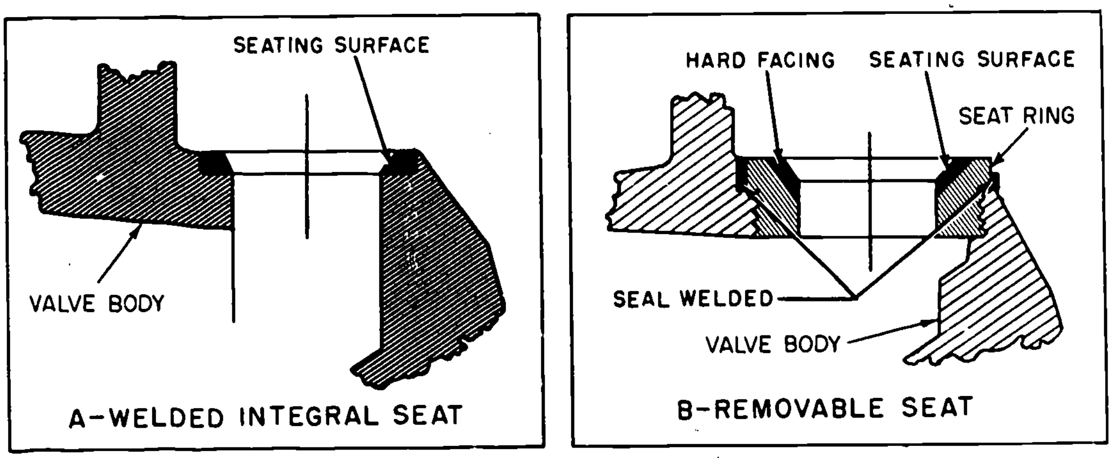
10. Upon completion of the lapping job, spot-in and grind-in the disk to seat.

Abrasive compound for grinding-in and lapping-in valve seats and disks is available in Navy stock in four grades. The grades and the recommended sequence of use are as follows:

GRADE	USE
Coarse	For lapping-in seats that have deep cuts and scratches or extensive erosion.
Medium	For following up the coarse grade; may be used also at the start of reconditioning process where damage is not too severe.
Fine	For use when the reconditioning process nears completion.
Microscopic fine	For finish lapping-in and for final grinding-in purposes.

REFACING.—If the seat of a valve has been deeply cut, scored, or corroded to the extent that lapping will not correct the condition, it must be machined, or, in an extreme case, a new seat installed.

Many valves have removable seats which are threaded, welded, threaded and welded, or pressed into the valve body. In A of figure 15-13, the valve seating surface has been welded so that it has become an integral part of the valve body. In B of figure 15-13, the seating surface has been welded so that it has become an integral part of the seat ring. The seat ring is

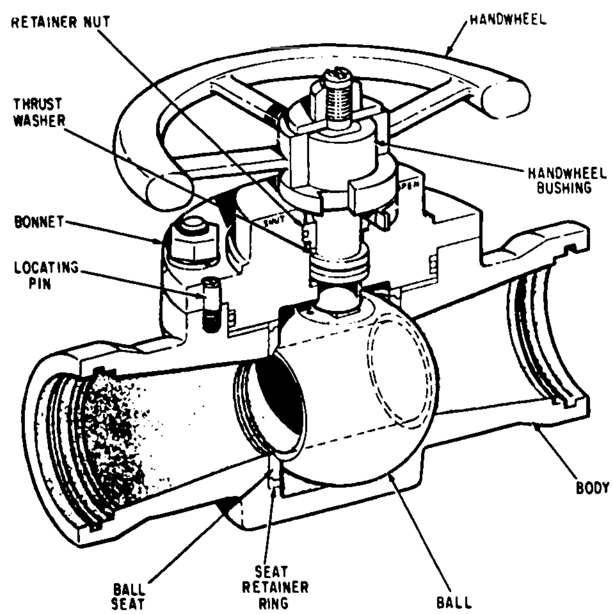


28.262

Figure 15-13.—Valve seat construction.

threaded into the body and seal-welded after installation. If the seating surface of A is damaged to the extent that it must be renewed, you need only to remove the existing weld material by machining and then rebuild the seating surface with successive deposits of new weld material. After a sufficient deposit of weld material has been made, you can machine a new seating surface. If the seating surface of B requires renewal, you must first machine the seal weld from the ring and remove it from the valve body. A new seat ring may then be installed or the existing seat surface may be removed, rebuilt, and machined as previously described.

The actual machining operations for valve seats and disks are described in chapter 8. After machining is completed, the seat and disk must be spotted-in, ground-in lightly, and respotted to ensure that the valve disk-seat contact is as it should be.



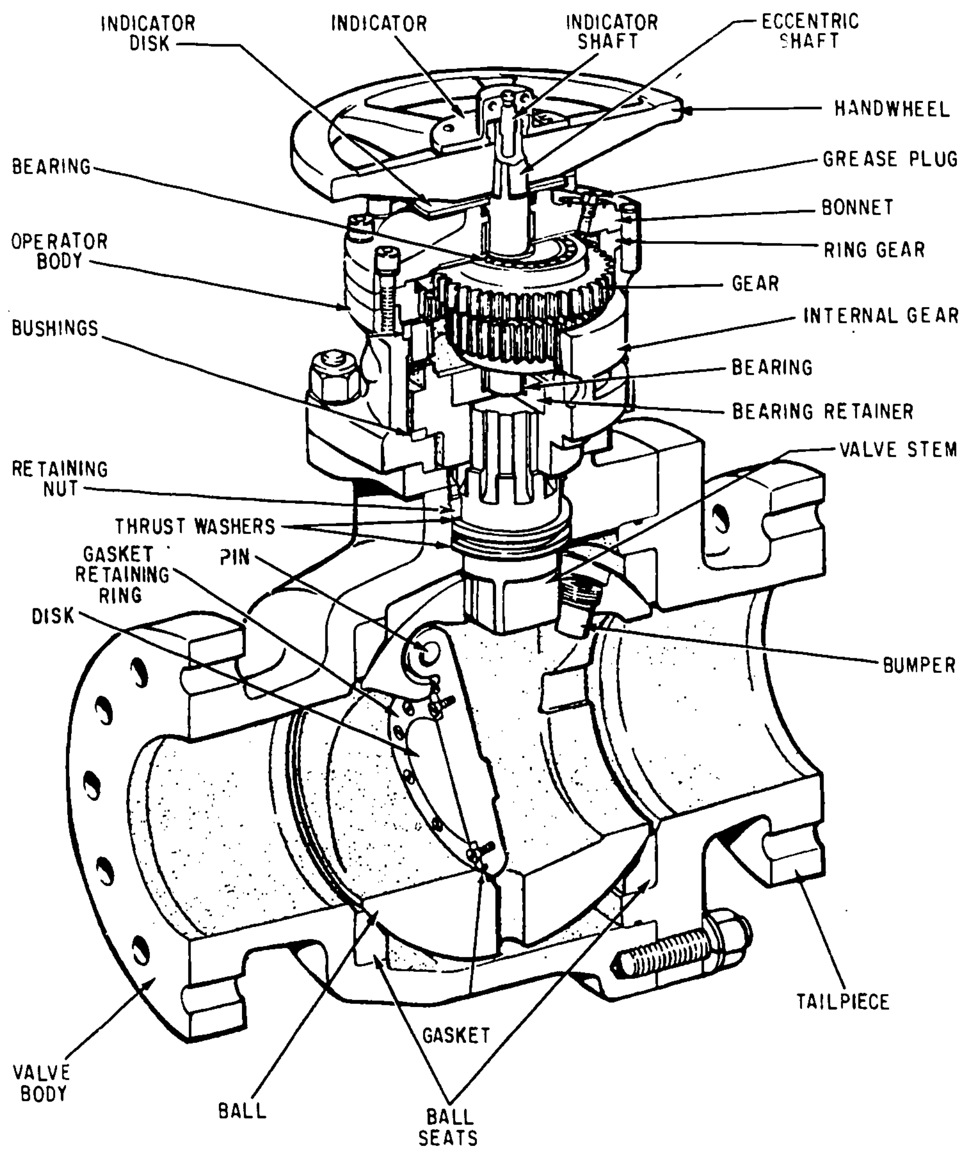
47.196

Figure 15-14.—Typical seawater ball valve.

Ball Valves

Ball valves, as the name implies, are stop valves that use a ball to stop or start the flow of fluid. The ball, shown in figure 15-14 performs the same function as the disk in a globe valve. When you turn the handwheel to open the valve, the ball rotates to a point where the hole through the ball is in line with the valve body inlet and outlet. When you shut the valve, which requires only a 90° rotation of the handwheel for most valves, the ball rotates so that the hole is perpendicular to the flow openings of the valve body, and the flow stops.

Most ball valves are the quick-acting type (requiring only a 90° turn of a simple lever or handwheel to completely open or close the valve), but many are planetary gear operated. This type of gearing requires a relatively small handwheel and operating force to operate a fairly large valve. The gearing does, however, increase the time for opening or closing the valve. Some ball valves have a swing-check located within the ball to give the valve a check valve feature. Figure 15-15 shows a ball-stop swing-check valve with planetary gear operation. Ball valves are normally found in the following systems on-board ship: seawater, sanitary, trim and drain, air, hydraulic and oil transfer. Repair procedures for ball valves can be found in Portsmouth Process Instructions discussed below. In the case of the smaller types, repairs



47.197

Figure 15-15.—Typical ball stop swing-check valve for seawater service.

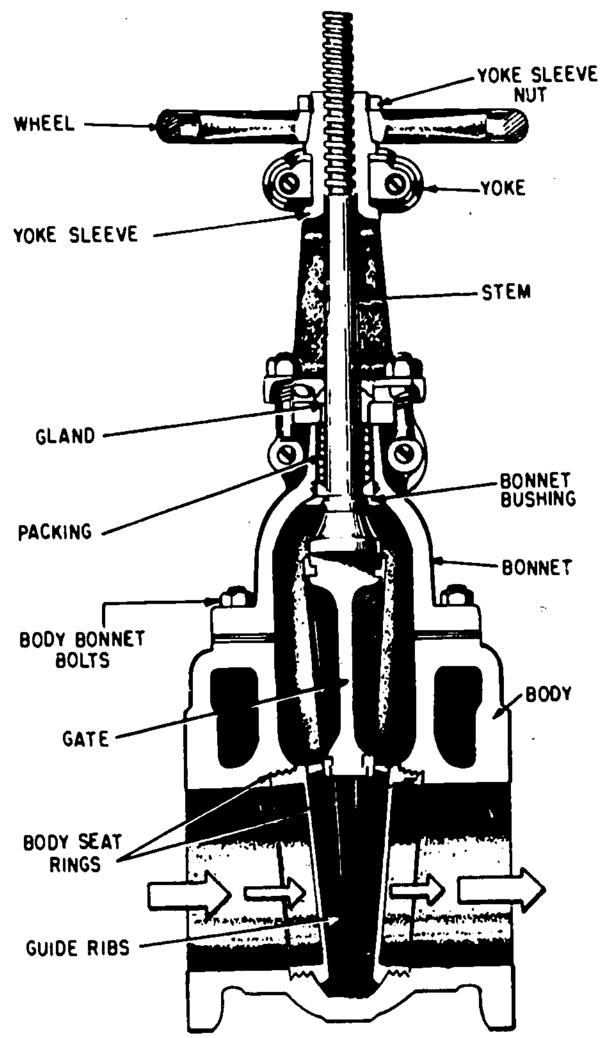
consist of part replacements rather than machining and rebuilding.

There are two basic instructions published by Portsmouth Naval Shipyard which are guidelines in the repair procedures of seawater ball valves and the balls themselves. In most cases the most common repair to the ball itself is to pit fill any erosion and recoat the ball. The guidelines for this process are covered in Portsmouth Process Instruction number 4820-917-338D change 1 of 31 January 1977. The other instruction which covers the actual valve body is the PPI 4820-921-339B. The latter instruction applies to the repair of seawater ball valves when the waterway lip area has been corroded/eroded to the extent that its function is reduced and serviceability is affected. The repair of valve ball waterway lips in this instruction applies only to straight waterway valves whose stem connection does not enter the waterway and those without sea chest blow holes. This instruction also applies to the repair of the stem cavity and O-ring sealing areas and to seawater ball valves whose back seat areas are corroded and eroded to the extent that leakage between the valve seat and back seat areas exceeds allowable leakage requirements. The detailed repair steps are in Portsmouth Process Instruction Number 4820-921-339B of 24 June 1977 which cancels number 4820-921-339A.

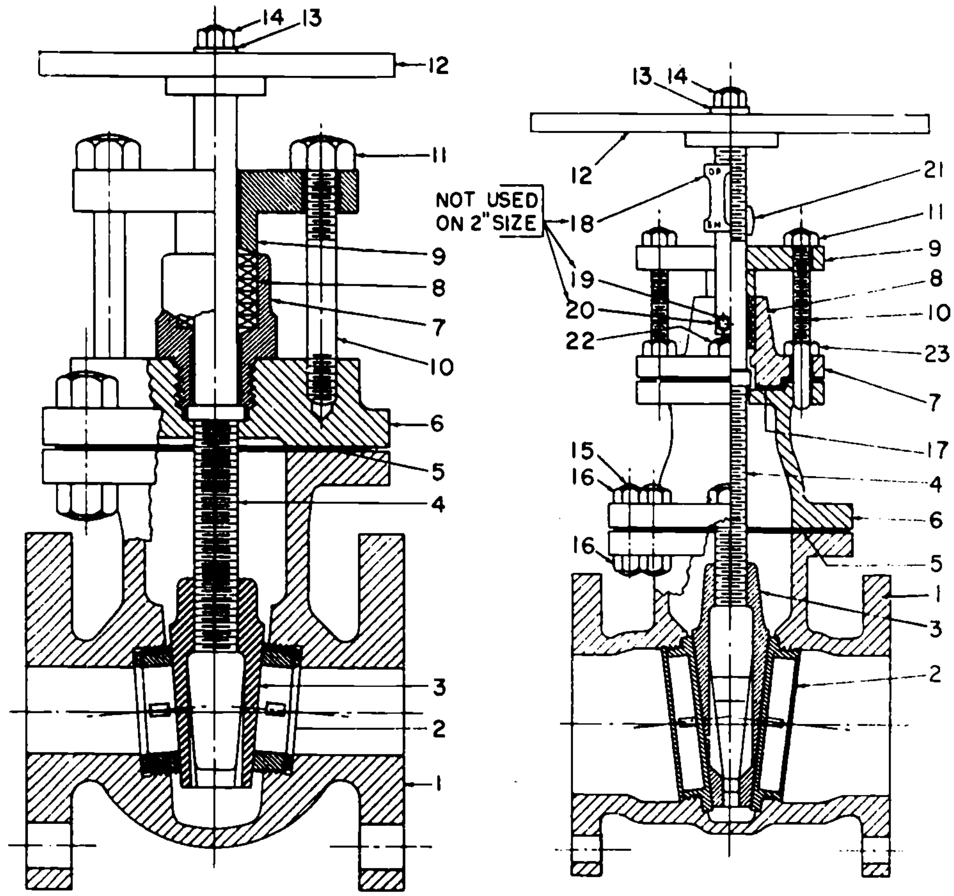
Gate Valve

Gate valves are used when a straight line flow of fluid with minimum flow restriction is desired. Gate valves are so named because the part (gate) which either stops or allows flow through the valve acts somewhat like the opening or closing of a gate. The gate is usually wedge shaped. When the valve is wide open, the gate is fully drawn up into the valve, leaving an opening for flow through the valve which is the same size as the pipe in which the valve is installed. Gate valves are not suitable for throttling purposes since the control of flow would be difficult due to turbulence, and fluid force against a partially open gate causes it to vibrate, resulting in extensive damage to the valve.

Gate valves are classified as either rising stem (fig. 15-16) or nonrising stem valves (fig. 15-17). On the nonrising stem gate valves, the stem is threaded on the lower end into the gate. As you rotate the handwheel on the stem, the gate travels up or down the stem on the threads while the stem remains vertically stationary. This type valve almost always has a pointer type indicator



11.317X
Figure 15-16.—Cutaway view of gate stop valve (rising stem type).



LIST OF PARTS			
PART NO.	NAME OF PART	PART NO.	NAME OF PART
1	BODY	13	HANDWHEEL WASHER
2	SEAT RING	14	HANDWHEEL NUT
3	GATE	15	BONNET STUD
4	STEM	16	BONNET STUD NUT
5	BONNET GASKET	17	STUFFING BOX GASKET
6	BONNET	18	INDICATOR PLATE
7	STUFFING BOX	19	LOCK WASHER
8	FACKING	20	INDICATOR PLATE SCREW
9	GLAND	21	INDICATOR NUT
10	GLAND STUD	22	STUFFING BOX STUD
11	GLAND STUD NUT	23	STUFFING BOX STUD NUT
12	HANDWHEEL		

38.118

Figure 15-17.—Cross-sectional views of gate stop valves (nonrising stem type).

threaded onto the upper end of the stem to indicate valve position.

The rising stem gate valve (fig. 15-16) has the stem attached to the gate, and the gate and the stem rise and lower together as the valve is operated. With this basic information on the principles of the gate valve, you are ready to learn about repair procedures and manufacturing of repair parts.

Defects such as light pitting or scoring and imperfect seat contact can be corrected best by lapping. Use a lapping tool designed for the type of valve to be reconditioned. NEVER use the gate as a lap.

The lapping process is the same for gate valves as for globe valves, but you turn the lap by a handle extending through the inlet or outlet end of the valve body. Insert the lapping tool, minus the handle, into the valve so that you cover one of the seat rings. Then attach the handle to the lap and begin the lapping work. You can lap the wedge gate to a true surface, using the same lap that you used on the seat rings. In some cases when a gate is worn beyond repair and a shim behind the seat will not give a proper seat, it is possible to plate the gate or seat, or both, as described in chapter 14, Electroplating. (Note: Shim has to be applied behind both seats to maintain proper angle.) It is also possible to weld repair the damaged gate and then machine it to its original specifications in either a mill or lathe, using an angle plate or fixture. One of the advantages of plating over the weld repair method is that no heat is involved in the selective brush plating method. Building up metal by welding always heats the surfaces being repaired and can cause loss of temper or other weaknesses in the metal.

Constant-Pressure Governor

Many turbine driven pumps are fitted with constant-pressure pump governors. A constant-pressure pump governor maintains a constant pump discharge pressure under varying conditions of load. The governor, which is installed in the steam line to the pump, controls the amount of steam admitted to the driving turbine, thereby controlling the pump discharge pressure.

Two types of constant-pressure pump governors are used by the Navy—the Leslie and the Atlas. The two types of governors are very similar in operating principles. Our discussion is based on the Leslie governor, but most of the information applies also to the Atlas governor.

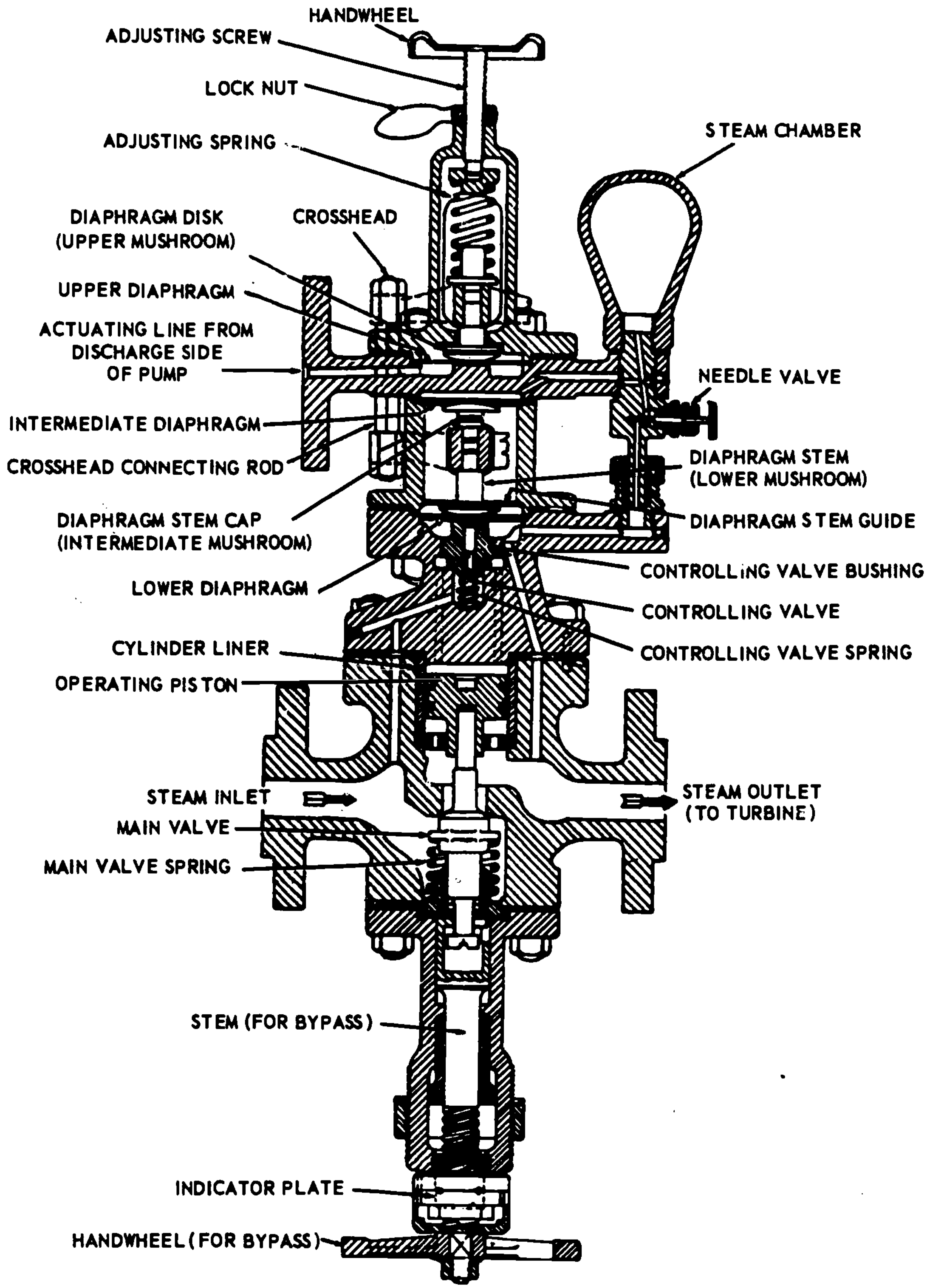
A Leslie constant-pressure pump governor for a main feed pump is shown in figure 15-18. The governors used on fuel oil service pumps, lube oil service pumps, fire and flushing pumps, and various other pumps are almost identical. The chief difference between governors used for different services is in the size of the upper diaphragm. A governor used for a pump that operates with a high discharge pressure has a smaller upper diaphragm than one used for a pump that operates with a low discharge pressure.

Two opposing forces are involved in the operation of a constant-pressure pump governor. Fluid from the pump discharge, at discharge pressure, is led through an actuating line to the space below the upper diaphragm. The pump discharge pressure exerts an UPWARD force on the upper diaphragm. Opposing this, an adjusting spring exerts a DOWNWARD force on the upper diaphragm.

When the downward force of the adjusting spring is greater than the upward force of the pump discharge pressure, the spring forces both the upper diaphragm and the upper crosshead downward. A pair of connecting rods connects the upper crosshead rigidly to the lower crosshead, so the entire assembly of upper and lower crossheads moves together. When the crosshead assembly moves downward, it pushes the lower mushroom and the lower diaphragm downward. The lower diaphragm is in contact with the controlling valve. When the lower diaphragm is moved downward, the controlling valve is forced down and open.

The controlling valve is supplied with a small amount of steam through a port from the inlet side of the governor. When the controlling valve is open, steam passes to the top of the operating piston. The steam pressure acts on the top of the operating piston, forcing the piston down and opening the main valve. The extent to which the main valve is opened controls the amount of steam admitted to the driving turbine. Increasing the opening of the main valve therefore increases the supply of steam to the turbine and so increases the speed of the turbine.

The increased speed of the turbine is reflected in an increased discharge pressure from



38.90

Figure 15-18.—Constant-pressure governor for main feed pump.

the pump. This pressure is exerted against the underside of the upper diaphragm. When the pump discharge pressure has increased to the point that the upward force acting on the underside of the upper diaphragm is greater than the downward force exerted by the adjusting spring, the upper diaphragm is moved upward. This action allows a spring to start closing the controlling valve which in turn allows the main valve spring to start closing the main valve against the now-reduced pressure on the operating piston. When the main valve starts to close, the steam supply to the turbine is reduced, the speed of the turbine is reduced, and the pump discharge pressure is reduced.

At first glance, it might seem that the controlling valve and the main valve would be constantly opening and closing and the pump discharge pressure would be continually varying over a wide range. This does not happen, however, because the governor is designed to prevent excessive opening or closing of the controlling valve. An intermediate diaphragm bears against an intermediate mushroom which in turn bears against the top of the lower crosshead. Steam is led from the governor outlet to the bottom of the lower diaphragm and also through a needle valve to the top of the intermediate diaphragm. A steam chamber provides a continuous supply of steam at the required pressure to the top of the intermediate diaphragm.

Any up or down movement of the crosshead assembly is therefore opposed by the force of the steam pressure acting on either the intermediate diaphragm or the lower diaphragm. The whole arrangement serves to prevent extreme reactions of the controlling valve in response to variations in pump discharge pressure.

Limiting the movement of the controlling valve in the manner just described reduces the amount of hunting the governor must do to find each new position. Under constant-load conditions, the controlling valve takes a position that causes the main valve to remain open by the required amount. A change in load conditions causes momentary hunting by the governor until it finds the new position required to maintain pump discharge pressure at the new load.

A pull-open device, consisting of a valve stem and a handwheel, is fitted to the bottom of the governor. Turning the handwheel to the open position draws the main valve open and allows full steam flow to the turbine. When the main valve is opened by means of this bypass arrangement, the turbine must be controlled manually. Under all normal operating conditions, the bypass remains closed and the pump discharge pressure is raised or lowered, as necessary, by increasing or decreasing the tension on the adjusting spring.

CONTROL AND MAIN VALVE.–If there is leakage in the generator through the control valve or its bushing, steam will flow to the top of the operating piston, opening the main valve, and holding it open, even though there is no tension on the adjusting spring. The main valve must be able to close off completely or else the governor cannot operate properly. The only remedy is to disassemble the governor and stop the steam leakage. In most instances, you must renew the control valve. If the leakage is through the bottom of the bushing and its seat, you must lap the seat. A cast iron lap is best for this type of work.

Rotate the lap through a small angle of rotation, lift it from the work occasionally, and move to a new position as the work progresses. This will ensure that the lap will slowly and gradually rotate around the entire seat circle. Do not bear down heavily on the handle of the lap. Replace the compound often, using only clean compound. If the lap should develop a groove or cut, redress the lap. Lapping should never be continued longer than necessary to remove all damaged areas.

When you are installing the control valve and its bushing, remember that the joint between the bottom of the bushing and its seat is a metal-to-metal contact. Install the bushing tightly, and when it is all the way down, tap the wrench lightly with a hammer, to ensure a steamtight joint.

When the controlling valve is installed, you must check the clearance between the top of the valve stem and the diaphragm. It is absolutely mandatory that this clearance be between .001 and .002 inch (fig. 15-19). If the clearance is less than .001 inch, the diaphragm will hold the control valve open, allowing steam to flow to the main valve at any time the throttle valve is open. If the clearance is more than .002 inch, the diaphragm will not fully open the control valve—which means that the main valve cannot open fully, and the unit cannot be brought up to full speed and capacity.

When the main valve seating area is damaged, it must be lapped in by the same process. ALWAYS lap in the main valve with the piston in the cylinder liner to ensure perfect centering.

If the damage to the seating surfaces is excessive, you must install new parts. Use only parts supplied by the manufacturer, if available.

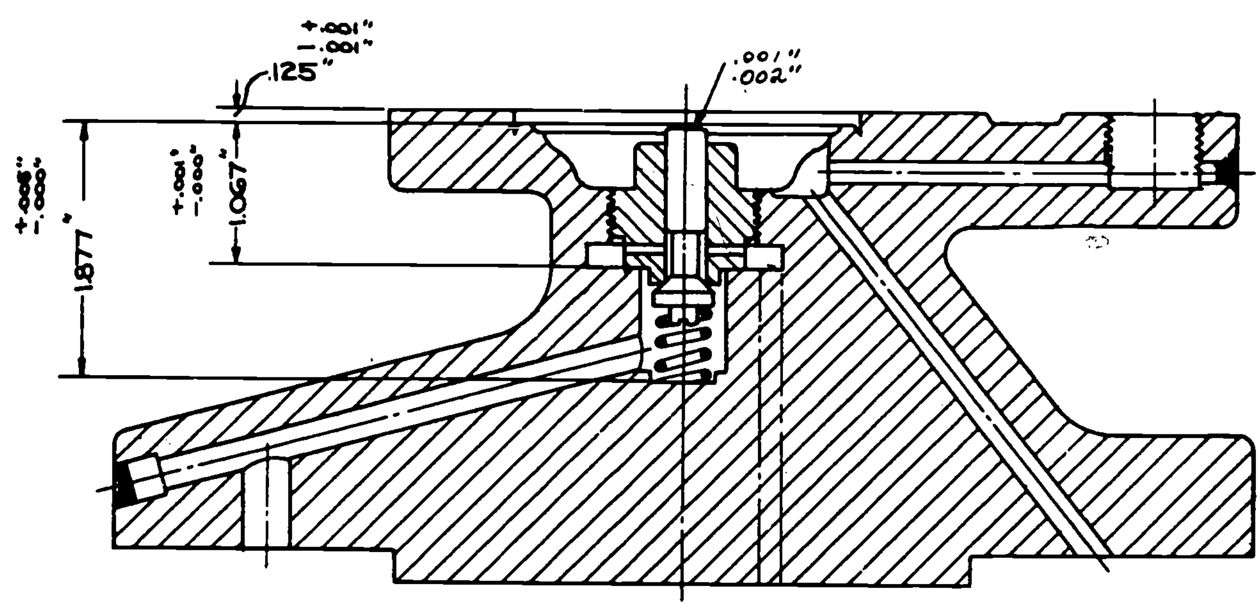
TOP CAP.—If the top flange of the top cap of the governor becomes damaged, you must be extremely careful machining it. Consult the manufacturer's technical manual for the correct clearances. (See fig. 15-19.)

All seating surfaces must be square with the axis of the control valve seat threads and must have the smoothest possible finish. Before you start the reassembly, be sure that all ports in the top cap and diaphragm chamber are free of dirt and other foreign matter. Check to ensure that the piston rings are free in their grooves. The cylinder liner must be smooth and free of grooves, pits, or rust.

When installing the cylinder liner, make certain that the top of the liner does not extend above the top of the valve body. The piston must work freely in the liner; if there is binding, the governor will not operate satisfactorily. Renew the controlling valve spring and the main valve spring if they are weak, broken or corroded, or if they have taken a permanent set. If necessary, renew all diaphragms; if you use the old diaphragms, install them in their original position; do not reverse.

Follow the instructions in the manufacturer's technical manual in reassembling the governor. All clearances must be as designed if the governor is to operate satisfactorily. You must check each moving part carefully to ensure freedom of movement.

When the governor is reassembled, test it as soon as possible so that you can make corrections, if necessary.



96.35

Figure 15-19.—Critical dimensions of Leslie top cap.

Double Seated Valves

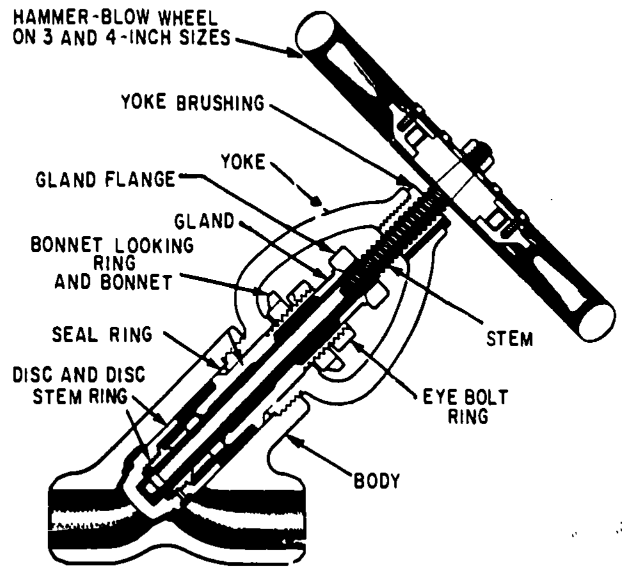
Depending on the extent of damage to the disk of a double seated valve, you can lap or weld-repair it and remachine it to fit the body. The normal seat angles remain the same as for globe valves and the spotting-in procedure will be the same. Most valve disks can be held on a spud or mounted on a mandrel and can be cut in the same manner as a globe valve. In this case as in the others, it is best to consult local quality assurance directives and local procedures in the repair of this type of valve. Also, in most cases the blueprints will show "ND" (no deviations) and must be closely adhered to, as far as type of weld and quality. In all cases shop LPO's should be able to provide the necessary information.

Duplex Strainer Plug Valves

The most common cause for repair to duplex strainers is scored or chipped O-ring grooves or scored or scratched liners. In some cases it may be necessary to perform a weld repair and then machine back to blueprint specifications on the plug cock. In the case of repair to the strainer body, you will usually hone it and in some cases you will use an oversized O-ring. The best repair method is to consult local type commander and quality assurance procedures to find out which is the best suited for your situation. Check with the shop's leading petty officer before undertaking any repair procedures.

Pressure Seal Bonnet Globe Valves

In many cases you may be required to repair pressure seal bonnet globe valves. This type of valve (fig. 15-20) is usually the welded bonnet type, and you will be involved inasfar as machining the bonnet seal area to specifications provided by either the applicable blueprint or the Hull Technician doing the welding. This basic type valve is used in steam systems; it is also commonly found in the nuclear systems in submarines and submarine tenders. This type of valve repair is also referred to as canopy seal



47.194

Figure 15-20.—1500-pound pressure seal bonnet globe valve.

valve. In some instances you may be required to work closely with the radiological control division since these valves are used in nuclear systems and must be closely monitored for radiation levels and possible contamination of equipment and tools used during the repair procedure. In most tenders the R-5 division has facilities to work on valves that require special handling. In these instances you would be required to provide the technical ability, and R-5 division personnel would do the monitoring.

Assembling High-Pressure Steam Valves

The bonnet joint of a high-pressure steam valve is always made up with a metallic or a flexible gasket and high-temperature-use alloy stud bolts and nuts. When you assemble such a valve, be sure that you use the correct kind of gasket and stud bolts. If you are the least bit doubtful of what you should use in a particular valve, ask your leading petty officer.

There are two ways to identify a high-temperature-use alloy stud bolt: (1) the thread runs the entire length of the body and one end of the bolt has a small center hole recess and

(2) the bolt will have either an "H" or an "A" stamped on one end.

High-temperature nuts have either an "H" or "A" stamped on the crown. If you do not see such an identification on a nut, do not use it on a high-pressure valve.

When assembling a valve, use antiseize compound on the stud bolt threads, and always be sure to back the disk away from the seat before tightening any of the bonnet nuts. In setting up on bonnet flange nuts, alternate approximately 180° and 90° from the starting point until you have all of them set up evenly and fairly tight. For final all-round setup on the nuts, use a strain gage to measure for correct tightening tension or a micrometer to measure elongation of the studs to compute the tension. Your leading petty officer can give you practical instruction on correct tension for different sizes of stud bolts.

Testing Valves

After a valve has been overhauled in the shop, it is standard practice to test it under hydrostatic pressure to prove the tightness of the seat and the bonnet joint. Figure 15-21 shows a Machinery Repairman in the process of applying a hydrostatic test to a high-pressure steam valve. In this particular setup, the valve is held on a thick rubber gasket by U-clamps and water delivered under pressure from a hydraulic test pump will be led into the bottom of the valve from a connection underneath the test stand.

28.263

Figure 15-21.—Applying hydrostatic test to a high-pressure steam valve.

After applying a test pressure to the lower part of the valve, the valve will be placed with the other flange down to test the bonnet joint.

When testing valves hydrostatically, be sure to use the specified test pressure. Too low a pressure will not prove the tightness of the valve and too high a pressure may cause damage to the valve.

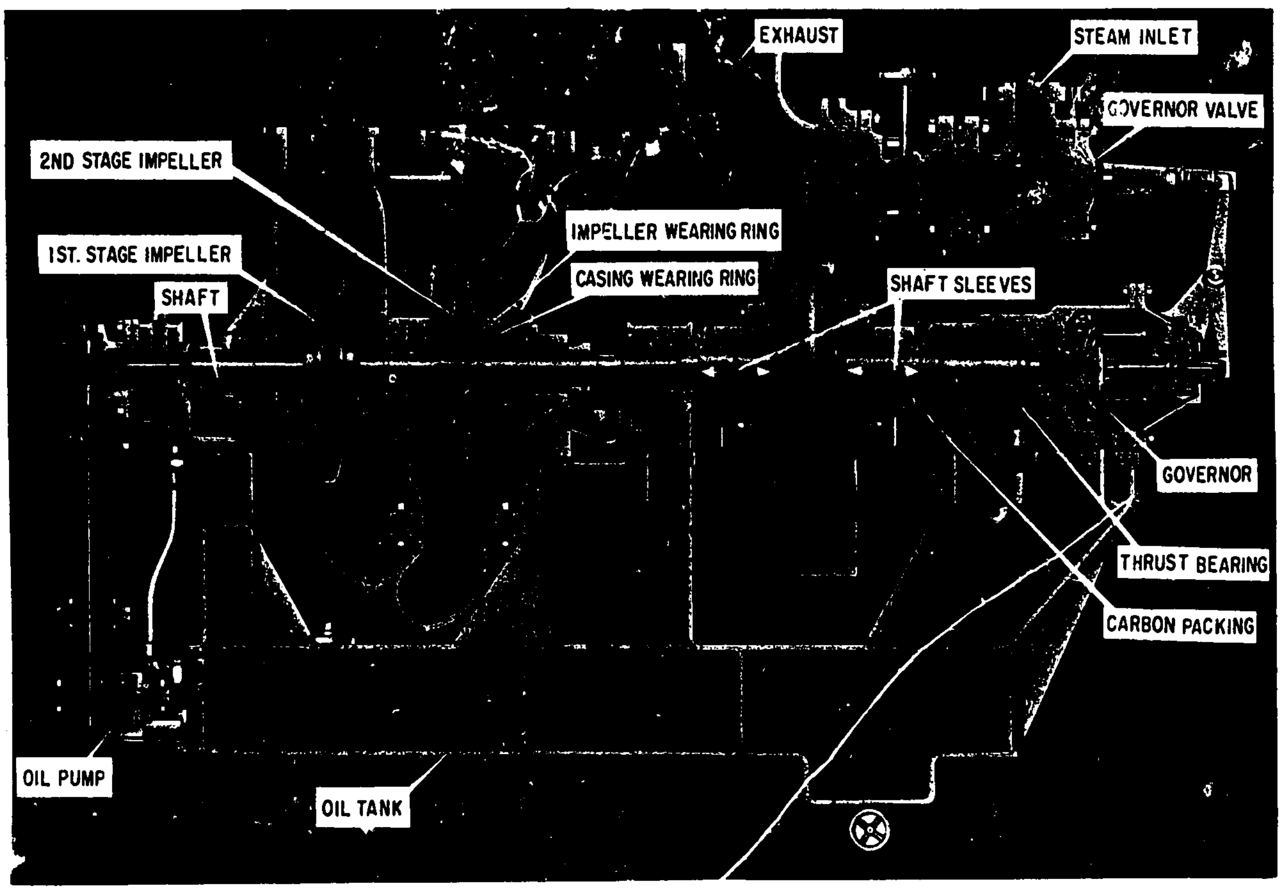
REPAIRING PUMPS

A description of the common types and uses of pumps on-board ship is provided in *Fireman*, NAVEDTRA 10520-E. The following discussion is limited to repair of centrifugal pumps because these pumps are the ones that a Machinery Repairman will usually be required to repair.

Figure 15-22 is a sketch of the internal parts of a centrifugal pump. Look at the arrangement of the impeller, casing wearing rings, impeller wearing rings, shaft, and shaft sleeves in particular.

In a centrifugal pump, the portion of the shaft in the way of the packing gland and the casing-impeller sealing areas are subject to wear in operation. They must be renewed from time to time to maintain the operating efficiency of the pump.

To prevent having to renew the entire shaft solely because of wear in the packing gland area, shafts in centrifugal pumps are often provided



38.109

Figure 15-22.—Two-stage main feed pump.

with tightly fitting renewable sleeves. To offset the need for renewing or making extensive repairs to the casing and impeller, these two parts also have renewable wearing surfaces, called the casing wearing rings and impeller wearing rings. You can see the arrangement clearly in figure 15-23.

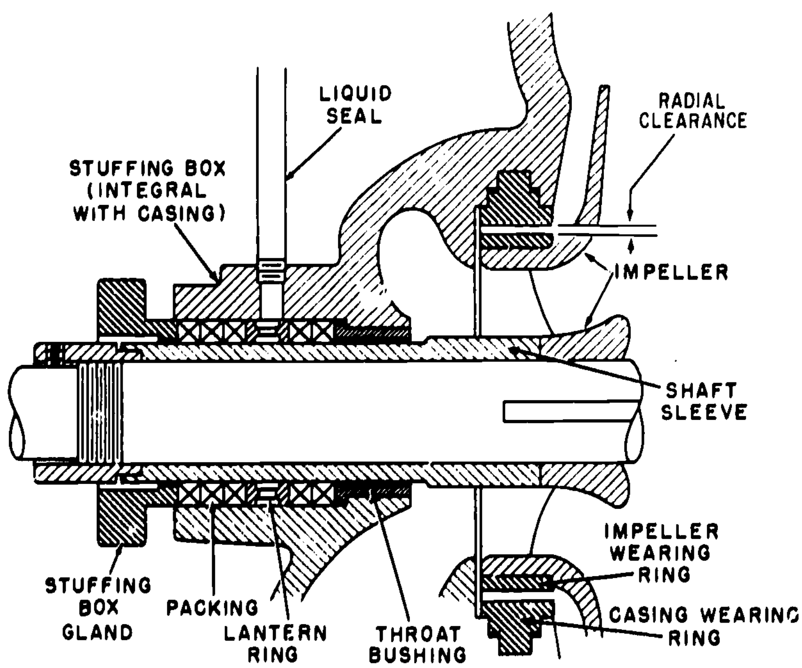
When it is necessary to renew these parts, the rotor assembly, consisting of the pump shaft, the impeller and its wearing ring, and the casing rings, is usually brought into the shop. The method of replacing these parts is described in the following paragraphs.

The repair parts generally are available from the ship's allowance, but often you may need to turn them out in the shop. Before you can proceed with these repairs, consult the manufacturer's technical manual and the applicable blueprints to get the correct information on vital clearances and other data.

In some pumps, the shaft sleeve is pressed onto the shaft tightly with a hydraulic press, and you must machine off the old sleeve in a lathe before you can install a new one. On centrifugal pumps, the shaft sleeve is a snug slip-on fit, butted up against a shoulder on the shaft and held securely in place with a nut. In the latter pumps, the sleeve-shaft shoulder joint is usually made up with a hard fiber washer to prevent leakage of liquid through the joint and out of the pump between sleeve and shaft.

The impeller wearing ring is usually lightly press-fitted to the hub of the impeller and keyed in with headless screws (also referred to as "Dutch keyed"). To remove the worn ring, withdraw the headless screws or drill them out and then machine the ring off in a lathe.

The amount of diametrical running clearances between the casing rings and impeller rings affects the efficiency of a centrifugal pump. Too much clearance will let an excessive amount of liquid leak back from the discharge side to the suction side of the pump. Insufficient clearance will cause the pump to "freeze." Before you install a new wearing ring on the impeller, measure the outside diameter of the impeller hub, the inside and outside diameters of the impeller wearing ring, and the inside



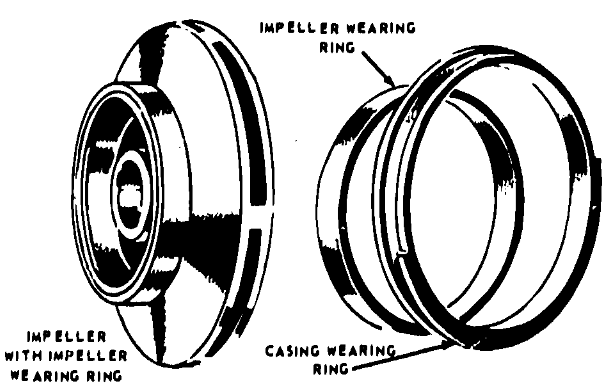
28.35

Figure 15-23.—Stuffing box on centrifugal pump.

diameter of the casing ring. (See fig. 15-24.) If the measurements do not agree with the fit and clearance data you have on hand, ask your leading petty officer for instructions before proceeding any further. Sometimes it is necessary to take a light cut on the inside diameter of the impeller ring to get its correct press fit on the impeller hub. The difference between the outside diameter of the impeller wearing ring and the inside diameter of the casing wearing ring is the diametrical running clearance between the ring. If this clearance is too small, correct it by taking a cut on either the outside diameter of the impeller ring or the inside diameter of the casing ring. Another thing to check is the concentricity of the two rings; if they do not run true, you must machine their mating surfaces so that they do run true, bearing in mind, of course, to keep the specified diametrical clearance.

When a pump like the one shown in figure 15-22 needs repairs, usually only the shaft assembly and casing wearing rings are brought to the shop. To renew the wearing rings and resurface the packing sleeves of the pump shown in figure 15-22, take the following steps:

1. Clamp the casing wearing ring on a faceplate and align the circumference of the ring concentrically with the axis of the lathe spindle.



76.281X
Figure 15-24.—Impeller, impeller wearing ring, and casing wearing ring for a centrifugal pump.

(The casing rings may be chucked in a 4-jaw chuck but there is danger of distorting the ring if this is done.)

2. Take a light cut on the inside diameter of the casing ring to clean up the surface. Do this to all casing rings.

3. Mount the shaft assembly between centers or in a chuck and align with the lathe axis.

4. Machine away the impeller wearing rings. Be careful not to cut into the impeller.

5. Take a light cut on the packing sleeves to clean up the surfaces.

6. Remove the shaft assembly from the lathe.

7. Make the impeller rings. The size of the inside diameter of the impeller rings should provide a press fit on the impeller; the outside diameter should be slightly larger than the inside diameter of the casing rings.

8. Press the impeller rings on the impeller and lock in place with headless screws, if so stated on blueprint.

9. Mount the shaft assembly back in the lathe and machine the diameter of the impeller rings to provide the proper clearance between impeller rings and casing rings. Blueprints and/or technical manuals will list the desired clearance in one of two ways: (1) If diametrical clearance is used, that is the total amount of clearance required; (2) if it is listed as radial clearance, the amount desired has to be doubled to get diametrical clearance.

MACHINE SHOP MAINTENANCE

The ship in which you serve and the shop in which you work were designed to accomplish a particular mission or job. As an MR3 or MR2, you will be expected to assist in the proper maintenance and preservation of the machines and spaces you use. Generally, you can give a workshop one good look and tell whether it is efficient and well run. The Ship's Maintenance Material Management (3-M) System has been implemented by the Navy as an answer to the ever present problem of maintaining a high degree of operational readiness. A thorough study of *Military Requirements for Petty Officers 3 & 2*, NavPers 10056-D, will give you all the information you need on the 3-M System.

Although the 3-M System is designed to improve the degree of readiness, its effectiveness and reliability are dependent upon you, the individual. The accuracy with which you perform your work, along with neat and complete recording of required data on the prescribed forms is one of the keys to the degree of readiness of your ship. Remember PREVENTIVE MAINTENANCE (scheduled checks) will lead to less CORRECTIVE MAINTENANCE (repair of equipment). Control over rust and corrosion will be a major problem. Equipment used often is not likely to "freeze up," but machinery which is seldom used may fail to operate at a crucial moment. It is a good policy to check and operate all shop machinery immediately after the weekly lubrication.

There will be rust film trouble in all climates, but it will occur more frequently in the tropics because of humidity (moisture). All bare metal surfaces should be kept clean and bright, and a light coat of machine oil should be applied to protect them. The rust prevention program should be a part of your daily cleanup routine. Using an approved rust preventive compound will help prevent rusting of decks and bare metal surfaces and machinery parts.

It is sometimes said that a machine tool operator can be judged by the condition of his tools, machines, and spaces. Good maintenance practices will save you many hours of extra work. Some good precautions for the maintenance of machinery are listed below:

- Before applying power to a machine, see that the machine is ready for starting. For example, move the carriage of a lathe by the hand feed to ensure that all locking devices have been released.

- Do not lay work or handtools on the ways of a machine.

- Avoid scoring the platen of a planer, drilling holes in the table of a drill press, or gouging the vise or footstock of a milling machine.

- Do not use the table of any machine for a workbench.

- When using a toolpost grinder on a lathe, cover the ways and other finished surfaces to protect them against grit.

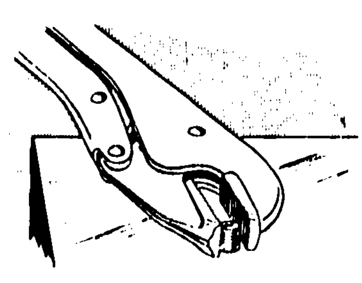
- See that pneumatic power-driven handtools are lubricated after each 8 hours of operation or more often where found necessary.

- Before taking an electric power-driven handtool from the toolroom, examine it carefully for mechanical and electrical defects and ensure that the electrical safety tag is up-to-date before using.

- When securing for sea, take all precautions to ensure that machinery or components will not sway or shift with the motion of the ship. The precautions should include the following:

    a. In securing top-heavy equipment such as a radial drill press arm, lower it to rest on the table or base of the machine and then make sure that it is locked and blocked securely.

    b. Secure chain falls, trolleys, overhead cranes, and other suspended equipment, such as counterweights on boring mills and drill presses.



1.21C

Figure 15-25.—Removing a broken stud with vise-grip pliers.

c. Secure tailstocks of lathes.

d. Secure spindles of horizontal boring mills.

e. Protect and secure tools stowed in cabinets or drawers. Secure drawers and cabinet doors.

REMOVING BROKEN BOLTS AND STUDS

When you must remove a broken bolt or stud, flood the part being worked on with plenty of penetrating oil or oil of wintergreen. Time permitting, soak the area for several hours or overnight. A week's soaking may loosen a bolt which would otherwise have to be drilled out.

If enough of the broken piece protrudes, take hold of it with vise-grip pliers, as shown in figure 15-25, and carefully try to ease it out. If you cannot turn the bolt, further soaking with penetrating oil may help. Or try removing the pliers and jarring the bolt with light hammer blows on the top and around the sides. This may loosen the threads so that you can remove the bolt with the vise-grip pliers.

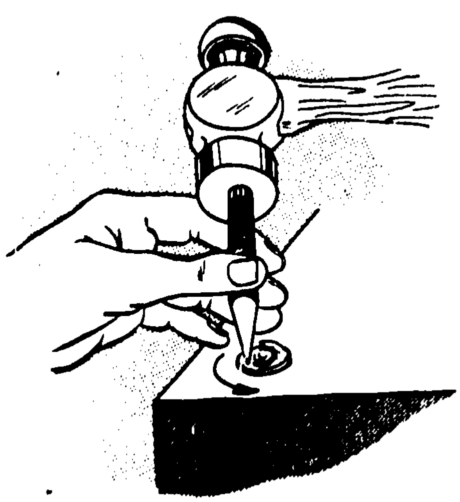
If a bolt has been broken off flush with the surface as shown in figure 15-26, it is sometimes possible to back it out with light blows of a prick punch or center punch. However, if the bolt was broken due to rusting, this method will not remove it. If you cannot remove it by carefully punching first on one side and then the other, use a screw and bolt extractor. (See fig. 15-27B.)

When using this extractor, file the broken portion of the bolt to provide a smooth surface at the center for a punch mark, if possible. Then carefully center punch the exact center of the bolt. (See fig. 15-27A.)

Refer to table 15-1 to select the proper drill to use according to the size of the broken bolt that you are trying to remove. If possible, drill through the entire length of the broken bolt. Then carefully work some penetrating oil through the hole so that it fills the cavity beneath the bolt and has a chance to work its way upward from the bottom of the bolt. The more time you let the penetrating oil work from both ends of the broken bolt, the better are your chances of removing it.

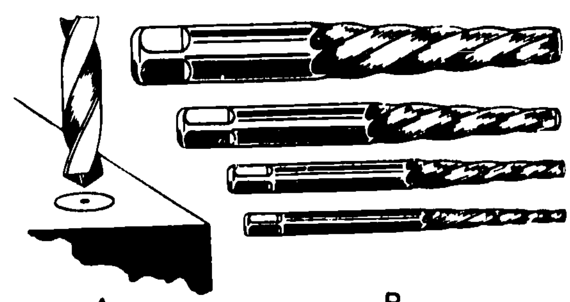
In drilling a hole in a stud that has broken off below the surface of the piece which it was holding (fig. 15-28A), use a drill guide to center the drill. This method may be preferred rather than a center punch mark.

After you have drilled the hole and added penetrating oil and let it soak, put the spiral end of the screw and bolt extractor into the hole. Set it firmly with a few light hammer blows and secure the tap wrench as shown in figure 15-28B. Carefully try to back the broken bolt



44.188

Figure 15-26.—Removing a broken bolt with a prick punch.



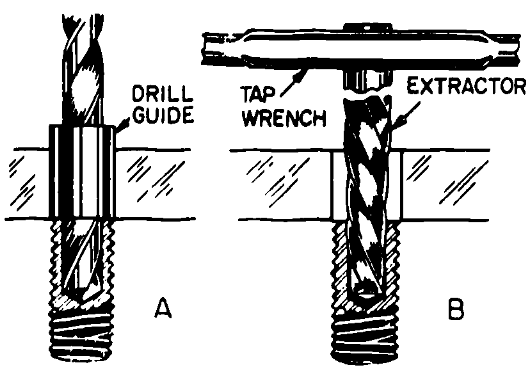
44.20BB.2

Figure 15-27.—Screw and bolt extractors for removing broken studs.

Table 15-1.—Chart for Screw and Bolt Extractors

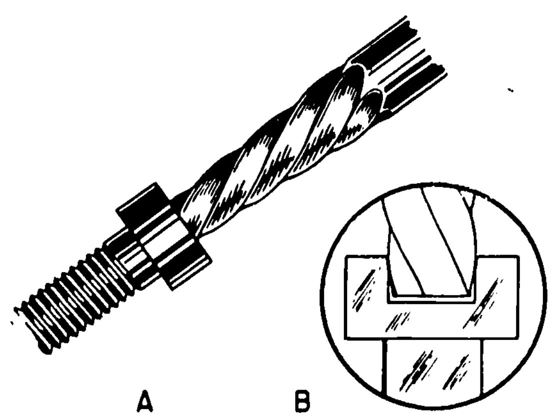
Extractor		Used For—		Use Drill Size Dia., Inches
Size No.	Overall Length, Inches	Nominal Screw And Bolt Size, Inches	Nominal Pipe Size, Inches	
1	2	3/16 - 1/4	--------	5/64
2	2 3/8	1/4 - 5/16	--------	7/64
3	2 11/16	5/16 - 7/16	--------	5/32
4	3	7/16 - 9/16	--------	1/4
5	3 3/8	9/16 - 3/4	1/4	17/64
6	3 3/4	3/4 - 1	3/8	13/32
7	4 1/8	1 - 1 3/8	1/2	17/32
8	4 3/8	1 3/8 - 1 3/4	3/4	13/16
9	4 5/8	1 3/4 - 2 1/8	1	1 1/16
10	5	2 1/8 - 2 1/2	1 1/4	1 5/16
11	5 5/8	2 1/2 - 3	1 1/2	1 9/16
12	6 1/4	3 - 3 1/2	2	1 15/16

44.220



44.20CC

Figure 15-28.—Removing a stud broken off below the surface.



44.20DD

Figure 15-29.—Removing an Allen head capscrew with a bolt extractor.

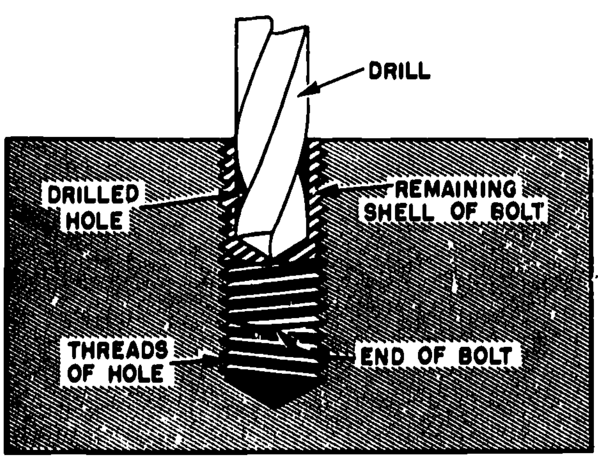
out of the hole. Turn the extractor counterclockwise. (This type of extractor is designed for right-hand threads only.)

Sometimes you can use a screw and bolt extractor to remove an Allen head capscrew when the socket has been stripped by the Allen wrench. (See fig. 15-29.) When attempting this removal, carefully grind off the end of the extractor so that it will not bottom before the spiral has had a chance to take hold. Figure 15-29 shows this end clearance. In doing this grinding operation, be very careful to keep the temperature of the extractor low enough so that you can handle the tip with your bare hands. If the hardness is drawn from the tip of the extractor by overheating during the grinding, the extractor will not take hold.

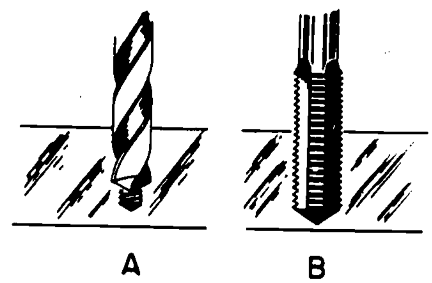
REMOVING A BROKEN BOLT AND RETAPPING THE HOLE

To remove a broken bolt and retap the hole, file the bolt smooth, if necessary, and centerpunch it for drilling. Then select a twist drill which is a little less than the tap-drill size for the particular bolt that has been broken. As shown in figure 15-30, this drill will just about but not quite touch the crests of the threads in the threaded hole or the roots of the threads on the threaded bolt. Carefully start drilling at the center punch mark, crowding the drill one way or the other as necessary so that the hole will be drilled in the exact center of the bolt.

The drill in figure 15-30 has almost drilled the remaining part of the bolt away and will eventually break through the bottom of the bolt. When this happens, all that will remain of the bolt will be a threaded shell. With a prick punch or other suitable tool, chip out and remove the first two or three threads, if possible, at the top of the shell. Then carefully start a tapered tap into these several clean threads and continue tapping until the shell has been cut away and the original threads have been restored.



44.20EE
Figure 15-30.—Removing a broken bolt and retapping hole to same size.

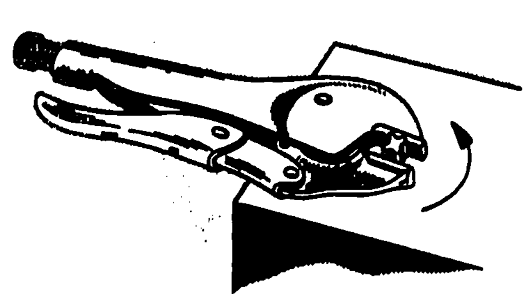


44.189
Figure 15-31.—Removing broken bolt and retapping hole to larger size.

In cases where the identical size of capscrew or bolt is not necessary as a replacement, center punch and drill out the old bolt with a drill larger than the broken bolt, as shown in figure 15-31A. Tap the hole first, and then finish it with a bottoming tap as shown in figure 15-31. Replace with a larger size capscrew or stud.

REMOVING A BROKEN TAP FROM A HOLE

To remove a broken tap from a hole, generously apply penetrating oil to the tap, working it down through the four flutes into the hole. Then, if possible, grasp the tap across the flats with vise-grip pliers. This operation is shown in figure 15-32. Carefully ease the tap out of the hole, adding penetrating oil as necessary.

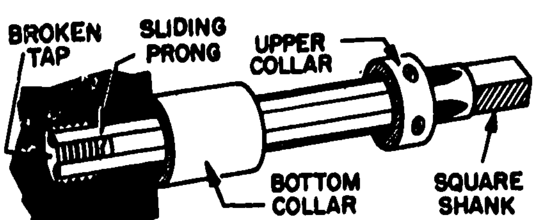


1.21D
Figure 15-32.—Removing a broken tap with vise-grip pliers.

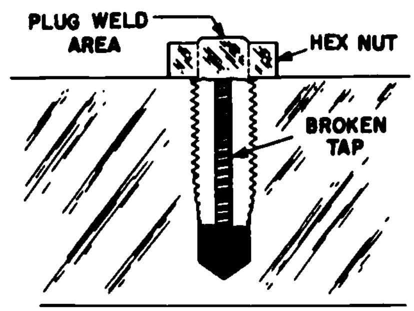
If the tap has broken off at the surface of the work or slightly below the surface of the work, the tap extractor shown in figure 15-33 may remove it. Again, apply a liberal amount of penetrating oil to the broken tap. Place the tap extractor over the broken tap and lower the upper collar to insert the four sliding prongs down into the four flutes of the tap. Then slide the bottom collar down to the surface of the work so that it will hold the prongs tightly against the body of the extractor. Tighten the tap wrench on the square shank of the extractor and carefully work the extractor back and forth to loosen the tap. You may need to remove the extractor and strike a few sharp blows with a small hammer and pin punch to jar the tap loose. Then reinsert the tap remover and carefully try to back the tap out of the hole.

Each size of tap will require its own size of tap extractor. Tap extractors come in the following sizes: 1/4, 5/16, 3/8, 7/16, 1/2, 9/16, 5/8, 3/4, 7/8 and 1 inch.

When a tap extractor will not remove a broken tap, you may be able to do so by the following method: Place a hex nut over the tap (fig. 15-34), and weld the nut to the tap. Be sure to choose a nut with a hole somewhat smaller than the tap diameter to reduce the possibility of welding the nut and the tap to the job itself. Allow the weld to cool before trying to remove the tap. When the nut, tap, and job have come to room temperature, it is often helpful to heat quickly the immediate area around the hole with an oxyacetylene torch. This quick heating expands the adjacent metal of the work after which you may be able to remove the tap more easily. If the heating is too slow, the tap will expand with the adjacent metal of the work and there will be no loosening effect.



44.191
Figure 15-33.—Removing a broken tap with a tap extractor.



44.192
Figure 15-34.—Using a plug weld to remove a broken t

MAKING PISTON RINGS

To make a cast iron piston ring, select a billet of sufficient size to permit removal of surface defects. For example, in making a ring that has a 10-inch outside diameter and a 9-inch inside diameter, use a billet with an outside diameter of 11 inches and an inside diameter of 8 inches. A billet this size has a wall thickness of 1 1/2 inches and will allow removal of 1/2 inch of metal from the inside surface and 1/2 inch of metal from the outside surface. To make the ring, proceed as follows:

1. Mount the billet in a chuck on the lathe.

2. Face the end.

3. Rough bore and then finish bore to the inside diameter of the ring. Bore a sufficient distance into the billet to make the desired width of the ring or rings.

4. Rough turn the outside of the billet to a diameter that is 0.010 inch larger per inch than the bore of the cylinder into which the ring is to be fitted. For example, for a 10-inch cylinder bore, the rough turn diameter would be 10.100 inches.

5. Cut off the ring to width with a parting tool.

6. Split the ring with a 45° cut, using a hacksaw. Place a piece of chart paper in the cut and then wrap a piece of wire around the circumference of the ring and draw it up until the ends butt up snugly.

7. Mount the ring on a faceplate to finish turn to exact cylinder bore size. Place faceplate clamps on the inside of the ring to prevent interfering with the operation. Place a piece of paper between the ring and the surface of the faceplate to keep the ring from slipping and also to keep the tool from cutting into the faceplate when turning. When you have centered the ring on the faceplate and taken up the clamps securely, remove the binding wire, and proceed with the finish turning operation.

SPRING WINDING

The methods and tools used for winding or coiling springs vary greatly in form and also in regard to productive capacity. The method used ordinarily depends upon the number of springs required and to some extent upon their form. When a comparatively small number of springs are needed in connection with repair work, etc., it is common practice to wind them in a lathe; whereas when springs are manufactured in large quantities, special machines are used.

Springs are often made with an "initial tension" which causes the coils to be drawn tightly together. This result is secured by twisting the wire as the spring is wound. A common example of a spring of this type is the ordinary screen door spring. When in a static condition (as before being installed on a door), such springs will not begin to deflect as soon as the load is applied, it being necessary to first overcome the initial tension already in the spring.

TABLES FOR SPRING WINDING

When springs are to be wound by using a lathe instead of a spring-coiling machine, the lathe is geared in the same manner as for screw cutting. Table 15-2 indicates which gearing should be used. The figures in the body of the table give the number of threads per inch for which the lathe should be geared to wind coil springs of a given wire gage. The figures in the column headed "A" are for closewound tension springs, while the figures in the columns headed "B" are for compression springs. Assume as an example, that you must wind a compression spring of No. 10 Brown and Sharpe gage wire. From the table, you will note that this spring should have four and one-half coils per inch, or in other words that the lathe should be geared the same as for cutting four and one-half threads per inch.

Table 15-3 gives data for winding piano wire tension springs and may be best explained by an example. Assume that you must wind three different springs; the first to be wound from 0.035-inch wire to fit in an 11/16-inch hole, the second to be wound from 0.040-inch wire to fit a 3/8-inch hole, and the third to be wound from 0.060-inch wire to be a sliding fit on a 1/2-inch diameter shaft. The table shows the proper sizes of mandrels for winding to be as follows: for the first spring 0.562 inch; for the second spring, 0.250 inch; and for the third spring, 0.437 inch. In the latter case, 0.011 inch is allowed for play between spring and shaft. The wire sizes given in the table conform to the English music wire gage.

In all cases when the mandrel diameter is larger than 3/8 inch, the mandrel is mounted in a lathe chuck. Mandrels less than 3/8 inch in diameter are mounted in a drill chuck. In fastening the wire in a lathe chuck, one jaw is usually loosened. When the mandrel is driven by a drill chuck, place the wire between the jaws and the mandrel. If a long spring is required, use a mandrel of corresponding length, which is ground to an angle of 60° at the end to fit into a female dead center for support. Place the wire in a bench lathe boring tool holder or a V-holder in the toolpost. Place a piece of brass about 1/8 inch by 1/2 inch by 3 inches between the wire and the toolpost screw. File a V-shaped groove in the brass to hold the wire in place. File the groove in the lengthwise direction of the brass

Table 15-2.—Gearing Lathe for Winding Wire Coil Springs

Tension Spring — I   Compression Spring -- II

Number of Wire Gage	Brown & Sharpe		Birmingham or Stub's		Washburn & Moen Mfg. Co.		Trenton Iron Co.		Prentiss		OldEnglish Brass Manufacturers'	
	I	II	I	II	I	II	I	II	I	II	I	II
000000	..	...	...	...	2	1	...	...	...	...	...	...
00000	..	...	...	...	2¼	1⅛	2	1	...	...	...	...
0000	2	1	2	1	2½	1¼	2½	1¼	...	...	...	...
000	2¼	1⅛	2¼	1⅛	2¾	1⅜	2¾	1⅜	2¾	1⅜	...	...
00	2¾	1⅜	2½	1¼	3	1½	3	1½	3	1½	...	...
0	3	1½	2⅞	1 7/16	3¼	1⅝	3	1½	3¼	1⅝	...	...
1	3¼	1⅝	3¼	1⅝	3½	1¾	3½	1¾	3¼	1⅝	...	...
2	3½	1¾	3½	1¾	3½	1¾	3½	1¾	3½	1¾	...	...
3	4	2	3½	1¾	4	2	4	2	4	2	...	...
4	4½	2¼	4	2	4	2	4	2	4	2	...	...
5	5½	2¾	4½	2¼	4½	2¼	4½	2¼	4½	2¼	...	...
6	6	3	4½	2¼	5	2½	5	2½	5	2½	...	...
7	6½	3¼	5½	2¾	5½	2¾	5½	2¾	5½	2¾	...	...
8	7	3½	6	3	6	3	6	3	6	3	...	...
9	8	4	6½	3¼	6½	3¼	6½	3¼	6½	3¼	...	...
10	9	4½	7	3½	7	3½	7	3½	7	3½	...	...
11	11	5½	8	4	8	4	8	4	8	4	...	...
12	12	6	9	4½	9	½	9	4½	9	4½	...	...
13	14	7	10	5	10	[illegible]	10	5	10	5	...	...
14	14	7	12	6	12	6	12	6	12	6	12	6
15	16	8	13	6½	13	6½	14	7	13	6½	13	6½
16	18	9	14	7	14	7	16	8	14	7	14	7
17	22	11	16	8	16	8	18	9	16	8	16	8
18	24	12	20	10	20	10	22	11	20	10	20	10
19	28	14	23	11½	23	11½	24	12	23	11½	24	12
20	28	14	28	14	28	14	28	14	28	14	28	14
21	32	16	28	14	28	14	32	16	28	14	28	14
22	36	18	32	16	32	16	32	16	32	16	32	16
23	44	22	40	20	40	20	40	20	36	18	36	18
24	48	24	44	22	40	20	44	22	40	20	40	20
25	56	28	48	24	48	24	48	24	46	23	40	20
26	56	28	52	26	52	26	52	26	48	24	48	24
27	64	32	56	28	56	28	56	28	52	26	52	26
28	72	36	64	32	56	28	56	28	56	28	56	28
29	88	44	72	36	64	32	64	32	56	28	64	32
30	96	48	80	40	64	32	64	32	64	32	72	36
31	112	56	96	48	72	36	72	36	64	32	80	40
32	..	...	104	52	72	36	80	40	72	36	88	44
33	..	...	112	56	88	44	88	44	72	36	92	46
34	..	...	...	...	96	48	96	48	80	40	104	52
35	..	...	...	...	104	52	104	52	88	44	104	52

Table 15-3.–Data for Winding Piano Wire Tension Springs

Diam. of Mandrel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Number of Piano Wire	Diam. of Piano Wire, Inches	Diam. of Mandrel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Number of Piano Wire	Diam. of Piano Wire, Inches
0.125	0.130	0.150	1	0.0098	0.187	0.209	0.258	10	0.0245
0.187	0.192	0.212	1	0.0098	0.250	0.272	0.321	10	0.0245
0.250	0.255	0.275	1	0.0098	0.312	0.336	0.385	10	0.0245
0.312	0.318	0.338	1	0.0098	0.375	0.401	0.450	10	0.0245
0.375	0.382	0.402	1	0.0098	0.437	0.465	0.514	10	0.0245
0.125	0.130	0.151	2	0.0105	0.500	0.533	0.582	10	0.0245
0.187	0.192	0.213	2	0.0105	0.562	0.600	0.649	10	0.0245
0.250	0.255	0.276	2	0.0105	0.625	0.665	0.714	10	0.0245
0.312	0.318	0.339	2	0.0105	0.187	0.212	0.266	11	0.0270
0.375	0.382	0.403	2	0.0105	0.250	0.277	0.331	11	0.0270
0.125	0.130	0.152	3	0.0115	0.312	0.340	0.394	11	0.0270
0.187	0.193	0.215	3	0.0115	0.375	0.406	0.460	11	0.0270
0.250	0.256	0.278	3	0.0115	0.437	0.470	0.524	11	0.0270
0.312	0.320	0.342	3	0.0115	0.500	0.535	0.589	11	0.0270
0.375	0.382	0.404	3	0.0115	0.562	0.600	0.654	11	0.0270
0.125	0.135	0.160	4	0.0125	0.625	0.665	0.719	11	[illegible]
0.187	0.197	0.222	4	0.0125	0.187	0.212	0.269	12	[illegible]
0.250	0.260	0.285	4	0.0125	0.250	0.279	0.336	12	0.0285
0.312	0.322	0.347	4	0.0125	0.312	0.342	0.399	12	0.0285
0.375	0.385	0.410	4	0.0125	0.375	0.408	0.465	12	0.0285
0.125	0.135	0.164	5	0.0145	0.437	0.472	0.529	12	0.0285
0.187	0.198	0.227	5	0.0145	0.500	0.537	0.594	12	0.0285
0.250	0.261	0.290	5	0.0145	0.562	0.602	0.659	12	0.0285
0.312	0.324	0.353	5	0.0145	0.625	0.667	0.724	12	0.0285
0.375	0.389	0.418	5	0.0145	0.187	0.217	0.278	13	0.0305
0.125	0.135	0.165	6	0.0150	0.250	0.282	0.343	13	0.0305
0.187	0.198	0.228	6	0.0150	0.312	0.346	0.407	13	0.0305
0.250	0.262	0.292	6	0.0150	0.375	0.411	0.472	13	0.0305
0.312	0.325	0.355	6	0.0150	0.437	0.475	0.536	13	0.0305
0.375	0.390	0.420	6	0.0150	0.500	0.540	0.601	13	0.0305
0.125	0.137	0.172	7	0.0175	0.562	0.604	0.665	13	0.0305
0.187	0.201	0.236	7	0.0175	0.625	0.670	0.731	13	0.0305
0.250	0.266	0.301	7	0.0175	[illegible]	0.284	[illegible]	14	0.0320
0.312	0.330	0.365	7	0.0175	0.312	0.348	[illegible]	14	0.0320
0.375	0.395	0.430	7	0.0175	0.375	0.414	[illegible]	14	0.0320
0.125	0.138	0.176	8	0.0190	0.437	0.478	0.543	14	0.0320
0.187	0.202	0.240	8	0.0190	0.500	0.545	[illegible]	14	0.0320
0.250	0.266	0.304	8	0.0190	0.562	0.609	0.673	14	0.0320
0.312	0.330	0.368	8	0.0190	0.625	0.677	0.741	14	0.0320
0.375	0.396	0.434	8	0.0190	0.250	0.284	0.354	15	0.0350
0.125	0.145	0.189	9	0.0220	0.312	0.350	[illegible]	[illegible]	0.0350
0.187	0.209	0.253	9	0.0220	0.375	0.417	[illegible]	[illegible]	[illegible]
0.250	0.271	0.315	9	0.0220	0.437	0.480	[illegible]	[illegible]	[illegible]
0.312	0.335	0.379	9	0.0220	0.500	0.547	[illegible]	[illegible]	0.0350
0.375	0.400	0.444	9	0.0220	0.562	0.611	[illegible]	[illegible]	[illegible]

Table 15-3.—Data for Winding Piano Wire Tension Springs—Continued

Diam. of Mandrel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Number of Piano Wire	Diam. of Piano Wire, Inches	Diam. of Mandrel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Number of Piano Wire	Diam. of Piano Wire, Inches
0.250	0.290	0.362	16	0.0360	0.312	0.369	0.467	23	0.0490
0.312	0.355	0.427	16	0.0360	0.375	0.436	0.534	23	0.0490
0.375	0.420	0.492	16	0.0360	0.437	0.500	0.598	23	0.0490
0.437	0.483	0.555	16	0.0360	0.500	0.565	0.663	23	0.0490
0.500	0.550	0.622	16	0.0360	0.562	0.628	0.726	23	0.0490
0.562	0.613	0.685	16	0.0360	0.625	0.700	0.798	23	0.0490
0.625	0.683	0.755	16	0.0360	0.312	0.371	0.477	24	0.0530
0.250	0.292	0.368	17	0.0380	0.375	0.438	0.544	24	0.0530
0.312	0.358	0.434	17	0.0380	0.437	0.504	0.610	24	0.0530
0.375	0.423	0.499	17	0.0380	0.500	0.568	0.674	24	0.0530
0.437	0.486	0.562	17	0.0380	0.362	0.630	0.736	24	0.0530
0.500	0.554	0.630	17	0.0380	0.625	0.702	0.808	24	0.0530
0.562	0.615	0.691	17	0.6380	0.312	0.374	0.486	25	0.0560
0.625	0.686	0.762	17	0.0380	0.375	0.441	0.553	25	0.0560
0.250	0.294	0.374	18	0.0400	0.437	0.508	0.620	25	0.0560
0.312	0.361	0.441	18	0.0400	0.500	0.571	0.683	25	0.0560
0.375	0.426	0.506	18	0.0400	0.562	0.634	0.746	25	0.0560
0.437	0.489	0.569	18	0.0400	0.625	0.706	0.818	25	0.0560
0.500	0.557	0.637	18	0.0400	0.312	0.375	0.495	26	0.0600
0.562	0.618	0.698	18	0.0400	0.375	0.442	0.562	26	0.0600
0.625	0.690	0.770	18	0.0400	0.437	0.511	0.631	26	0.0600
0.312	0.363	0.447	19	0.0420	0.500	0.573	0.693	26	0.0600
0.375	0.427	0.511	19	0.0420	0.562	0.635	0.755	26	0.0600
0.437	0.491	0.575	19	0.0420	0.625	0.710	0.830	26	0.0600
0.500	0.558	0.642	19	0.0420	0.375	0.445	0.573	27	0.0640
0.562	0.619	0.703	19	0.0420	0.437	0.513	0.641	27	0.0640
0.625	0.69[illegible]	0.775	19	0.0420	0.500	0.575	0.703	27	0.0640
0.312	0.364	0.450	20	0.0430	0.562	0.637	0.765	27	0.0640
0.375	0.429	0.515	20	0.0430	0.625	0.713	0.841	27	0.0640
0.437	0.493	0.579	20	0.0430	0.375	0.446	0.583	28	0.0685
0.500	0.560	0.646	20	0.0430	0.437	0.514	0.651	28	0.0685
0.562	0.621	0.707	20	0.0430	0.500	0.575	0.712	28	0.0685
0.625	0.693	0.779	20	0.0430	0.562	0.638	0.775	28	0.0685
0.312	0.365	0.454	21	0.0445	0.625	0.714	0.851	28	0.0685
0.375	0.431	0.520	21	0.0445	0.375	0.448	0.591	29	0.0715
0.437	0.495	0.584	21	0.0445	0.437	0.516	0.659	29	0.0715
0.500	0.561	0.650	21	0.0445	0.500	0.577	0.720	29	0.0715
0.562	0.623	0.712	21	0.0445	0.562	0.640	0.783	29	0.0715
0.625	0.695	0.784	21	0.0445	0.625	0.714	0.857	29	0.0715
0.312	0.367	0.461	22	0.0470	0.375	0.451	0.603	30	0.0760
0.375	0.433	0.527	22	0.0470	0.437	0.518	0.670	30	0.0760
0.437	0.497	0.591	22	0.[illegible]470	0.500	0.580	0.732	30	0.0760
0.500	0.563	0.657	22	0.0470	0.562	0.643	0.795	30	0.0760
0.562	0.625	0.719	22	0.0470	0.625	0.717	0.869	30	0.0760
0.625	0.698	0.792	22	0.0470	0.375	0.455	0.617	31	0.0810

Table 15-3.—Data for Winding Piano Wire Tension Springs—Continued

Diam. of Mandrel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Number of Piano Wire	Diam. of Piano Wire, Inches	Diam. of Mandrel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Number of Piano Wire	Diam. of Piano Wire, Inches
0.437	0.522	0.684	31	0.081	0.375	0.480	0.682	34	0.101
0.500	0.585	0.747	31	0.081	0.437	0.550	0.752	34	0.101
0.562	0.647	0.809	31	0.081	0.500	0.610	0.812	34	0.101
0.625	0.722	0.884	31	0.081	0.562	0.673	0.875	34	0.101
0.375	0.461	0.633	32	0.086	0.625	0.750	0.952	34	0.101
0.437	0.527	0.699	32	0.086	0.375	0.490	0.708	35	0.109
0.500	0.590	0.762	32	0.086	0.437	0.560	0.778	35	0.109
0.562	0.651	0.823	32	0.086	0.500	0.622	0.840	35	0.109
0.625	0.727	0.899	32	0.086	0.562	0.686	0.904	35	0.109
0.375	0.467	0.649	33	0.091	0.625	0.765	0.983	35	0.109
0.437	0.533	0.715	33	0.091	0.375	0.500	0.736	36	0.118
0.500	0.595	0.777	33	0.091	0.437	0.572	0.808	36	0.118
0.562	0.657	0.839	33	0.091	0.500	0.637	0.873	36	0.118
0.625	0.733	0.915	33	0.091	0.562	0.702	0.938	36	0.118

plate. Make the groove the proper depth for the size of wire from which the springs are being wound. Tighten this clamping arrangement with the toolpost wrench. Use just enough tension on the wrench to keep the wire from slipping.

Further information and strengths of wire can be found in the *Machinery's Handbook*.

QUALITY ASSURANCE

Quality assurance is an inspection of manufactured parts to ensure that they meet blueprint specifications. Quality assurance is also used to lay out procedures in assembling and disassembling different components. Quality assurance should be used in all steps of manufacturing, such as checking diameters and lengths, etc. Basic quality assurance guidelines are usually set by type commanders such as SERVLANT, SUBLANT, SERVPAC, SUBPAC. Until it is coordinated under one system, you will have to follow local guidelines. In most ships and at shore installations there is also a calibration program where all measuring instruments are periodically checked for accuracy against standards. Usually, this program is coordinated by the IM shop. Before using measuring tools from the toolroom, you as the machine operator, should check for an up-to-date sticker affixed to the measuring device, and then check the instrument against the standard usually kept in the toolroom. In most cases, upon completion of a manufactured part, the shop quality assurance inspector will check the part against the blueprint for accuracy and document the result on a Quality Assurance Form. On this form is recorded the name of the ship, part manufacturer, print number used, serial number and calibration date of the instrument used to check the workpiece, the name of the person who manufactured the part, and the person who made the final quality assurance inspection. To determine type commander quality assurance guidelines, your shop leading petty officer should be able to find up-to-date information and have access to the appropriate directives and documents.

CALIBRATION SERVICING LABELS AND TAGS

Standards require a sticker or equivalent certification, showing the date and place of calibration, before they can be used to check operating instruments. Instruments calibrated by MIRCS require labels and tags to indicate the status of calibration or testing. In marking labels and tags, MIRCS personnel should insert in the DATE and DUE columns the appropriate month, day, and year, such as 8 Dec 1980. The Metrology Engineering Center's 3-letter code designation of the servicing MIRCS is written or stamped on applicable labels and tags. The various labels and tags for calibration standards or test and measuring equipment within MIRCS are shown in figures 15-35 and 15-36.

Calibrated

The CALIBRATED label is placed on each standard or test and measuring equipment that has been checked against a standard of higher accuracy, using approved Navy calibration procedures and checklists, and adjusted to meet (1) a predetermined specification (manufacturer's tolerance, or other) or (2) a specified value of magnitude and uncertainty. This specified value may be expressed for single-value instruments, such as standard cells, or for multivalue instruments, such as potentiometers. When an instrument is calibrated to meet a predetermined specification, only the knowledge that the instrument is within this specification is significant, and a black on white label is used. When an instrument is calibrated to meet an expressed value of magnitude and uncertainty, the actual measured value and associated uncertainty are reported, a red on white label is used, and a Report of Calibration is provided with the instrument.

Special Calibration

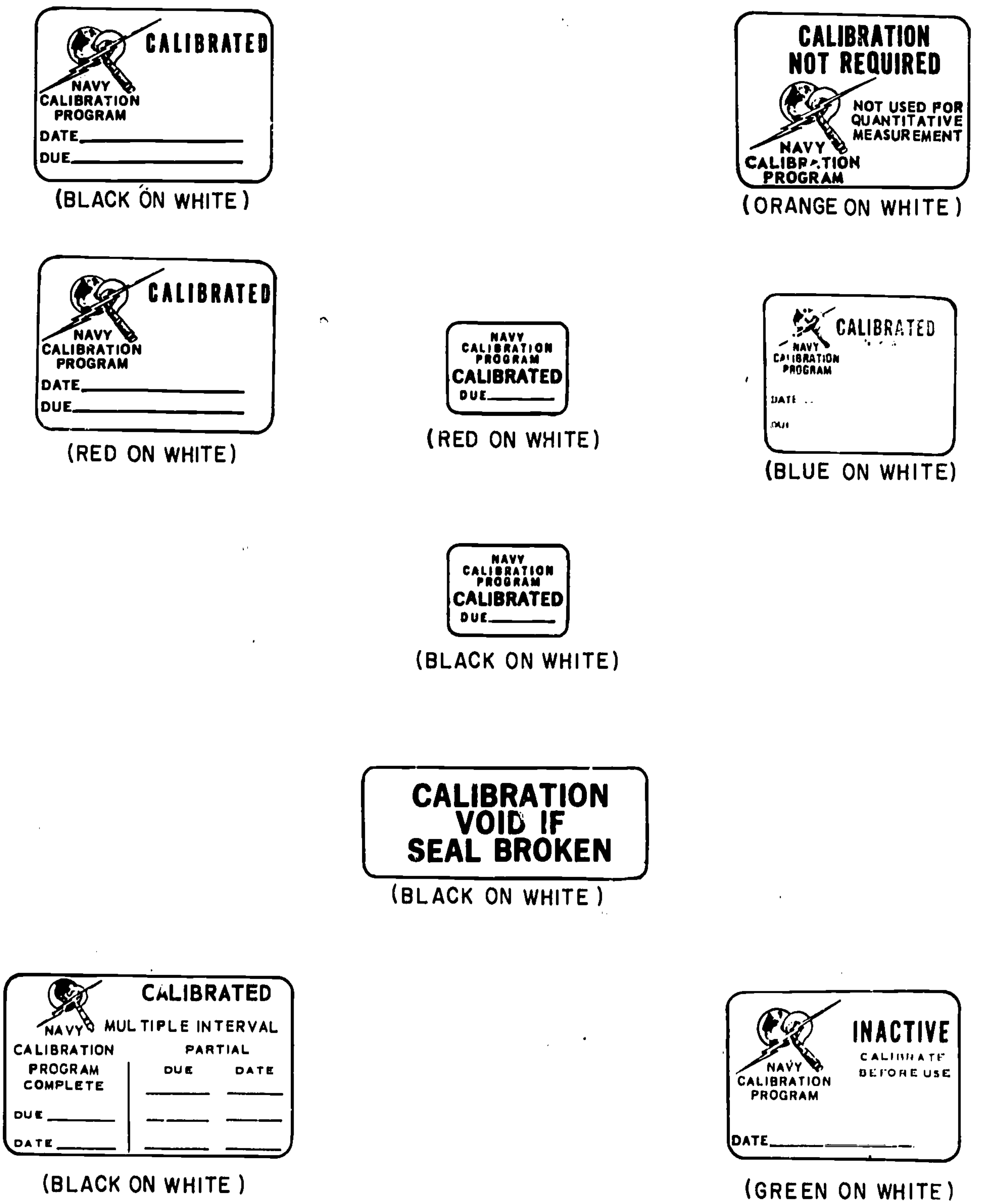
On occasion, specific user requirements do not involve the full instrument capability. In such instances a calibration is not performed over the entire range of the instrument. Only the needed quantities and ranges are calibrated. A SPECIAL CALIBRATION label (black and yellow) is used to draw attention to the special conditions under which the instrument is calibrated. In addition to the label, a special calibration tag is attached to the instrument. This tag is filled in by the servicing activity to adequately describe the conditions which are to be observed in the use of the instrument. The label and tag remain on the instrument until the next calibration. The 3-inch by 2-inch special calibration label may be used alone in lieu of the label and tag combination when space is available on the instrument and reasons for special calibration can be shown on the label itself.

Calibration Not Required—Not Used for Quantitative Measurement

Some instruments normally fall within the category of equipment requiring calibration, but are not used for quantitative measurements for various reasons. With several like instruments, for example, only one or two are calibrated and used for quantitative measurements; the others are used as indicators only. Also, some instruments do not require calibration because they receive an operational check each time they are used, or malfunctions and/or loss of accuracy are readily apparent during their normal use. These instruments comprise components, such as amplifiers, junction boxes, line transformers and line regulators. A label (orange on white), indicating that calibration is not required because the instrument is not used for quantitative measurements, is placed on the instrument.

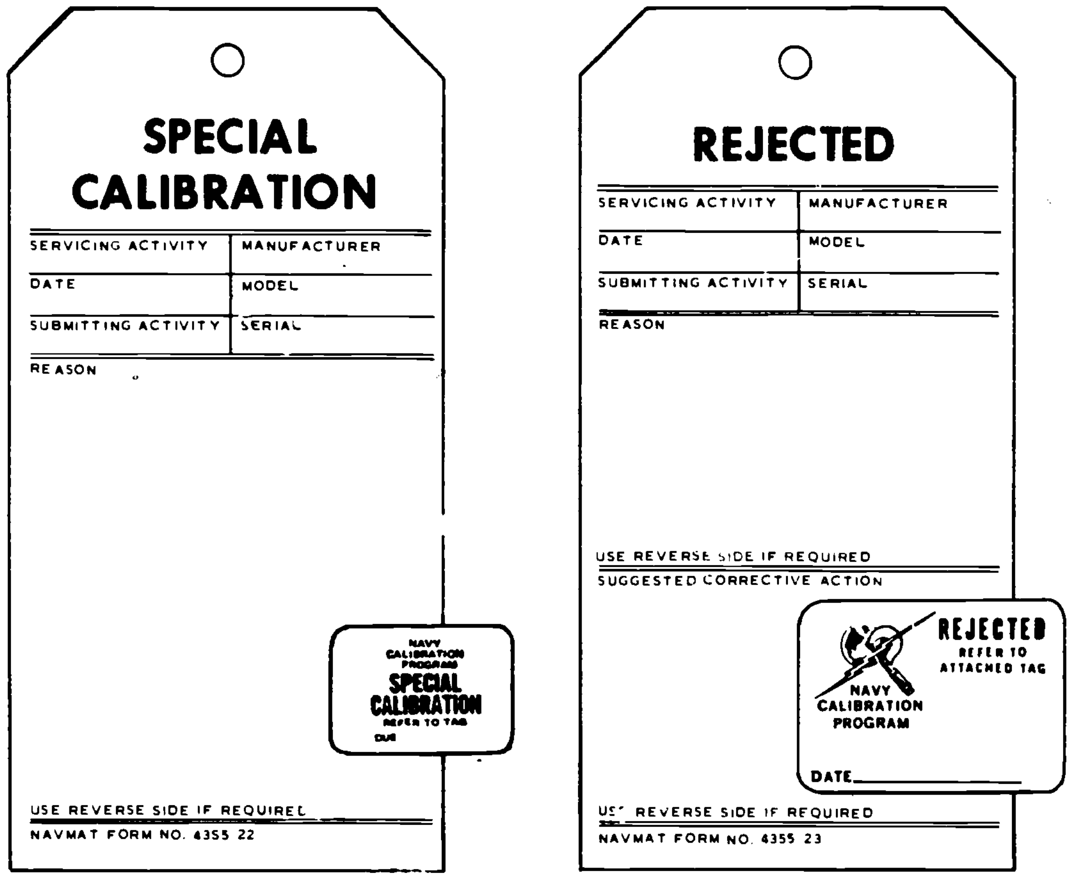
Calibrated-In-Place

The CALIBRATED-IN-PLACE label is used by on-site calibration teams to identify items, such as liquidometer indicators, that are calibrated in place and should not be forwarded to the calibration laboratory. These labels (blue on white) alert the ships' forces that the items should not be off-loaded when ships come into port.



1/3.6

Figure 15-35.—Labels.



(BLACK ON YELLOW) (BLACK ON RED)

173.6A

Figure 15-36.—Labels and Tags.

Calibration Void If Seal Broken

This label (black on white) is used to prevent tampering with certain adjustments which would affect the calibration.

Rejected

If an instrument fails to meet the acceptance criteria during calibration and cannot be adequately serviced, a REJECTED label (black on red) is placed on the instrument and all other servicing labels are removed. In addition to the REJECTED label, a REJECTED tag is placed on the instrument. The tag is filled in by the servicing activity giving the reason for rejection and other information as required. The rejected label and tag remain on the instrument until it is repaired and reserviced. The instrument is not to be used while bearing a rejected label.

Inactive

The INACTIVE label is placed on an instrument of the type which normally requires calibration and is found to have no foreseeable usage requirements. The inactive label remains on the instrument until it is reserviced. The instrument is not to be used while bearing the inactive label.

MIRCS Labels

Labels printed or procured locally are to be used on gages, tachometers, and other normal work instruments, such as those used on engines, boilers, and systems not used to test or calibrate other instruments. The labels may be printed using the MIRCS three-letter code, with date and date due, as shown in figure 15-35.

APPENDIX I

TABULAR INFORMATION OF BENEFIT TO MACHINERY REPAIRMAN

Table I-1.—Decimal Equivalents of Fractions (Inch)

fractions	# 64ths	# 32ds	# 16ths	# 8ths	# 4ths	decimal equiv.	fractions	# 64ths	# 32ds	# 16ths	# 8ths	# 4ths	decimal equiv.
1/64	1					0.015625	33/64	33					0.515625
1/32	2	1				0.03125	17/32	34	17				0.53125
3/64	3					0.046875	35/64	35					0.546875
1/16	4	2	1			0.0625	/16	36	18	9			0.5625
5/64	5					0.078125	37/64	37					0.578125
3/32	6	3				0.09375	19/32	38	19				0.59375
7/64	7					0.109375	39/64	39					0.609375
1/8	8	4	2	1		0.125	5/8	40	20	10	5		0.625
9/64	9					0.140625	41/64	41					0.640625
5/32	10	5				0.15625	21/32	42	21				0.65625
11/64	11					0.171875	43/64	43					0.671875
3/16	12	6	3			0.1875	11/16	44	22	11			0.6875
13/64	13					0.203125	45/64	45					0.703125
7/32	14	7				0.21875	23/32	46	23				0.71875
15/64	15					0.234375	47/64	47					0.734375
1/4	16	8	4	2	1	0.250	3/4	48	24	12	6	3	0.750
17/64	17					0.265625	49/64	49					0.765625
9/32	18	9				0.28125	25/32	50	25				0.78125
19/64	19					0.296875	51/64	51					0.796875
5/16	20	10	5			0.3125	13/16	52	26	13			0.8125
21/64	21					0.328125	53/64	53					0.828125
11/32	22	11				0.34375	27/32	54	27				0.84375
23/64	23					0.359375	55/64	55					0.859375
3/8	24	12	6	3		0.375	7/8	56	28	14	7		0.875
25/64	25					0.390625	57/64	57					0.890625
13/32	26	13				0.40625	29/32	58	29				0.90625
27/64	27					0.421875	59/64	59					0.921875
7/16	28	14	7			0.4375	15/16	60	30	15			0.9375
29/64	29					0.453125	61/64	61					0.953125
15/32	30	15				0.46875	31/32	62	31				0.96875
31/64	31					0.484375	63/64	63					0.984375
1/2	32	16	8	4	2	0.500	1 inch	64	32	16	8	4	1.000

Table I-2.—Decimal Equivalents of Millimeters

mm	inches	mm	inches	mm	inches	mm	inches	mm	inches
0.1	0.00394	3.5	0.13779	6.9	0.27165	10.3	0.40551	13.8	0.54330
0.2	0.00787	3.6	0.14173	7.0	0.27559	10.4	0.40944	13.9	0.54724
0.3	0.01181	3.7	0.14566	7.1	0.27952	10.5	0.41388	14.0	0.55118
0.4	0.01575	3.8	0.14960	7.2	0.28346	10.6	0.41732	14.1	0.55511
0.5	0.01968	3.9	0.15354	7.3	0.28740	10.7	0.42125	14.2	0.55905
0.6	0.02362	4.0	0.15748	7.4	0.29133	10.8	0.42519	14.3	0.56299
0.7	0.02756	4.1	0.16141	7.5	0.29527	10.9	0.42913	14.4	0.56692
0.8	0.03149	4.2	0.16535	7.6	0.29921	11.0	0.43307	14.5	0.57086
0.9	0.03543	4.3	0.16929	7.7	0.30314	11.1	0.43700	14.6	0.57480
1.0	0.03937	4.4	0.17322	7.8	0.30708	11.2	0.44094	14.7	0.57873
1.1	0.04330	4.5	0.17716	7.9	0.31102	11.3	0.44488	14.8	0.58267
1.2	0.04724	4.6	0.18110	8.0	0.31496	11.4	0.44881	14.9	0.58661
1.3	0.05118	4.7	0.18503	8.1	0.31889	11.5	0.45275	15.0	0.59055
1.4	0.05512	4.8	0.18897	8.2	0.32283	11.6	0.45669	15.5	0.61023
1.5	0.05905	4.9	0.19291	8.3	0.32677	11.7	0.46062	16.0	0.62992
1.6	0.06299	5.0	0.19685	8.4	0.33070	11.8	0.46456	16.5	0.64960
1.7	0.06692	5.1	0.20078	8.5	0.33464	11.9	0.46850	17.0	0.66929
1.8	0.07086	5.2	0.20472	8.6	0.33858	12.0	0.47244	17.5	0.68897
1.9	0.07480	5.3	0.20866	8.7	0.34251	12.1	0.47637	18.0	0.70866
2.0	0.07874	5.4	0.21259	8.8	0.34645	12.2	0.48031	18.5	0.72834
2.1	0.08267	5.5	0.21653	8.9	0.35039	12.3	0.48425	19.0	0.74803
2.2	0.08661	5.6	0.22047	9.0	0.35433	12.4	0.48818	19.5	0.76771
2.3	0.09055	5.7	0.22440	9.1	0.35826	12.5	0.49212	20.0	0.78740
2.4	0.09448	5.8	0.22834	9.2	0.36220	12.6	0.49606	20.5	0.80708
2.5	0.09842	5.9	0.23228	9.3	0.36614	12.7	0.49999	21.0	0.82677
2.6	0.10236	6.0	0.23622	9.4	0.37007	12.8	0.50393	21.5	0.84645
2.7	0.10629	6.1	0.24015	9.5	0.37401	12.9	0.50787	22.0	0.86614
2.8	0.11023	6.2	0.24409	9.6	0.37795	13.0	0.51181	22.5	0.88582
2.9	0.11417	6.3	0.24803	9.7	0.38188	13.1	0.51574	23.0	0.90551
3.0	0.11811	6.4	0.25196	9.8	0.38582	13.2	0.51968	23.5	0.92519
3.1	0.12204	6.5	0.25590	9.9	0.38976	13.3	0.52362	24.0	0.94488
3.2	0.12598	6.6	0.25984	10.0	0.39370	13.4	0.52755	24.5	0.96456
						13.5	0.53149	25.0	0.98425
3.3	0.12992	6.7	0.26377	10.1	0.39763	13.6	0.53543	25.5	1.00393
3.4	0.13385	6.8	0.26771	10.2	0.40157	13.7	0.53936	26.0	1.02362

Table I-3.—Dividing a Circle into Parts

To find the length of the chord for dividing the circumference of a circle into a required number of equal parts, multiply the factor in the table by the diameter.

no. of spaces	chord length	no. of spaces	chord length	no. of spaces	chord length
3	0.866	21	0.149	39	0.0805
4	0.7071	22	0.1423	40	0.0785
5	0.5878	23	0.1362	41	0.0765
6	0.5	24	0.1305	42	0.0747
7	0.4339	25	0.1253	43	0.073
8	0.3827	26	0.1205	44	0.0713
9	0.342	27	0.1161	45	0.0698
10	0.309	28	0.112	46	0.0682
11	0.2818	29	0.1081	47	0.0668
12	0.2584	30	0.1045	48	0.0654
13	0.2393	31	0.1012	49	0.0641
14	0.2224	32	0.098	50	0.0628
15	0.2079	33	0.0951	51	0.0616
16	0.1951	34	0.0932	52	0.0604
17	0.1837	35	0.0896	53	0.0592
18	0.1736	36	0.0872	54	0.0581
19	0.1645	37	0.0848	55	0.0571
20	0.1564	38	0.0826		

Table I-4.—Formulas for Dimension, Area, and Volume

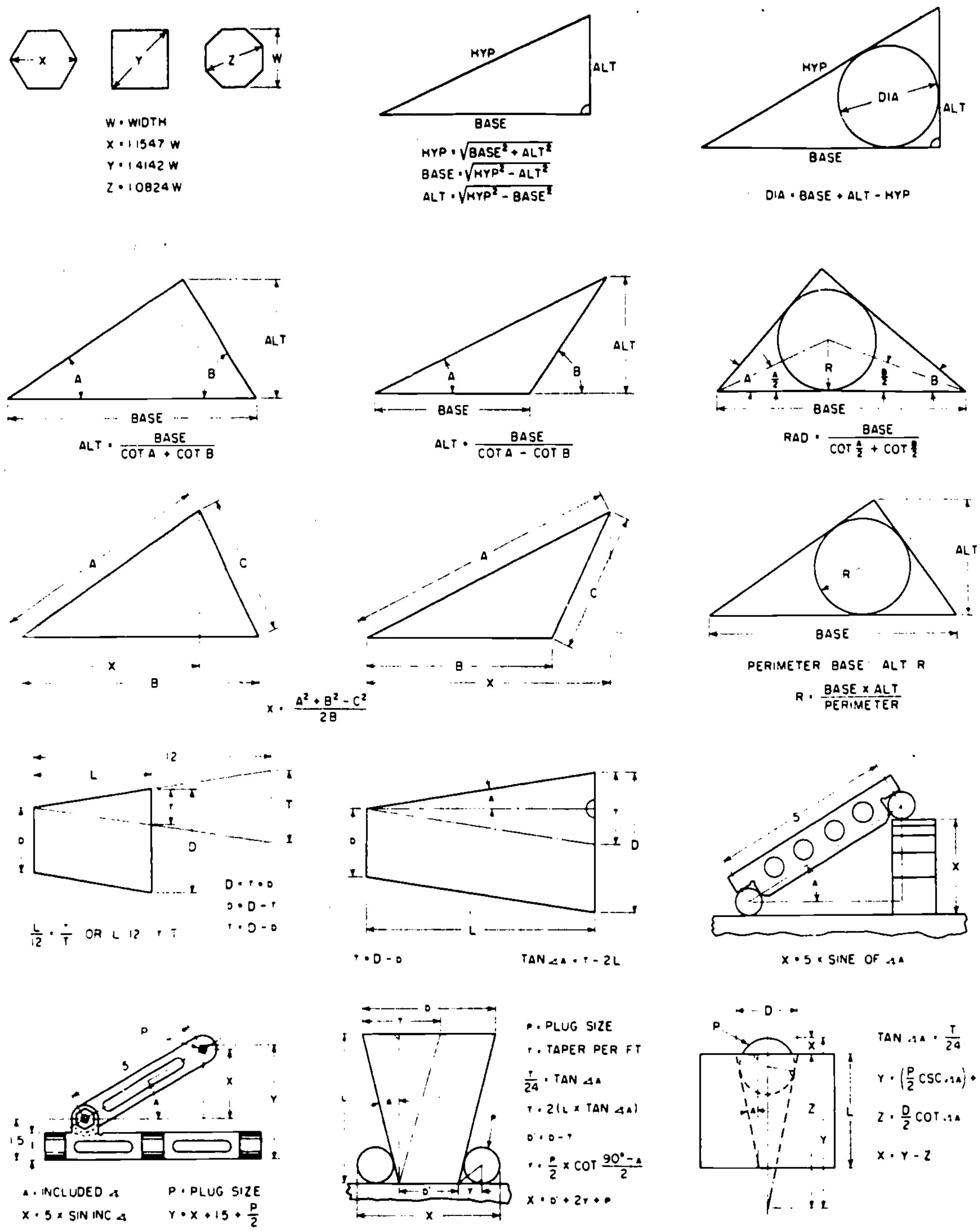
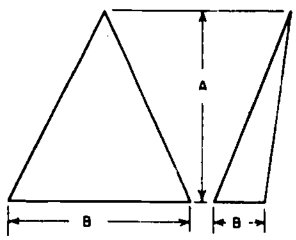


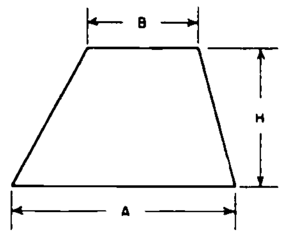
Table I-4.—Formulas for Dimension, Area, and Volume—Continued

TRIANGLE



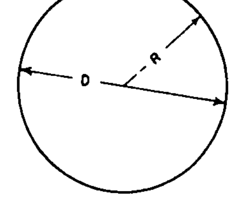
$AREA = \frac{1}{2} B \times A$

TRAPEZOID



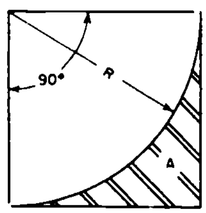
$AREA = \frac{1}{2}(A+B)H$

CIRCLE



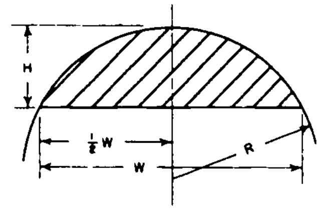
$AREA = 3.1416\,R^2$

FILLET



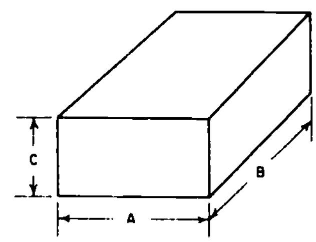
$A = R^2 - \frac{3.1416\,R^2}{4}$

SEGMENT



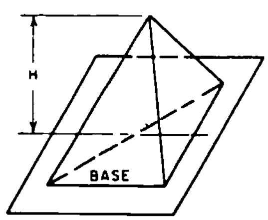
$R = \frac{(\frac{1}{2}W)^2 + H^2}{2H}$

RECTANGULAR PRISM



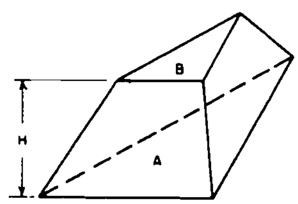
$VOLUME = ABC$

TRIANGULAR PYRAMID



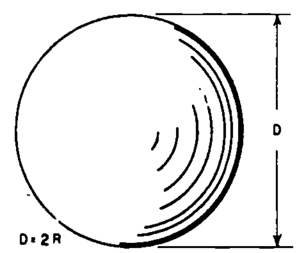
$VOLUME = \frac{AREA\ OF\ BASE \times H}{3}$

FRUSTUM OF PYRAMID



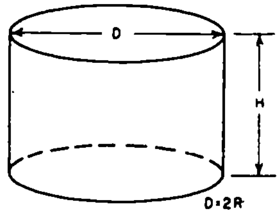
$VOLUME = \frac{H(A+B+\sqrt{AB})}{3}$

SPHERE



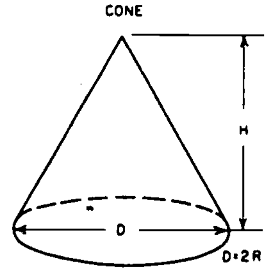
$VOLUME = \frac{4 \times 3.1416\,R^3}{3}$

CYLINDER



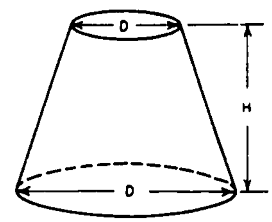
$VOLUME = 3.1416\,R^2 \times H$

CONE



$VOLUME = \frac{3.1416\,R^2 \times H}{3}$

FRUSTUM OF CONE



$VOL = 0.2618\,H(D^2 + d^2 + Dd)$

Table I-5.—Formulas for Circles

Circumference of a circle	= Diameter multiplied by 3.1416 = Diameter divided by 0.3183
Diameter of a circle	= Circumference multiplied by 0.3183 = Circumference divided by 3.1416
Side of a square inscribed in a given circle	= Diameter multiplied by 0.7071 = Circumference multiplied by 0.2251 = Circumference divided by 4.4428
Side of a square with area of a given circle	= Diameter multiplied by 0.8862 = Diameter divided by 1.1284 = Circumference multiplied by 0.2821 = Circumference divided by 3.545
Diameter of a circle with area of a given square	= Side multiplied by 1.128
Diameter of a circle circumscribing a given square	= Side multiplied by 1.4142
Area of a circle	= The square of the diameter multiplied by 0.7854 = The square of the radius multiplied by 3.1416
Area of the surface of a sphere or globe	= The square of the diameter multiplied by 3.1416

Table I-6.—Number, Letter and Fractional Identification of Drill Sizes (Letter drills are larger than number drills; they begin where number drills end.)

no. & letter drills	frac-tional drills	dec-imal equiv.	no. & letter drills	frac-tional drills	dec-imal equiv.	no. & letter drills	frac-tional drills	dec-imal equiv.	no. & letter drills	frac-tional drills	dec-imal equiv.
80	...	.0135	42	...	.0935		13/64	.2031		13/32	.4062
79	...	.0145		3/32	.0937	6	...	.2040	Z	...	.4130
	1/64	.0156	41	...	.0960	5	...	.2055		27/64	.4219
78	...	.0160	40	...	.0980	4	...	.2090		7/16	.4375
77	...	.0180	39	...	.0995	3	...	.2130		29/64	.4531
76	...	.0200	38	...	.1015		7/32	.2187		15/32	.4687
75	...	.0210	37	...	.1040	2	...	.2210		31/64	.4844
74	...	.0225	36	...	.1065	1	...	.2280		1/2	.5000
73	...	.0240		7/64	.1094	A	...	.2340		33/64	.5156
72	...	.0250	35	...	.1100		15/64	.2344		17/32	.5312
71	...	.0260	34	...	.1110	B	...	.2380		35/64	.5469
70	...	.0280	33	...	.1130	C	...	.2420		9/16	.5625
69	...	.0292	32	...	.1160	D	...	.2460		37/64	.5781
68	...	.0310	31	...	.1200	E	1/4	.2500		19/32	.5937
	1/32	.0312		1/8	.1250	F	...	.2570		39/64	.6094
67	...	.0320	30	...	.1285	G	...	.2610		5/8	.6250
66	...	.0330	29	...	.1360		17/64	.2656		41/64	.6406
65	...	.0350	28	...	.1405	H	...	.2660		21/32	.6562
64	...	.0360		9/64	.1406	I	...	.2720		43/64	.6719
63	...	.0370	27	...	.1440	J	...	.2770		11/16	.6875
62	...	.0380	26	...	.1470	K	...	.2810		45/64	.7031
61	...	.0390	25	...	.1495		9/32	.2812		23/32	.7187
60	...	.0400	24	...	.1520	L	...	.2900		47/64	.7344
59	...	.0410	23	...	.1540	M	...	.2950		3/4	.7500
58	...	.0420		5/32	.1562		19/64	.2969		49/64	.7656
57	...	.0430	22	...	.1570	N	...	.3020		25/32	.7812
56	...	.0465	21	...	.1590		5/16	.3125		51/64	.7969
	3/64	.0469	20	...	.1610	O	...	.3160		13/16	.8125
55	...	.0520	19	...	.1660	P	...	.3230		53/64	.8281
54	...	.0550	18	...	.1695		21/64	.3281		27/32	.8437
53	...	.0595		11/64	.1719	Q	...	.3320		55/64	.8594
	1/16	.0625	17	...	.1720	R	...	.3390		7/8	.8750
52	...	.0635	16	...	.1770		11/32	.3437		57/64	.8906
51	...	.0670	15	...	.1800	S	...	.3480		29/32	.9062
50	...	.0700	14	...	.1820	T	...	.3580		59/64	.9219
49	...	.0730	13	...	.1850		23/64	.3594		15/16	.9375
48	...	.0760		3/16	.1875	U	...	.3680		61/64	.9531
	5/64	.0781	12	...	.1890		3/8	.3750		31/32	.9687
47	...	.0785	11	...	.1910	V	...	.3770		63/64	.9844
46	...	.0810	10	...	.1935	W	...	.3860		1	1.0000
45	...	.0820	9	...	.1960		25/64	.3906			
44	...	.0860	8	...	.1990	X	...	.3970			
43	...	.0890	7	...	.2010	Y	...	.4040			

Table I-7.—Units of Weight, Volume, and Temperature

AVOIRDUPOIS WEIGHT

16 drams or 437.5 grains = 1 ounce
16 ounces or 7,000 grains = 1 pound
2,000 pounds = 1 net or short ton
2,240 pounds = 1 gross or long ton
2,204.6 pounds = 1 metric ton

BOARD MEASURE

One board foot measure is a piece of wood 12 inches square by 1 inch thick, or 144 cubic inches. A piece of wood 2 by 4, 12 feet long contains 8 feet board measure.

DRY MEASURE

2 pints = 1 quart
8 quarts = 1 peck
4 pecks = 1 bushel
1 standard U.S. bushel = 1.2445 cubic feet
1 British imperial bushel = 1.2837 cubic feet

LIQUID MEASURE

4 gills = 1 pint
2 pints = 1 quart
4 quarts = 1 gallon
1 U.S. gallon = 231 cubic inches
1 British imperial gallon = 1.2 U.S. gallons
7.48 U.S. gallons = 1 cubic foot

LONG MEASURE

12 inches = 1 foot
3 feet = 1 yard
1,760 yards = 1 mile
5,280 feet = 1 mile
16.5 feet = 1 rod

PAPER MEASURE

24 sheets = 1 quire
20 quires = 1 ream
2 reams = 1 bundle
5 bundles = 1 bale

SHIPPING MEASURE

1 U.S. shipping ton = 40 cubic feet
1 U.S. shipping ton = 32.143 U.S. bushels
1 U.S. shipping ton = 31.16 imperial bushels
1 British shipping ton = 42 cubic feet
1 British shipping ton = 33.75 U.S. bushels
1 British shipping ton = 32.718 imperial bushels

SQUARE MEASURE

144 square inches = 1 square foot
9 square feet = 1 square yard
30.25 square yards = 1 square rod
160 square rods = 1 acre
640 acres = 1 square mile

TEMPERATURE

Freezing, Fahrenheit scale = 32 degrees
Freezing, celcius scale = 0 degrees
Boiling, Fahrenheit scale = 212 degrees
Boiling, celcius scale = 100 degrees

If any degree on the celcius scale, either above or below zero, be multiplied by 1.8, the result will, in either case, be the number of degrees above or below 32 degrees Fahrenheit.

TROY WEIGHT

24 grains = 1 pennyweight
20 pennyweights = 1 ounce
12 ounces = 1 pound

WEIGHT OF WATER

1 cubic centimeter = 1 gram or 0.035 ounce
1 cubic inch = 0.5787 ounce
1 cubic foot = 62.48 pounds
1 U.S. gallon = 8.355 pounds
1 British imperial gallon = 10 pounds
32 cubic feet = 1 net ton (2,000 pounds)
35.84 cubic feet = 1 long ton (2,240 pounds)
1 net ton = 240 U.S. gallons
1 long ton = 268 U.S. gallons

ENGLISH-METRIC EQUIVALENTS

1 inch = 2.54 centimeters
1 centimeter = 0.3937 inch
1 meter = 39.37 inches
1 kilometer = 0.62 mile
1 quart = 0.946 liter
1 U.S. gallon = 3.785 liters
1 British gallon = 4.543 liters
1 liter = 1.06 quarts
1 pound = 0.454 kilogram
1 kilogram = 2.205 pounds
1 watt = 44.24 foot-pounds per minute
1 horsepower = 33,000 foot-pounds per minute
1 kilowatt = 1.34 horsepower

Table I-8.—Screw Thread and Tap Drill Sizes (American National)

	threads per inch		dimensions, inches				tap drill 75% full thread			
screw size	NC coarse thread	NF fine thread	major diameter	pitch diameter	single depth of thread	minor diameter	tap drill	decimal equiv.	body drill	decimal equiv.
0		80	0.060	0.0519	0.00812	0.0438	3/64	0.0469	52	0.0635
1	64		0.073	0.0629	0.01015	0.0527	53	0.0595	47	0.0785
1		72	0.073	0.0640	0.00902	0.0550	53	0.0595	47	0.0785
2	56		0.086	0.0744	0.01160	0.0628	50	0.0700	42	0.0935
2		64	0.086	0.0759	0.01015	0.0657	50	0.0700	42	0.0935
3	48		0.099	0.0855	0.01353	0.0719	47	0.0785	37	0.1040
3		56	0.099	0.0874	0.01160	0.0758	45	0.0820	37	0.1040
4	40		0.112	0.0958	0.01624	0.0795	43	0.0890	31	0.1200
4		48	0.112	0.0985	0.01353	0.0849	42	0.0935	31	0.1200
5	40		0.125	0.1088	0.01624	0.0925	38	0.1015	29	0.1360
5		44	0.125	0.1102	0.01476	0.0955	37	0.1040	29	0.1360
6	32		0.138	0.1177	0.02030	0.0974	36	0.1065	27	0.1440
6		40	0.138	0.1218	0.01624	0.1055	33	0.1130	27	0.1440
8	32		0.164	0.1437	0.02030	0.1234	29	0.1360	18	0.1695
8		36	0.164	0.1460	0.01804	0.1279	29	0.1360	18	0.1695
10	24		0.190	0.1629	0.02706	0.1359	25	0.1495	9	0.1960
10		32	0.190	0.1697	0.02030	0.1494	21	0.1590	9	0.1960
12	24		0.216	0.1889	0.02706	0.1619	16	0.1770	2	0.2210
12		28	0.216	0.1928	0.02320	0.1696	14	0.1820	2	0.2210
1/4	20		0.2500	0.2175	0.03248	0.1850	7	0.2010		
1/4		28	0.2500	0.2268	0.02320	0.2036	3	0.2130		
5/16	18		0.3125	0.2764	0.03608	0.2403	F	0.2570		
5/16		24	0.3125	0.2854	0.02706	0.2584	I	0.2720		
3/8	16		0.3750	0.3344	0.04059	0.2938	5/16	0.3125		
3/8		24	0.3750	0.3479	0.02706	0.3209	Q	0.3320		
7/16	14		0.4375	0.3911	0.04639	0.3447	U	0.3680		
7/16		20	0.4375	0.4050	0.03248	0.3725	25/64	0.3906		
1/2	13		0.5000	0.4500	0.04996	0.4001	27/64	0.4219		
1/2		20	0.5000	0.4675	0.03248	0.4350	29/64	0.4531		
9/16	12		0.5625	0.5084	0.05413	0.4542	31/64	0.4844		
9/16		18	0.5625	0.5264	0.03608	0.4903	33/64	0.5156		
5/8	11		0.6250	0.5660	0.05905	0.5069	17/32	0.5313		
5/8		18	0.6250	0.5889	0.03608	0.5528	37/64	0.5781		
3/4	10		0.7500	0.6850	0.06495	0.6201	21/32	0.6562		
3/4		16	0.7500	0.7094	0.04059	0.6688	11/16	0.6875		
7/8	9		0.8750	0.8028	0.07217	0.7307	49/64	0.7656		
7/8		14	0.8750	0.8286	0.04639	0.7822	13/16	0.8125		
1	8		1.0000	0.9188	0.08119	0.8376	7/8	0.8750		
1		14	1.0000	0.9536	0.04639	0.9072	15/16	0.9375		
1 1/8	7		1.1250	1.0322	0.09279	0.9394	63/64	0.9844		
1 1/8		12	1.1250	1.0709	0.05413	1.0167	1 3/64	1.0469		
1 1/4	7		1.2500	1.1572	0.09279	1.0644	1 7/64	1.1094		
1 1/4		12	1.2500	1.1959	0.05413	1.1417	1 11/64	1.1719		
1 3/8	6		1.3750	1.2667	0.10825	1.1585	1 7/32	1.2188		
1 3/8		12	1.3750	1.3209	0.05413	1.2667	1 19/64	1.2969		
1 1/2	6		1.5000	1.3917	0.10825	1.2835	1 11/32	1.3438		
1 1/2		12	1.5000	1.4459	0.05413	1.3917	1 27/64	1.4219		
1 3/4	5		1.7500	1.6201	0.12990	1.4902	1 9/16	1.5625		
2	4 1/2		2.0000	1.8557	0.14434	1.7113	1 25/32	1.7813		

Table I-9.—Full Thread Produced in Tapped Holes (Percentage)

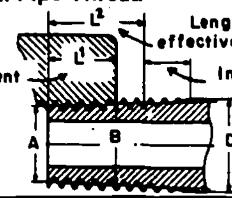
tap	tap drill	decimal tap drill	usual hole size	thread percentage
0-80	56	0.0465	0.0480	74
	3/64	0.0469	0.0484	71
1-64	54	0.0550	0.0565	81
	53	0.0595	0.0610	59
1-72	53	0.0595	0.0610	67
	1/16	0.0625	0.0640	50
2-56	51	0.0670	0.0687	74
	50	0.0700	0.0717	62
	49	0.0730	0.0747	49
2-64	50	0.0700	0.0717	70
	49	0.0730	0.0747	56
3-48	48	0.0760	0.0779	78
	5/64	0.0781	0.0800	70
	47	0.0785	0.0804	69
	46	0.0810	0.0829	60
	45	0.0820	0.0839	56
3-56	46	0.0810	0.0829	69
	45	0.0820	0.0839	65
	44	0.0860	0.0879	48
4-40	44	0.0860	0.0880	74
	43	0.0890	0.0910	65
	42	0.0935	0.0955	51
	3/32	0.0938	0.0958	50
4-48	42	0.0935	0.0955	61
	3/32	0.0938	0.0958	60
	41	0.0960	0.0980	52
5-40	40	0.0980	0.1003	76
	39	0.0995	0.1018	71
	38	0.1015	0.1038	65
	37	0.1040	0.1063	58
5-44	38	0.1015	0.1038	72
	37	0.1040	0.1063	63
	36	0.1065	0.1088	55
6-32	37	0.1040	0.1063	78
	36	0.1065	0.1091	71
	7/64	0.1094	0.1120	64
	35	0.1100	0.1126	63
	34	0.1110	0.1136	60
	33	0.1130	0.1156	55
6-40	34	0.1110	0.1136	75
	33	0.1130	0.1156	69
	32	0.1160	0.1186	60
8-32	29	0.1360	0.1389	62
	28	0.1405	0.1434	51
8-36	29	0.1360	0.1389	70
	28	0.1405	0.1434	57
	9/64	0.1406	0.1435	57
10-24	27	0.1440	0.1472	79
	26	0.1470	0.1502	74
	25	0.1495	0.1527	69
	24	0.1520	0.1552	64
	23	0.1540	0.1572	61
	5/32	0.1563	0.1595	56
	22	0.1570	0.1602	55
10-32	5/32	0.1563	0.1595	75
	22	0.1570	0.1602	73
	21	0.1590	0.1622	68
	20	0.1610	0.1642	64
	19	0.1660	0.1692	51
12-24	11/64	0.1719	0.1754	75
	17	0.1730	0.1765	73
	16	0.1770	0.1805	66
	15	0.1800	0.1835	60
	14	0.1820	0.1855	56
12-28	16	0.1770	0.1805	77
	15	0.1800	0.1835	70

Table I-9.—Full Thread Produced in Tapped Holes (Percentage)—Continued

tap	tap drill	decimal tap drill	usual hole size	thread percentage
12-28	14	0.1820	0.1855	66
	13	0.1850	0.1885	59
	3/16	0.1875	0.1910	54
1/4-20	9	0.1960	0.1998	77
	8	0.1990	0.2028	73
	7	0.2010	0.2048	70
	13/64	0.2031	0.2069	66
	6	0.2040	0.2078	65
	5	0.2055	0.2093	63
	4	0.2090	0.2128	57
1/4-28	3	0.2130	0.2168	72
	7/32	0.2188	0.2226	59
	2	0.2210	0.2248	55
5/16-18	F	0.2570	0.2608	72
	G	0.2610	0.2651	66
	17/64	0.2656	0.2697	59
	H	0.2660	0.2701	59
5/16-24	H	0.2660	0.2701	78
	I	0.2720	0.2761	67
	J	0.2770	0.2811	58
3/8-16	5/16	0.3125	0.3169	72
	O	0.3160	0.3204	68
	P	0.3230	0.3274	59
3/8-24	21/64	0.3281	0.3325	79
	Q	0.3320	0.3364	71
	R	0.3390	0.3434	58
7/16-14	T	0.3580	0.3626	81
	23/64	0.3594	0.3640	79
	U	0.3680	0.3726	[illegible]
	3/8	0.3750	0.3796	62
	V	0.3770	0.3816	60
7/16-20	W	0.3860	0.3906	72
	25/64	0.3906	0.3952	65
	X	0.3970	0.4016	55
1/2-13	27/64	0.4219	0.4266	73
	7/16	0.4375	0.4422	58
1/2-20	29/64	0.4531	0.4578	65
9/16-12	15/32	0.4688	0.4736	82
	31/64	0.4844	0.4892	68
9/16-18	1/2	0.5000	0.5048	80
	33/64	0.5156	0.5204	58
5/8-11	17/32	0.5313	0.5362	75
	35/64	0.5469	0.5518	62
5/8-18	9/16	0.5625	0.5674	80
	37/64	0.5781	0.5831	58
3/4-10	41/64	0.6406	0.6456	80
	21/32	0.6563	0.6613	68
3/4-16	11/16	0.6875	0.6925	71
7/8-9	49/64	0.7656	0.7708	72
	25/32	0.7812	0.7864	61
7/8-14	51/64	0.7969	0.8021	79
	13/16	0.8125	0.8177	62
1-8	55/64	0.8594	0.8653	83
	7/8	0.8750	0.8809	73
	57/64	0.8906	0.8965	64
	29/32	0.9063	0.9122	54
1-12	29/32	0.9063	0.9123	81
	59/64	0.9219	0.9279	67
	15/16	0.9375	0.9435	52
1-14	59/64	0.9219	0.9279	78
	15/16	0.9375	0.9435	61

Table 1-10.—American National Pipe Thread

A = Pitch diameter of thread at end of pipe
B = Pitch diameter of thread at gauging notch
D = Outside diameter of pipe
L^1 = Normal engagement by hand between external and internal thread



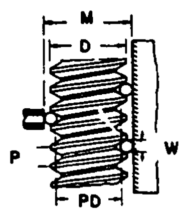
Size Inches	Threads per inch	Pitch Diameter A Inches	Pitch Diameter B Inches	Length L2 Inches	Length L1 Inches	Pipe O.D. D Inches	Depth of Thread Inches	Tap Drills for Pipe Threads: Minor Diameter Small End of Pipe	Tap Drills for Pipe Threads: Size Drill
1/8	27	.36351	.37476	.2639	.180	.405	.02963	.3339	R
1/4	18	.47739	.48989	.4018	.200	.540	.04444	.4329	7/16
3/8	18	.61201	.62701	.4078	.240	.675	.04444	.5676	37/64
1/2	14	.75843	.77843	.5337	.320	.840	.05714	.7013	23/32
3/4	14	.96768	.98887	.5457	.339	1.050	.05714	.9105	59/64
1	11½	1.21363	1.23863	.6828	.400	1.315	.06957	1.1441	1 5/32
1¼	11½	1.55713	1.58338	.7068	.420	1.660	.06957	1.4876	1½
1½	11½	1.79609	1.82234	.7235	.420	1.900	.06957	1.7265	1 47/64
2	11½	2.26902	2.29627	.7565	.436	2.375	.06957	2.1995	2 7/32
2½	8	2.71953	2.76216	1.1375	.682	2.875	.10000	2.6195	2 5/8
3	8	3.34062	3.38850	1.2000	.766	3.500	.10000	3.2406	3¼
3½	8	3.83750	3.88881	1.2500	.821	4.000	.10000	3.7375	3¾
4	8	4.33438	4.38712	1.3000	.844	4.500	.10000	4.2344	4¼

Table 1-11.—3 Wire Method—American National Std.

$$M = D - (1.5156 \times P) + (3 \times W)$$

$$PD = M + \frac{.86603}{\text{No. of thds. per inch}} - (3 \times W)$$

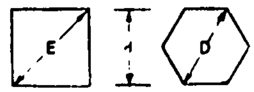
$$\text{To Check Angle } \frac{M_1 - M_2}{W_1 - W_2}$$



M = Measurement over best size wire.
M_1 = Measurement over maximum size wire
M_2 = Measurement over minimum size wire
D = Outside Diameter of Thread.
P.D. = Pitch Diameter.
W = Diameter Best size wire. 0.57735 × pitch
W_1 = Diameter maximum size wire. 0.90 × pitch.
W_2 = Diameter minimum size wire. 0.56 × pitch

No. Thds. per inch	Pitch Thds. per inch	Best Wire Size .57735 x Pitch	Maximum Wire Size	Minimum Wire Size
4	.250000	.144337	.225000	.140000
4½	.222222	.128300	.200000	.124444
5	.200000	.115470	.180000	.112000
5½	.181818	.104969	.163636	.101818
6	.166666	.096224	.149999	.093333
7	.142857	.082478	.128571	.080000
7½	.133333	.076979	.120000	.074666
8	.125000	.072168	.112500	.070000
9	.111111	.064149	.100000	.062222
10	.100000	.057735	.090000	.056000
11	.090999	[illegible]	.081818	.050909
11½	.086956	.050204	.078260	.048695
12	.083333	.048112	.075000	.046666
13	.076923	.044411	.069231	.043077
14	.071428	.041239	.064285	.040000
16	.062500	.036084	.056250	.035000
18	.055555	.032074	.050000	[illegible]
20	.050000	.028867	.045000	.028000
22	.045454	.026242	.040909	.025454
24	.041666	.024055	.037499	.023333
26	.038461	.022205	.034615	.021538
27	.037037	.021383	.033333	.022543
28	.035714	.020620	.032143	.020000
30	.033333	.019244	.030000	.018666
32	.031250	.018042	028125	.017500
36	.027777	.016037	.024999	.015555
40	.025000	.014433	.022500	.014000
44	.022727	.013121	.020454	.014727
48	.020833	.012027	.018750	.011666
50	.020000	.011547	.018000	.011200
56	.017857	.010309	.016071	.010000
64	.015625	.009021	.014063	.008750
72	.013888	.008018	.012499	.007777
80	.012500	.007216	.011250	.007000

Table I-12.—Diagonals of Squares and Hexagons



E = 1.4142 d
D = 1.1547 d

d	D	E	d	D	E	d	D	E
1/4	0.2886	0.3535	1 1/4	1.4434	1.7677	2 5/16	2.6702	3.2703
9/32	0.3247	0.3977	1 9/32	1.4794	1.8119	2 3/8	2.7424	3.3587
5/16	0.3608	0.4419	1 5/16	1.5155	1.8561	2 7/16	2.8145	3.4471
11/32	0.3968	0.4861	1 11/32	1.5516	1.9003	2 1/2	2.8867	3.5355
3/8	0.4329	0.5303	1 3/8	1.5877	1.9445	2 9/16	2.9583	3.6239
13/32	0.4690	0.5745	1 13/32	1.6238	1.9887	2 5/8	3.0311	3.7123
7/16	0.5051	0.6187	1 7/16	1.6598	2.0329	2 11/16	3.1032	3.8007
15/32	0.5412	0.6629	1 15/32	1.6959	2.0771	2 3/4	3.1754	3.8891
1/2	0.5773	0.7071	1 1/2	1.7320	2.1213	2 13/16	3.2476	3.9794
17/32	0.6133	0.7513	1 17/32	1.7681	2.1655	2 7/8	3.3197	4.0658
9/16	0.6494	0.7955	1 9/16	1.8042	2.2097	2 15/16	3.3919	4.1542
19/32	0.6855	0.8397	1 19/32	1.8403	2.2539	3	3.4641	4.2426
5/8	0.7216	0.8839	1 5/8	1.8764	2.2981	3 1/16	3.5362	4.3310
21/32	0.7576	0.9281	1 21/32	1.9124	2.3423	3 1/8	3.6084	4.4194
11/16	0.7937	0.9723	1 11/16	1.9485	2.3865	3 3/16	3.6806	4.5078
23/32	0.8298	1.0164	1 23/32	1.9846	2.4306	3 1/4	3.7527	4.5962
3/4	0.8659	1.0606	1 3/4	2.0207	2.4708	3 5/16	3.8249	4.6846
25/32	0.9020	1.1048	1 25/32	2.0568	2.5190	3 3/8	3.8971	4.7729
13/16	0.9380	1.1490	1 13/16	2.0929	2.5632	3 7/16	3.9692	4.8613
27/32	0.9741	1.1932	1 27/32	2.1289	2.6074	3 1/2	4.0414	4.9497
7/8	1.0102	1.2374	1 7/8	2.1650	2.6516	3 9/16	4.1136	5.0381
29/32	1.0463	1.2816	1 29/32	2.2011	2.6958	3 5/8	4.1857	5.1265
15/16	1.0824	1.3258	1 15/16	2.2372	2.7400	3 11/16	4.2579	5.2149
31/32	1.1184	1.3700	1 31/32	2.2733	2.7842	3 3/4	4.3301	5.3033
1	1.1547	1.4142	2	2.3094	2.8284	3 13/16	4.4023	5.3917
1 1/32	1.1907	1.4584	2 1/32	2.3453	2.8726	3 7/8	4.4744	5.4801
1 1/16	1.2268	1.5026	2 1/16	2.3815	2.9168	3 15/16	4.5466	5.5684
1 3/32	1.2629	1.5468	2 3/32	2.4176	2.9610	4	4.6188	5.6568
1 1/8	1.2990	1.5910	2 1/8	2.4537	3.0052	4 1/8	4.7631	5.8336
1 5/32	1.3351	1.6352	2 5/32	2.4898	3.0494	4 1/4	4.9074	6.0104
1 3/16	1.3712	1.6793	2 3/16	2.5259	3.0936	4 3/8	5.0518	6.1872
1 7/32	1.4073	1.7235	2 1/4	2.5981	3.1820	4 1/2	5.1961	6.3639

Table I-13.—Circles

Circumference of a circle = diameter X 3.1416

Diameter of a circle = circumference X .31831

Area of a circle = the square of the diameter X .7854

Surface of a ball (sphere) = the square of the diameter X 3.1416

Side of a square inscribed in a circle = diameter X .70711

Diameter of a circle to circumscribe a square = one side X 1.4142

Cubic inches (volume) in a ball = cube of the diameter X .5236

When doubled, the diameter of a pipe increases capacity four times

Radius of a circle X 6.283185 = circumference

Square of the circumference of a circle X .07958 area

1/2 circumference of a circle X 1/2 its diameter area

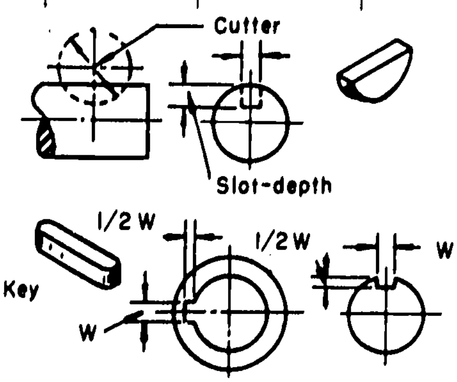
Circumference of a circle X .159155 = radius

Square root of the area of a circle X .56419 = radi

Square root of the area of a circle X 1.12838 diameter

Table I-14.—Keyway Dimensions

shaft dia	square keyways	Woodruff keyways*			
		key	thickness	cutter dia	slot depth
0.500	1/8 × 1/16	404	0.1250	0.500	0.1405
0.562	1/8 × 1/16	404	0.1250	0.500	0.1405
0.625	5/32 × 5/64	505	0.1562	0.625	0.1669
0.688	3/16 × 3/32	606	0.1875	0.750	0.2193
0.750	3/16 × 3/32	606	0.1875	0.750	0.2193
0.812	3/16 × 3/32	606	0.1875	0.750	0.2193
0.875	7/32 × 7/64	607	0.1875	0.875	0.2763
0.938	1/4 × 1/8	807	0.2500	0.875	0.2500
1.000	1/4 × 1/8	808	0.2500	1.000	0.3130
1.125	5/16 × 5/32	1009	0.3125	1.125	0.3228
1.250	5/16 × 5/32	1010	0.3125	1.250	0.3858
1.375	3/8 × 3/16	1210	0.3750	1.250	0.3595
1.500	3/8 × 3/16	1212	0.3750	1.500	0.4535
1.625	3/8 × 3/16	1212	0.3750	1.500	0.4535
1.750	7/16 × 7/32				
1.875	1/2 × 1/4				
2.000	1/2 × 1/4				
2.250	5/8 × 5/16				
2.500	5/8 × 5/16				
2.750	3/4 × 3/8				
3.000	3/4 × 3/8				
3.250	3/4 × 3/8				
3.500	7/8 × 7/16				
4.000	1 × 1/2				

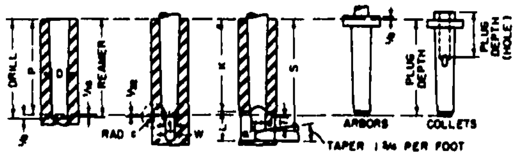


*The depth of a Woodruff keyway is measured from the edge of the slot.

Table I-15.—Tapers Per Foot and Corresponding Angles

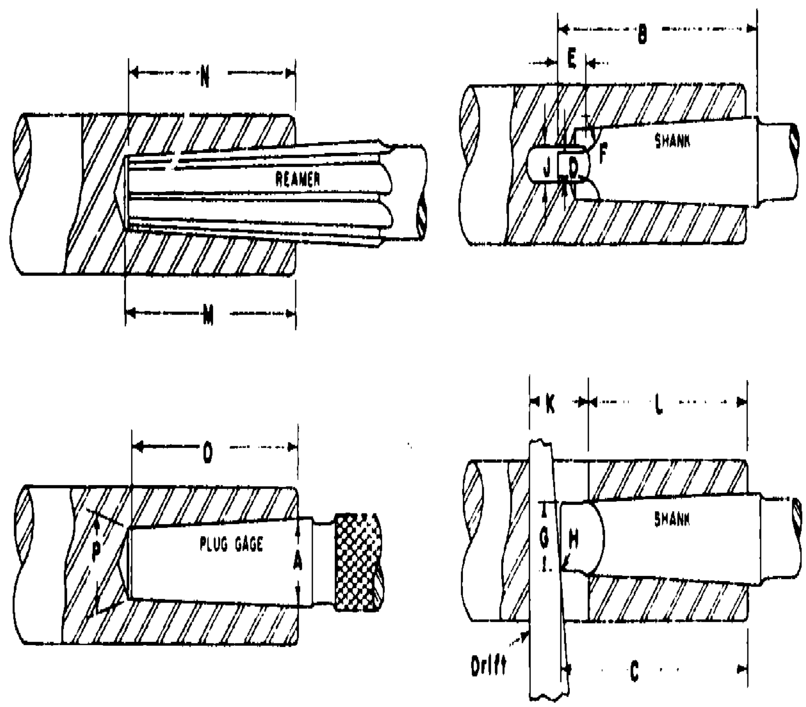
taper per foot	included angle			angle with center line			taper per foot	included angle			angle with center line			taper per foot	included angle			angle with center line		
	deg.	min.	sec.	deg.	min.	sec.		deg.	min.	sec.	deg.	min.	sec.		deg.	min.	sec.	deg.	min.	sec.
1/64	0	4	28	0	2	14	31/32	4	37	20	2	18	40	3¾	17	45	40	8	52	50
1/32	0	8	58	0	4	29	1	4	46	18	2	23	9	3⅞	18	20	34	9	10	17
1/16	0	17	54	0	8	57	1 1/16	5	4	12	2	32	6	4	18	55	28	9	27	44
3/32	0	26	52	0	13	26	1⅛	5	21	44	2	40	52	4⅛	19	30	18	9	45	9
⅛	0	35	48	0	17	54	1 3/16	5	39	54	2	49	57	4¼	20	5	2	10	2	31
5/32	0	44	44	0	22	22	1¼	5	57	48	2	58	54	4⅜	20	39	44	10	19	52
3/16	0	53	44	0	26	52	1 5/16	6	15	38	3	7	49	4½	21	14	2	10	37	1
7/32	1	2	34	0	31	17	1⅜	6	33	26	3	16	43	4⅝	21	48	54	10	54	27
¼	1	11	36	0	35	48	1 7/16	6	51	20	3	25	40	4¾	22	23	22	11	11	41
9/32	1	20	30	0	40	15	1½	7	9	10	3	34	35	4⅞	22	57	48	11	28	54
5/16	1	29	30	0	44	45	1 9/16	7	26	58	3	43	29	5	23	32	12	11	46	6
11/32	1	38	22	0	49	11	1⅝	7	44	48	3	52	24	5⅛	24	6	28	12	3	14
⅜	1	47	24	0	53	42	1 11/16	8	2	38	4	1	19	5¼	24	40	42	12	20	21
13/32	1	56	24	0	58	12	1¾	8	20	26	4	10	13	5⅜	25	14	48	12	37	24
7/16	2	5	18	1	2	39	1 13/16	8	38	16	4	19	8	5½	25	48	48	12	54	24
15/32	2	14	16	1	7	8	1⅞	8	56	2	4	28	1	5⅝	26	22	52	13	11	26
½	2	23	10	1	11	35	1 15/16	9	13	50	4	36	55	5¾	26	56	46	13	28	23
17/32	2	32	4	1	16	2	2	9	31	38	4	45	48	5⅞	27	30	34	13	45	17
9/16	2	41	4	1	20	32	2⅛	10	7	10	5	3	35	6	28	4	2	14	2	1
19/32	2	50	2	1	25	1	2¼	10	42	42	5	21	21	6⅛	28	37	58	14	18	59
⅝	2	59	2	1	29	31	2⅜	11	18	10	5	39	5	6¼	29	11	34	14	35	47
21/32	3	7	56	1	33	58	2½	11	53	36	5	56	48	6⅜	29	45	18	14	52	39
11/16	3	16	54	1	38	27	2⅝	12	29	2	6	14	31	6½	30	18	26	15	9	13
23/32	3	25	50	1	42	55	2¾	13	4	24	6	32	12	6⅝	30	51	48	15	25	54
¾	3	34	44	1	47	22	2⅞	13	39	42	6	49	51	6¾	31	25	2	15	42	31
25/32	3	43	44	1	51	52	3	14	15	0	7	7	30	6⅞	31	58	10	15	59	5
13/16	3	52	38	1	56	19	3⅛	14	50	14	7	25	7	7	32	31	12	16	15	36
27/32	4	1	36	2	0	48	3¼	15	25	24	7	42	42	7⅛	33	4	8	16	32	4
⅞	4	10	32	2	5	16	3⅜	16	0	34	8	0	17	7¼	33	36	40	16	48	20
29/32	4	19	34	2	9	47	3½	16	35	40	8	17	50	7⅜	34	9	50	17	4	55
15/16	4	28	24	2	14	12	3⅝	17	10	40	8	35	20							

Table I-16.—Tapers in Inches (Brown and Sharpe)



taper no.	taper per foot	plug dia. at small end, D	plug depth (P) B&S stand.	plug depth (P) for mill mach.	plug depth (P) misc.	keyway from end of spindle, K	shank depth, S	keyway length, L	keyway width, W	arbor tongue length, T	arbor tongue dia., d	arbor tongue thickness t	tongue circle radius, c	tongue radius, a	limit for tongue to project through test tool
1	0.50200	0.20000	15/16			15/16	1 5/16	1/8	0.135	3/16	0.170	1/8	3/16	0.030	0.003
2	0.50200	0.25000	1 1/16			1 11/64	1 1/2	1/2	0.188	1/4	0.220	5/32	3/16	0.030	0.003
3	0.50200	0.31250	1 1/2			1 15/32	1 7/8	5/8	0.197	5/16	0.282	3/16	3/16	0.040	0.003
					1 3/4	1 23/32	2 1/8	5/8	0.197	5/16	0.282	3/16	3/16	0.040	0.003
					2	1 31/32	2 3/8	5/8	0.197	5/16	0.282	3/16	3/16	0.040	0.003
4	0.50240	0.35000		1 1/4		1 11/64	1 21/32	11/16	0.228	11/32	0.320	7/32	5/16	0.050	0.003
			1 11/16			1 41/64	2 3/32	11/16	0.228	11/32	0.320	7/32	5/16	0.050	0.003
5	0.50180	0.45000		1 3/4		1 11/16	2 1/16	3/4	0.260	5/8	0.420	1/4	5/16	0.060	0.003
					2	1 15/16	2 5/16	3/4	0.260	5/8	0.420	1/4	5/16	0.060	0.003
			2 1/8			2 1/16	2 9/16	3/4	0.260	5/8	0.420	1/4	5/16	0.060	0.003
6	0.50329	0.50000	2 3/8			2 19/64	2 7/8	7/8	0.291	7/16	0.460	9/32	7/16	0.060	0.005
7	0.50147	0.60000			2 1/2	2 11/32	3 1/32	15/16	0.322	15/32	0.560	5/16	5/8	0.070	0.005
			2 7/8			2 23/32	3 11/32	15/16	0.322	15/32	0.560	5/16	5/8	0.070	0.005
				3		2 29/32	3 11/32	15/16	0.322	15/32	0.560	5/16	5/8	0.070	0.005
8	0.50100	0.75000	3 9/16			3 29/64	4 1/8	1	0.353	1/2	0.710	11/32	5/8	0.080	0.005
9	0.50085	0.90010		4		3 7/8	4 5/8	1 1/8	0.385	9/16	0.860	3/8	7/16	0.100	0.005
			4 1/4			4 1/8	4 7/8	1 1/8	0.385	9/16	0.860	3/8	7/16	0.100	0.005
10	0.51612	1.04465	5			4 27/32	5 23/32	1 5/16	0.447	21/32	1.010	7/16	7/16	0.110	0.005
				5 11/16		5 11/64	6 11/32	1 5/16	0.447	21/32	1.010	7/16	7/16	0.110	0.005
					6 7/32	6 1/16	6 15/16	1 5/16	0.447	21/32	1.010	7/16	7/16	0.110	0.005
11	0.50100	1.24995	5 15/16			5 25/32	6 21/32	1 5/16	0.447	21/32	1.210	7/16	1/2	0.130	0.005
				6 3/4		[illegible]	7 15/32	1 5/16	0.447	21/32	1.210	7/16	1/2	0.130	0.005
12	0.49973	1.50010	7 1/8	7 1/8		8 15/16	7 15/16	1 1/2	0.510	3/4	1.460	1/2	1/2	0.150	0.005
					8 3/4										
13	0.50020	1.75005	7 3/4			7 9/16	8 9/16	1 1/2	0.510	3/4	1.710	1/2	5/8	0.170	0.010
14	0.50000	2.00000	8 1/4	8 1/4		8 1/32	9 5/32	1 11/16	0.572	27/32	1.960	9/16	3/4	0.190	0.010
15	0.50000	2.25000	8 3/4			8 17/32	9 21/32	1 11/16	0.572	27/32	2.210	9/16	7/8	0.210	0.010
16	0.50000	2.50000	9 1/4			9	10 1/4	1 7/8	0.635	15/16	2.450	5/8	1	0.230	0.010
17	0.50000	2.75000	9 3/4												
18	0.50000	3.00000	10 1/4												

Table I-17.—American Standard Tapers (Morse)

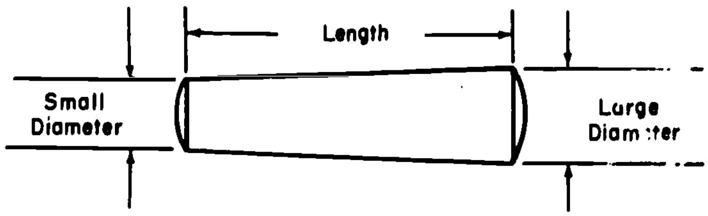


	diameter		shank					tang					tang slot					
taper no.	plug at small end, P	gage line, A	whole length, B	depth, C	drilled hole depth, M	reamed hole depth, N	stan. plug depth, O	thickness, D	length, E	radius, F	dia., G	radius, H	width, J	length, K	end of socket to tang slot, L	taper per inch	taper per foot	drift no.
0	0.252	0.356	2 11/32	2 7/32	2 1/16	2 1/32	2	0.156	1/4	5/32	15/64	3/64	0.166	9/16	1 15/16	0.052000	0.62400	0
1	0.369	0.475	2 9/16	2 7/16	2 3/16	2 5/32	2 1/8	0.203	3/8	3/16	11/32	3/64	0.213	3/4	2 1/16	0.049882	0.59858	1
2	0.572	0.700	3 1/8	2 15/16	2 21/32	2 39/64	2 9/16	0.250	7/16	1/4	17/32	1/16	0.260	7/8	2 1/2	0.049951	0.59941	2
3	0.778	0.938	3 7/8	3 11/16	3 5/16	3 1/4	3 3/16	0.312	9/16	9/32	23/32	5/64	0.322	1 3/16	3 1/16	0.050196	0.60235	3
4	1.020	1.231	4 7/8	4 5/8	4 3/16	4 1/8	4 1/16	0.469	5/8	5/16	31/32	3/32	0.479	1 1/4	3 7/8	0.051938	0.62326	4
5	1.475	1.748	6 1/8	5 7/8	5 5/16	5 1/4	5 3/16	0.625	3/4	3/8	1 13/32	1/8	0.635	1 1/2	4 15/16	0.052626	0.63151	5
6	2.116	2.494	8 9/16	8 1/4	7 13/32	7 21/64	7 1/4	0.750	1 1/8	1/2	2	5/32	0.760	1 3/4	7	0.052137	0.62565	5†
7	2.750	3.270	11 5/8	11 1/4	10 5/32	10 5/64	10	1.125	1 3/8	3/4	2 5/8	3/16	1.135	2 5/8	9 1/2	0.052000	0.62400	

* Dimensions agree essentially with those of the American Standard on Machine Tapers

† The No. 5 drift will also eject No. 6 taper shank tools

Table I-18.—Drill Sizes For Taper Pins



Drill size should be approximately 0.005 smaller than small diameter

Taper = 1/4 in. per foot

Small diameter = large diameter − length X 0.02083

NUMBER	7/0	6/0	5/0	4/0	3/0	2/0	0	1	2	3	4	5	6	7	8	9	10	11	
DIAMETER AT LARGE END	0.0625	0.078	0.094	0.109	0.125	0.141	0.156	0.172	0.193	0.219	0.250	0.289	0.341	0.409	0.492	0.591	0.707	0.857	
LENGTH	DIAMETER OF SMALL END OF PIN AND DRILL SIZE																		LENGTH
¼	0.0573 54	0.0728 50	....	....	....	....	....	....	....	....	....			....	....	....	....	....	¼
⅜	0.0547 55	0.0702 51	0.0862 45	....	....	....			....	....				....		....	....	....	⅜
½	0.0521 56	0.0676 52	0.0836 46	0.0986 41	0.1146 34	0.1306 30	0.1456 9/64	0.1616 5/32	..				....					....	½
⅝	0.0495 56	0.0650 52	0.0810 5/64	0.0960 3/32	0.1120 7/64	0.1280 ⅛	0.1430 9/64	0.1590 23				....		....		....		....	⅝
¾	0.0469 56	0.0624 53	0.0784 48	0.0934 43	0.1094 36	0.1254 31	0.1404 28	0.1564 24	0.1774 11/64	0.2034 6	0.2344 1				....	..	....	....	¾
⅞	....	0.0598 54	0.0758 48	0.0908 43	0.1068 37	0.1228 31	0.1378 28	0.1538 26	0.1748 18	0.2008 8	0.2318 1							....	⅞
1	....	0.0572 54	0.0732 50	0.0882 44	0.1042 38	0.1202 32	0.1352 30	0.1512 26	0.1722 18	0.1982 10	0.2292 2	0.2682 G	0.3202 O	..				.....	1
1⅛	....		..	0.0856 45	0.1016 38	0.1176 33	0.1326 30	0.1486 27	0.1696 18	0.1956 11	0.2266 2	0.2656 G	0.3176 5/16		.....	..	..	.....	1⅛
1¼	..		.....	0.0830 48	0.0990 41	0.1150 33	0.1300 ⅛	0.1460 9/64	0.1670 [illegible]	0.1930 3/16	0.2240 7/32	0.2630 F	0.3150 N	0.3830 ⅜		..	....	.....	1¼
1⅜			..	....	0.0964 3/32	0.1124 7/64	0.1274 ⅛	0.1434 9/64	0.1644 20	0.1904 3/16	0.2214 3	0.2604 F	0.3124 N	0.3804 ⅜			..	.....	1⅜
1½				..	0.0938 43	0.1098 36	0.1248 31	0.1408 28	0.1618 5/32	0.1878 14	0.2188 3	0.2578 ¼	0.3098 N	0.3778 U	0.4608 29/64	..	....	...	1½
1¾		....	....	..	....	0.1045 36	0.1195 32	0.1355 30	0.1565 24	0.1825 15	0.2135 4	0.2525 D	0.3045 19/64	0.3725 U	0.4555 7/16	....	....	....	1¾
2	....	....				0.0993 41	0.1143 34	0.1303 ⅛	0.1513 25	0.1773 11/64	0.2083 13/64	0.2473 C	0.2993 M	0.3673 23/64	0.4503 7/16	0.5493 17/32	....	.....	2
2¼	...	....	....	....			..	0.1251 31	0.1461 27	0.1721 18	0.2031 6	0.2421 B	0.2941 L	0.3621 T	0.4451 7/16	0.5441 17/32	.....	....	2¼
2½	.....	....	..		...			0.1200 32	0.1409 28	0.1669 20	0.1979 10	0.2369 15/64	0.2889 9/32	0.3569 S	0.4399 27/64	0.5389 17/32	0.6549 41/64	....	2½
2¾	......	.....	...		....	....			0.1357 30	0.1617 5/32	0.1927 3/16	0.2317 1	0.2837 J	0.3517 11/32	0.4347 27/64	0.5337 33/64	0.6497 41/64	....	2¾

546

Table I-18.—Drill Sizes For Taper Pins—Continued

NUMBER	7/0	6/0	5/0	4/0	3/0	2/0	0	1	2	3	4	5	6	7	8	9	10	11	
DIAMETER AT LARGE END	0.0625	0.078	0.094	0.109	0.125	0.141	0.156	0.172	0.193	0.219	0.250	0.289	0.341	0.409	0.492	0.591	0.707	0.857	¼
LENGTH	DIAMETER OF SMALL END OF PIN AND DRILL SIZE																		LENGTH
3	...	...	...	...	...	...	...	...	0.1305 30	0.1565 24	0.1875 14	0.2265 2	0.2785 1	0.3465 R	0.4295 27/64	0.5285 33/64	0.6435 ⅝	0.7975 25/32	3
3¼	...	...	...	...	...	...	...	...	...	...	0.1823 16	0.2213 7/32	0.2733 17/64	0.3413 Q	0.4243 Z	0.5233 33/64	0.6383 ⅝	0.7923 25/32	3¼
3½	...	...	...	...	...	...	...	...	...	...	0.1771 11/64	0.2161 3	0.2681 G	0.3361 Q	0.4191 13/32	0.5181 ½	0.6331 ⅝	0.7871 25/32	3½
3¾	...	...	...	...	...	...	...	...	...	...	...	...	0.2629 F	0.3309 21/64	0.4139 13/32	0.5129 ½	0.6279 39/64	0.7819 49/64	3¾
4	...	...	...	...	...	...	...	...	...	...	...	...	0.2577 ¼	0.3257 P	0.4087 Y	0.5077 ½	0.6227 39/64	0.7767 49/64	4
4¼	...	...	...	...	...	...	...	...	...	...	...	...	...	0.3205 O	0.4035 X	0.5025 31/64	0.6175 39/64	0.7715 49/64	4¼
4½	...	...	...	...	...	...	...	...	...	...	...	...	...	0.3153 5/16	0.3983 25/64	0.4973 31/64	0.6123 19/32	0.7663 ¾	4½
4¾	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.3931 W	0.4921 31/64	0.6071 19/32	0.7611 ¾	4¾
5	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.3879 ⅜	0.4869 15/32	0.6019 19/32	0.7559 ¾	5
5¼	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.4817 15/32	0.5967 37/64	0.7507 47/64	5¼
5½	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.4765 15/32	0.5915 37/64	0.7455 47/64	5½
5¾	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.4713 29/64	0.5863 37/64	0.7403 47/64	5¾
6	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.4660 29/64	0.5810 9/16	0.7350 23/32	6
6¼	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.5759 9/16	0.7298 23/32	6¼
6½	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.5706 9/16	0.7246 23/32	6½
6¾	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.5654 35/64	0.7194 45/64	6¾
7	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.5602 35/64	0.7142 45/64	7
7¼	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.7090 45/64	7¼
7½	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	...	0.7038 11/16	7½

Table I-19.—Grinding of Twist Drills

(Do Not Dip High-Speed Drills In Water)

Drilling different grades of materials sometimes requires modification of the commercial 118° drill point for maximum results. Hard materials require a blunter point with the more acute angle for softer materials.

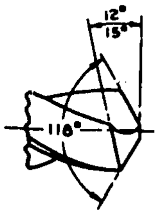


Fig.I-19-1

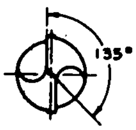


Fig.I-19-2

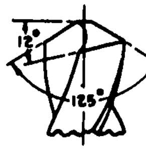


Fig.I-19-3

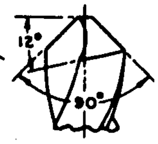


Fig.I-19-4

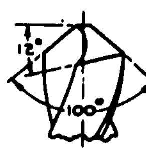


Fig.I-19-5

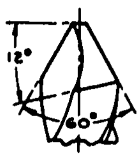


Fig.I-19-6

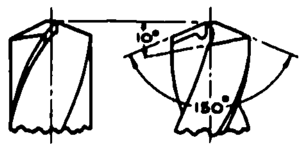


Fig.I-19-7

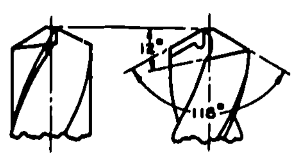


Fig.I-19-8

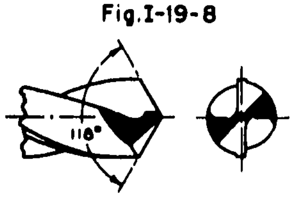


Fig.I-19-9

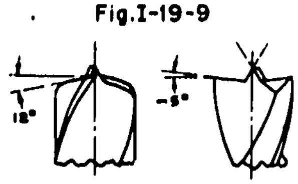


Fig.I-19-10

ANGLE OF POINTS

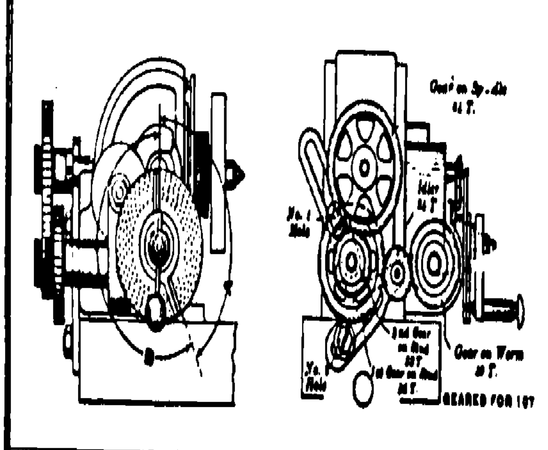
		Point
Fig. I-19-1 and I-19-2	Average Class of Work	118° included angle 12° to 15° lip clearance
Fig. I-19-3	Alloy Steels, Monel Metal, Stainless Steel, Heat Treated Steels, Drop Forgings (Automobile Connecting Rods) Brinell Hardness No. 240	125° included angle 10° to 12° lip clearance
Fig. I-19-4	Soft and Medium Cast Iron, Aluminum, Marble, Slate, Plastics, Wood, Hard Rubber, Bakelite, Fibre	90° to 130° included angle 12° lip clearance Flat cutting lip for marble
Fig. I-19-5	Copper, Soft and Medium Hard Brass	100° to 118° included angle 12° to 15° lip clearance
	Magnesium Alloys	60° to 118° included angle 15° lip clearance Slightly flat face of cutting lips reducing rake angle to 5°
Fig. I-19-6	Wood, Rubber, Bakelite, Fibre, Aluminum, Die Castings, Plastics	60° included angle 12° to 15° lip clearance
Fig. I-19-7	Steel 7% to 13% Manganese, Tough Alloy Steels, Armor Plate and hard materials	150° included angle 7° to 10° lip clearance Slightly flat face of cutting lips
Fig. I-19-8	Brass, Soft Bronze	118° included angle 12° to 15° lip clearance Slightly flat face of cutting lips
Fig. I-19-9	Crankshafts, Deep Holes in Soft Steel, Hard Steel, Cast Iron, Nickel and Manganese Alloys	118° included angle Chisel Point 9° lip clearance
Fig. I-19-10	Thin Sheet Metal, Copper, Fibre, Plastics, Wood	-5° to +12° lip angles For drills over 1/4″ diamoter make angle of bit point to suit work

Table I-20.—Allowances For Fits

(Grinding Limits for Cylindrical Parts)

Diameter (inches)	Limits (inches)
Running Fits -- Ordinary Speed	
Up to 1/2	− 0.00025 to − 0.00075
1/2 to 1	− 0.00075 to − 0.0015
1 to 2	− 0.0015 to − 0.0025
2 to 3-1/2	− 0.0025 to − 0.0035
3-1/2 to 6	− 0.0035 to − 0.005
Running Fits -- High-Speed, Heavy Pressure and Rocker Shafts	
Up to 1/2	− 0.0005 to − 0.001
1/2 to 1	− 0.001 to − 0.002
1 to 2	− 0.002 to − 0.003
2 to 3-1/2	− 0.003 to − 0.0045
3-1/2 to 6	− 0.0045 to − 0.0065
Sliding Fits	
Up to 1/2	− 0.00025 to − 0.0005
1/2 to 1	− 0.0005 to − 0.001
1 to 2	− 0.001 to − 0.002
2 to 3-1/2	− 0.002 to − 0.0035
3-1/2 to 6	− 0.003 to − 0.005

Diameter (inches)	Limits (inches)
Driving Fits -- Ordinary	
Up to 1/2	+ 0.00025 to + 0.0005
1/2 to 1	+ 0.001 to + 0.002
1 to 2	+ 0.002 to + 0.003
2 to 3-1/2	+ 0.003 to + 0.004
3-1/2 to 6	+ 0.004 to + 0.005
Forced Fits	
Up to 1/2	+ 0.00075 to + 0.0015
1/2 to 1	+ 0.0015 to + 0.0025
1 to 2	+ 0.0025 to + 0.004
2 to 3-1/2	+ 0.004 to + 0.006
3-1/2 to 6	+ 0.006 to + 0.009
Driving Fits -- For such Pieces as are Required to be Readily Taken Apart	
Up to 1/2	+ 0 to + 0.00025
1/2 to 1	+ 0.00025 to + 0.0005
1-1/2 to 2	+ 0.0005 to + 0.00075
2 to 3-1/2	+ 0.00075 to + 0.001
3-1/2 to 6	+ 0.001 to + 0.0015



Number of divisions	Index circle	No. of turns of index	Graduation
2	Any	20	
3	39	13 13/39	65
	33	13 11/33	65
	18	13 6/18	65
4	Any	10	
5	Any	8	
6	39	6 26/39	132
	33	6 22/33	132
	18	6 12/18	132
7	49	5 35/49	140
	21	5 15/21	142
8	Any	5	
9	27	4 12/27	88
	18	4 8/18	87
10	Any	4	
11	33	3 21/33	126
12	39	3 13/39	65
	33	3 11/33	65
	18	3 6/18	65

Number of divisions	Index circle	No. of turns of index	Graduation
13	39	3 3/39	14
14	49	2 42/49	169
	21	2 18/21	170
15	39	2 26/39	132
	33	2 22/33	132
	18	2 12/18	132
16	20	2 10/20	98
17	17	2 6/17	69
18	27	2 6/27	43
	18	2 4/18	43
19	19	2 2/19	19
20	Any	2	
21	21	1 19/21	18*
22	33	1 27/33	161
23	23	1 17/23	147
24	39	1 26/39	132
	33	1 22/33	132
	18	1 12/18	132
25	20	1 12/20	118

Number of divisions	Index circle	No. of turns of index	Graduation
26	39	1 21/39	106
27	27	1 13/27	95
28	49	1 21/49	83
	21	1 9/21	85
29	29	1 11/29	75
30	39	1 13/39	65
	33	1 11/33	65
	18	1 6/18	65
31	31	1 9/31	56
32	20	1 5/20	48
33	33	1 7/33	41
34	17	1 3/17	33
35	49	1 7/49	26
	21	1 3/21	28
36	27	1 3/27	21
	18	1 2/18	21
37	37	1 3/37	15
38	19	1 1/19	9
39	39	1 1/39	3

Number of divisions	Index circle	No. of turns of index	Graduation
40	Any	1	
41	41	40/41	3*
42	21	20/21	9*
43	43	40/43	12*
44	33	30/33	17*
45	27	24/27	21*
	18	16/18	21*
46	23	20/23	172
47	47	40/47	168
48	18	15/18	165
49	49	40/49	161
50	20	16/20	158

Number of divisions	Index circle	No. of turns of index	Graduation	Gear on worm	No. 1 hole: 1st gear on stud	No. 1 hole: 2nd gear on stud	Gear on spindle	Idlers: No. 1 hole	Idlers: No. 2 hole
51	17	14/17	33*	24			48	24	44
52	39	30/39	152						
53	49	35/49	140	56	40	24	72		
	21	15/21	142	56	40	24	72		
54	27	20/27	147						
55	33	24/33	144						
56	49	35/49	140						
	21	15/21	142						
57	49	35/49	140	56			40	24	44
	21	15/21	142	56			40	24	44
58	29	20/29	136						
59	39	26/39	132	48			32	44	
	33	22/33	132	48			32	44	
	18	12/18	132	48			32	44	
60	39	26/39	132						
	33	22/33	132						
	18	12/18	132						
61	39	26/39	132	48			32	24	44
	33	22/33	132	48			32	24	44
	18	12/18	132	48			32	24	44
62	31	20/31	127						
63	39	26/39	132	24			48	24	44
	33	22/33	132	24			48	24	44
	18	12/18	132	24			48	24	44
64	16	10/16	123						
65	39	24/39	121						
66	33	20/33	120						
67	49	28/49	112	28			48	44	
	21	12/21	113	28			48	44	
68	17	10/17	116						

Number of divisions	Index circle	No. of turns of index	Graduation	Gear on worm	No. 1 hole: 1st gear on stud	No. 1 hole: 2nd gear on stud	Gear on spindle	Idlers: No. 1 hole	Idlers: No. 2 hole
69	20	12/20	118	40			56	24	44
70	49	28/49	112						
	21	12/21	113						
71	27	15/27	110	72			40	24	
	18	10/18	109	72			40	24	
72	27	15/27	110						
	18	10/18	109						
73	49	28/49	112	28			48	24	44
	21	12/21	113	28			48	24	44
74	37	20/37	107						
75	15	8/15	105						
76	19	10/19	103						
77	20	10/20	98	32			48	44	
78	39	20/39	101						
79	20	10/20	98	48			24	44	
80	20	10/20	98						
81	20	10/20	98	48			24	24	44
82	41	20/41	96						
83	20	10/20	98	32			48	24	44
84	21	10/21	94						
85	17	8/17	92						
86	43	20/43	91						
87	15	7/15	92	40			24	24	44
88	33	15/33	89						
89	27	12/27	88	72			32	44	
	18	8/18	87	72			32	44	
90	27	12/27	88						
	18	8/18	87						
91	39	18/39	91	24			48	24	44
92	23	10/23	86						

Graduations in table are for setting of the arms of sector when index crank moves through arc "A", except when marked * when the index crank moves through arc "B".

Graduations in table are for setting of the arms of sector when index crank moves through arc "A", except when marked * when the index crank moves through arc "B".

Table I-22.—Machinability Ratings/Other Properties of Various Metals

SAE Number	AISI Number	Tensile Strength psi	Yield Point psi	Elongation in 2 in. (%)	Reduction In Area (%)	Hardness Brinell	Machinability Rating (%)
Carbon Steels							
1015	C1015	65,000	40,000	32	65	137	50
1020	C1020	67,000	45,000	32	65	137	52
x1020	C1022	69,000	47,000	30	58	143	62
1025	C1025	70,000	41,000	31	58	130	58
1030	C1030	75,000	46,000	30	56	138	60
1035	C1035	88,000	55,000	30	56	175	60
1040	C1040	93,000	58,000	27	52	190	60
1045	C1045	99,000	60,000	24	47	200	55
1095	C1095	100,000	60,000	23	47	201	45
Free-Cutting Steels							
x1113	B1113	83,000	73,000	15	45	193	120-140
1112	B1112	67,000	40,000	27	47	140	100
	C1120	69,000	36,000	32	55	117	80
Manganese Steels							
x1314		71,000	45,000	28	52	135	94
x1335	A1335	95,000	60,000	20	35	185	70
Nickel Steels							
2315	A2317	85,000	56,000	29	60	163	50
2330	A2330	98,000	65,000	25	50	207	45
2340	A2340	110,000	30,000	22	47	225	40
2345	A2345	108,000	75,000	23	46	235	50
Nickel-Chromium Steels							
3120	A3120	75,000	60,000	30	65	151	50
3130	A3130	100,000	72,000	24	55	212	45
3140	A3140	96,000	64,000	26	56	195	57
3150	A3150	104,000	73,000	19	51	229	50
3250		107,000	75,000	24	55	217	44
Molybdenum Steels							
4119		91,000	52,000	28	62	179	60
x4130	A4130	89,000	60,000	32	65	179	58
4140	A4140	90,000	63,000	27	58	187	56
4150	A4150	105,000	71,000	21	54	220	54
x4340	A4340	115,000	95 000	18	45	235	58
4615	A4615	82 000	55,000	30	61	167	58
4640	A4640	100,000	87,000	21	50	201	60
4815	A4815	105,000	73,000	24	58	212	55

Table I-22.—Machinability Ratings/Other Properties of Various Metals—Continued

SAE Number	AISI Number	Tensile Strength psi	Yield Point psi	Elongation in 2 in. (%)	Reduction in Area (%)	Hardness Brinell	Machinability Rating (%)
Chromium Steels							
5120	A5120	73,000	55,000	32	67	143	50
5140	A5140					174-229	60
52100	E52101	109,000	80,000	25	57	235	45
Chromium-Vanadium Steels							
6120	A6120					179-217	50
6150	A6150	103,000	70,000	27	51	217	50
Other Alloys and Metals							
Aluminum (11S)		49,000	42,000	14		95	300-2,000
Brass, Leaded		55,000	45,000	32		RF 100	150-600
Brass, Red or Yellow		25-35,000	15-30,000			40-55	200
Bronze Lead-Bearing		22-32,000	8-20,000	3-16	5-18	30-65	200-500
Cast Iron, Hard		45,000				220-240	50
Cast Iron, Medium		40,000				193-220	65
Cast Iron, Soft		30,000				160-193	80
Cast Steel (0.35 C)		86,000	55,000	25	34	170-212	70
Copper (F.M.)		35,000	33,000	34		RF 85	65
Ingot Iron		41-45,000	18-25,000	45	70	101-131	50
Low-Alloy, High-Strength Steel		98,000	65,000	18	34	187	80
Magnesium Alloys							500-2,000
Malleable Iron							
Standard		53-60,000	35-40,000	18-25		110-145	120
Pearlitic		80,000	55,000	14		180-200	90
Pearlitic		97,000	75,000	4		227	80
Stainless Steel (12% Cr F.M.)		120,000	86,000	23	64	207	70
18-8 Stainless Steel (Type 303 F.M.)		80,000	30,000	60	75	150	45
18-8 Stainless Steel (Type 304)		80,000	40,000	65	70	150	25

Properties for wrought materials are for hot-rolled condition.

Properties in this table are only a rough guide to the machining of various common steels and alloys.

Table I-23.—Selection Chart for Cutting Fluids

		Ferrous Metals				Nonferrous Metals	
	Group	I	II	III	IV	V	VI
	Machinability	Over 70 %	50-70 %	40-50 %	Under 40 %	Over 100 %	Under 100 %
	Materials	Low-carbon Steels, High-carbon Steels: Malleable Iron, Cast Steel, Stainless Iron	Low-carbon Steels, High-carbon Steels: Cast Iron	Stainless Steels: Ingot Iron, Wrought Iron	Stainless Steels: Tool Steels, High-speed Steels	Aluminum and Alloys, Brasses and Bronzes: Magnesium and Alloys, Zinc	Aluminum and Alloys, Brasses and Bronzes: Copper, Nickel, Inconel, Monel
Severity	Type of Machining Operation						
(Greatest) 1.	Broaching, internal	Em. Sul.	Sul. Em.	Sul. Em.	Sul. Em.	MO. Em.	Sul. ML.
2.	Broaching; surface	Em. Sul.	Em. Sul.	Sul. Em.	Sul. Em.	MO. Em.	Sul. ML.
2.	Threading; pipe	Sul.	Sul. ML	Sul.	Sul.		Sul.
3.	Tapping; plain	Sul.	Sul.	Sul.	Sul.	Em. Dry	Sul. ML.
3.	Threading; plain	Sul.	Sul.	Sul.	Sul.	Em. Sul.	Sul.
4.	Gear shaving	Sul. L.	Sul. L.	Sul. L.	Sul. L.		
4.	Reaming; plain	ML. Sul.	ML. Sul.	ML. Sul.	ML. Sul.	ML. MO. Em.	ML. MO. Sul.
4.	Gear cutting	Sul. ML. Em.	Sul.	Sul.	Sul. ML.		Sul. ML.
5.	Drilling; deep	Em. ML.	Em. Sul.	Sul.	Sul.	MO. ML. Em.	Sul. ML.
6.	Milling; plain	Em. ML. Sul.	Em.	Em.	Sul.	Em. MO. Dry	Sul. Em.
6.	Milling; multiple cutter	ML.	Sul.	Sul.	Sul. ML.	Em. MO. Dry	Sul. Em.
7.	Boring; multiple head	Sul. Em.	Sul. HDS	Sul. HDS	Sul. Em.	K. Dry Em.	Sul. Em.
7.	Multiple-spindle automatic screw machines and turret lathes: drilling, forming, turning, reaming, cutting-off, tapping, threading	Sul. Em. ML.	Sul. Em. ML.	Sul. Em. ML. HDS	Sul. ML. Em. HDS	Em. Dry ML.	Sul.
8.	High speed, light feed automatic screw machines: drilling, forming, tapping, threading, turning, reaming, box milling, cutting off	Sul. Em. ML.	Sul. Em. ML.	Sul. Em. ML.	Sul. ML. Em.	Em. Dry ML.	Sul.
9.	Drilling	Em.	Em.	Em.	Em. Sul.	Em. Dry	Em.
9.	Planing, shaping	Em. Sul. ML.	Em. Sul. ML.	Sul. Em	Em. Sul.	Em. Dry	Em.
9.	Turning; single point tool, form tools	Em. Sul. ML.	Em. Sul. ML.	Em. Sul. ML.	Em. Sul. ML.	Em. Dry ML.	Em. Sul.
(Least) 10.	Sawing; circular, hack	Sul. ML. Em.	Sul. Em. ML.	Sul. Em. ML.	Sul. Em. ML.	Dry MO. Em.	Sul. Em. ML.
	Grinding; 1. plain	Em.	Em.	Em.	Em.	Em.	Em.
	2. form (thread, etc.)	Sul.	Sul.	Sul.	Sul.	MO. Sul.	Sul.

Key

K. = Kerosene
L. = Lard Oil
MO. = Mineral oils
ML. = Mineral-lard oils
Sul. = Sulphurized oils, with or without chlorine
Em. = Soluble or emulsifiable oils and compounds
Dry = No cutting fluid needed
HDS = Heavy duty soluble oil

APPENDIX II

FORMULAS FOR SPUR GEARING

Having	To Get	Rule	Formula
Diametral pitch	Circular pitch	Divide 3.1416 by the diametral pitch	$CP = \frac{3.1416}{DP}$
Pitch diameter and number of teeth.	Circular pitch	Divide the pitch diameter by the product of 0.3183 and the number of teeth.	$PD = \frac{OD}{0.3183\ NT}$
Outside diameter and number of teeth.	Circular pitch	Divide the outside diameter by the product of 0.3183 and the number of teeth plus 2.	$CP = \frac{OD}{0.3183\ NT + 2}$
Number of teeth and circular pitch.	Pitch diameter	The product of the number of teeth, the circular pitch, and 0.3183.	$PD = 0.3183\ CP\ NT$
Number of teeth and outside diameter.	Pitch diameter	Divide the product of the number of teeth and the outside diameter by the number of teeth plus 2.	$PD = \frac{NT\ OD}{NT + 2}$
Outside diameter and circular pitch.	Pitch diameter	Subtract from the outside diameter the product of the circular pitch and 0.6366.	$PD = OD - 0.6366\ CP$
Addendum and number of teeth.	Pitch diameter	Multiply the number of teeth by the addendum.	$PD = NT\ ADD$
Number of teeth and circular pitch.	Outside diameter	The product of the number of teeth plus 2, the circular pitch, and 0.3183.	$OD = (NT + 2)\ 0.3183\ CP$

Having	To Get	Rule	Formula
Pitch diameter and circular pitch.	Outside diameter--	Add to the pitch diameter the product of the circular pitch and 0.6366.	$OD = PD + 0.6366\ CP$
Number of teeth and addendum.	Outside diameter--	Multiply the addendum by the number of teeth plus 2.	$OD = (NT + 2) ADD$
Pitch diameter and circular pitch.	Number of teeth --	Divide the product of the pitch diameter and 3.1416 by the circular pitch.	$NT = \frac{3.1416\ PD}{CP}$
Circular pitch ------	Chordal Thickness --	One half the circular pitch.	$(t_c) = \frac{CP}{2}$
Circular pitch ------	Addendum -------	Multiply the circular pitch by 0.3183.	$ADD = 0.3183\ CP$
Circular pitch ------	Working depth ----	Multiply the circular pitch by 0.6366.	$WKD = 0.6366\ CP$
Circular pitch ------	Whole depth -----	Multiply the circular pitch by 0.6866.	$WD = 0.6866\ CP$
Circular pitch ------	Clearance ------	Multiply the circular pitch by 0.05.	$CL = 0.05\ CP$
Circular pitch ------	Diametral pitch--	Divide 3.1416 by the circular pitch.	$DP = \frac{3.1416}{CP}$
Pitch diameter and number of teeth.	Diametral pitch--	Divide the number of teeth by the pitch diameter.	$DP = \frac{NT}{PD}$
Pitch diameter of gear and pinion.	Center distance ---	Add pitch diameter of gear (PD_g) to pitch diameter of pinion (PD_p) divide by 2.	$C = \frac{PD_g + PD_p}{2}$
Outside diameter and number of teeth.	Diametral pitch---	Divide the number of teeth plus 2 by the outside diameter.	$DP = \frac{NT + 2}{OD}$

Appendix II—FORMULAS FOR SPUR GEARING

Having	To Get	Rule	Formula
Number of teeth and diametral pitch.	Pitch diameter --	Divide the number of teeth by the diametral pitch.	$PD = \frac{NT}{DP}$
Outside diameter and diametral pitch.	Pitch diameter --	Subtract from the outside diameter the quotient of 2 divided by the diametral pitch.	$PD = OD - \frac{2}{DP}$
Number of teeth and diametral pitch.	Outside diameter---	Divide the number of teeth plus 2 by the diametral pitch.	$OD = \frac{NT+2}{DP}$
Pitch diameter and diametral pitch.	Outside diameter	Add to the pitch diameter the quotient of 2 divided by the diametral pitch.	$OD = PD + \frac{2}{DP}$
Pitch diameter and number of teeth.	Outside diameter	Divide the number of teeth plus 2 by the quotient of the number of teeth divided by pitch diameter.	$OD = NT + 2 \div \frac{NT}{PD}$
Pitch diameter and diametral pitch.	Number of teeth	Multiply the pitch diameter by the diametral pitch.	$NT = PD\ DP$
Outside diameter and the diametral pitch.	Number of teeth	Multiply the outside diameter by the diametral pitch and subtract 2.	$NT = OD\ DP - 2$
Diametral pitch	Chordal Thickness --	Divide 1.5708 by the diametral pitch.	$t_c = \frac{1.5708}{DP}$
Diametral pitch	Addendum ---- ---	Divide 1 by the diametral pitch.	$ADD = \frac{1}{DP}$
Diametral pitch	Working depth----	Divide 2 by the diametral pitch.	$WKD = \frac{2}{DP}$
Diametral pitch	Whole depth ------	Divide 2.157 by the diametral pitch.	$WD = \frac{2.157}{DP}$
Diametral pitch	Clearance -------	Divide 0.157 by the diametral pitch.	$CL = \frac{0.157}{DP}$

APPENDIX III

DERIVATION OF FORMULAS FOR DIAMETRAL PITCH SYSTEM

1. TOOTH ELEMENTS based on a #1 diametral pitch gear (fig. III-A).

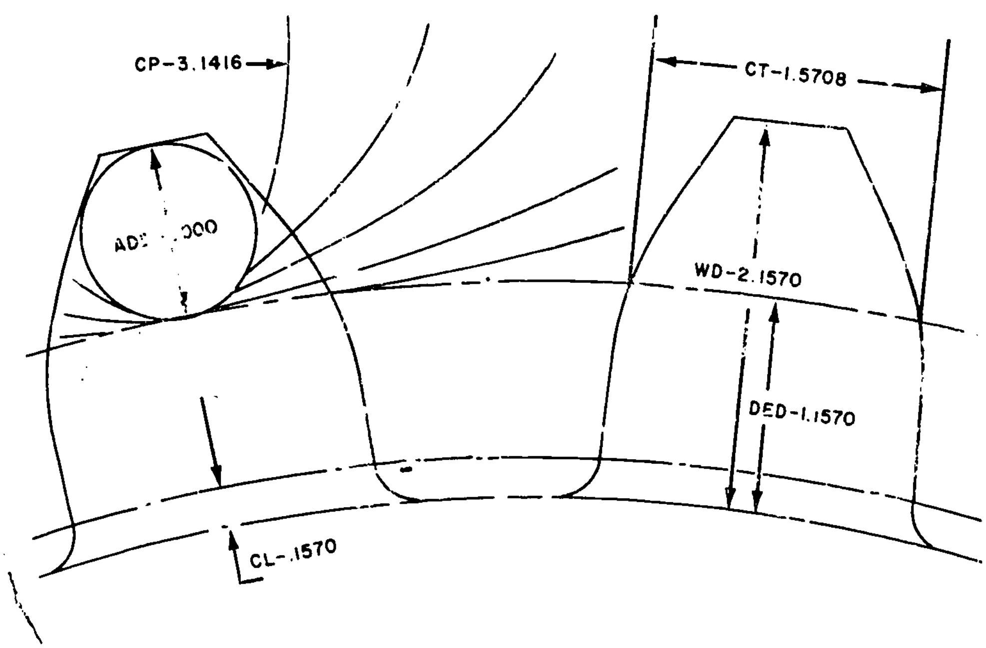
a. Addendum (ADD) - 1.000

(1) The distance from the top of the tooth to the pitch line.

b. Circular Pitch (CP) - 3.1416

(1) The length of an arc equal to the circumference of a one-inch circle, covers one tooth and one space on the pitch circle.

(2) Measure the circular pitch on the pitch line. If you could draw a circle inside the tooth using the 1-inch ADD as the diameter the circumference of the circle would be 3.1416. Using your imagination, break the circle at one point on the circumference, imagining the circumference is a string. Lay the imaginary string on the pitch line at one side of the tooth. Stretch the other end as far as possible on the pitch line: it will stretch to a corresponding point on the next adjacent tooth on the pitch line.



28.438

Figure III-A.—Tooth elements on a #1 diametral pitch gear.

c. Circular Thickness (CT) - 1.5708

(1) One-half of the circular pitch, measured at the pitch line.

d. Clearance (CL) - 0.15708

(1) One-tenth of the chordal thickness; move decimal one place to the left.

e. Dedendum (DED) - 1.15708

(1) The sum of an addendum plus a clearance.

(2)
```
  1.000  - ADD
+ .1570  - CL
  1.1570 - DED
```

f. Working Depth (WKD) - 2.000

(1) The sum of two addendums.

(2)
```
  1.000 - ADD
+ 1.000 - ADD
  2.000 - WKD
```

g. Whole Depth (WD) - 2.15708

(1) The sum of an addendum and a dedendum.

(2)
```
  1.000  - ADD
+ 1.1570 - DED
  2.1570 - WD
```

h. Diametral Pitch (DP) -

(1) The ratio of the number of teeth per inch of pitch diameter.

(2) $\frac{NT}{PD} = DP$

i. Chordal Addendum - a_c

(1) The distance from the top of a gear tooth to a chord subtending (extending under) the intersections of the tooth thickness arc and the sides of the tooth.

(2) $a_c = ADD + \frac{(CT)^2}{4(PD)}$

j. Chordal Thickness - t_c

(1) The thickness of the tooth, measured at the pitch circle.

$$t_c = PD \sin\left(\frac{90^\circ}{N}\right)$$

2. GEAR ELEMENTS

a. Number of Teeth (NT)

(1) Connecting link between the tooth elements and gear elements

(2) Number of teeth in gear

(3) $\frac{PD}{ADD} = NT$

b. Pitch Diameter (PD)

(1) Diameter of the pitch circle.

(2) For every tooth in the gear there is an addendum on the pitch diameter.

(3) ADD x NT = (PD)

c. Outside Diameter (OD)

(1) The diameter of the gear

(2) Since there is an addendum (ADD) on the pitch diameter (PD) for each tooth, the two elements are directly related. Therefore, the outside diameter is simply the pitch diameter (PD) plus two addendums (ADD), or simulated teeth. The formulas read:

(a) ADD X NT = PD

(b) ADD X (NT + 2) = OD or

(c) PD + 2 ADD = OD

d. Linear Pitch (LP)

(1) The linear pitch is the same as the circular pitch except that it is the

lineal measurement of pitch on a gear rack.

(2) CP = LP

(3) Figure III-B illustrates linear pitch.

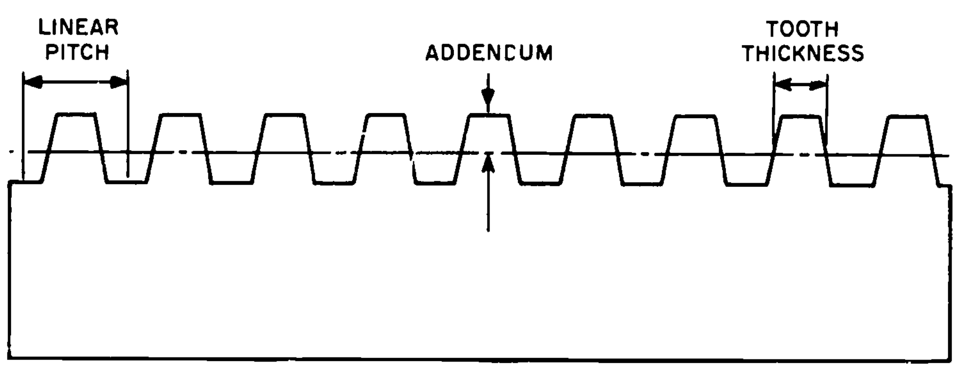
3. GEAR AND TOOTH ELEMENT RELATIONSHIP

TOOTH	GEAR
a. ADD	h. PD
b. DED	i. OD
c. CP	j. a_c
d. CT	k. t_c
e. WD	
f. CL	
g. DP	

(1) NT is the connecting link between tooth elements and gear elements.
(2) To complete calculate a gear, one tooth and one gear element must be known.
(3) For every tooth in the GEAR there is a CP on the PC.
(4) For every tooth in the GEAR there is an ADD on the PD.

FORMULAS

1. $ADD = \frac{1.000}{DP}$
2. $CP = \frac{3.1416}{DP}$
3. $CT = \frac{1.5708}{DP}$
4. $CL = \frac{.15708}{DP}$
5. $DED = \frac{1.15708}{DP}$
6. $WKD = \frac{2.000}{DP}$
7. $WD = \frac{2.15708}{DP}$
8. $DP = \frac{NT}{PD}$ or transpose any other formula with DP involved.
9. $NT = \frac{PD}{ADD}$
10. PD = ADD X NT
11. OD = ADD X (NT + 2)



28.439

Figure III-B.—Linear pitch.

APPENDIX IV

GLOSSARY

When you enter a new occupation, you must learn the vocabulary of the trade so that you understand your fellow workmen and can make yourself understood by them. Shipboard life requires that Navy personnel learn a relatively new vocabulary—even new terms for many commonplace items. The reasons for this need are many, but most of them boil down to convenience and safety. Under certain circumstances, a word or a few words may mean an exact thing or may mean a certain sequence of actions which makes it unnecessary to give a lot of explanatory details.

This glossary is not all-inclusive, but it does contain many terms that every Machinery Repairman should know. The terms given in this glossary may have more than one definition; only those definitions as related to the Machinery Repairman are given.

AISI: American Iron and Steel Institute.

ABRASIVE: A hard, tough substance which has many sharp edges.

ALLOWANCE: Difference between maximum size limits of mating parts.

ALLOYING: Procedure of adding elements other than those usually comprising a metal or alloy to change its characteristics and properties.

ALLOYING ELEMENTS: Elements added to nonferrous and ferrous metals and alloys to change their characteristics and properties.

ANNEALING: The softening of metal by heating and slow cooling.

ARBOR: The principal axis member, or spindle, of a machine by which a motion of revolution is transmitted.

ASTM: American Society for Testing Metals.

BABBITT: A lead base alloy used for bearings.

BEND ALLOWANCE: An additional amount of metal used in a bend in metal fabrication.

BENCH MOLDING: The process of making small molds on a bench.

BEVEL: A term for a plane having any angle other than 90° to a given reference plane.

BINARY ALLOY: An alloy of two metals.

BISECT: To divide into two equal parts.

BLOWHOLE: A hole in a casting caused by trapped air or gases.

BOND: Appropriate substance used to hold grains together in grinding wheels.

BORING BAR: A tool used for boring, counterboring, reboring, facing, grooving, etc. where true alignment is of primary importance.

BRINELL: A type of hardness test.

BRITTLENESS: That property of a material which causes it to break or snap suddenly with little or no prior sign of deformation.

BRONZE: A nonferrous alloy composed of copper and tin and sometimes other elements.

CALIBRATION: The procedure required to adjust an instrument or device to produce a standardized output with a given input.

CARBON: An alloying element.

CASTING: A metal object made by pouring melted metal into a mold.

CHAMFER: A bevel surface formed by cutting away the angle of one or two intersecting faces of a piece of material.

CONTOUR: The outline of a figure or body.

DRIFT PIN: A conical-shaped pin gradually tapered from a blunt point to a diameter larger than the hole diameter.

DUCTILITY: The ability to be molded or shaped without breakage.

EXTRACTORS: Tool used in removal of broken taps.

FABRICATE: To shape, assemble, and secure in place the component parts in order to form a complete whole for manufacture.

FALSE CHUCK: Sometimes applied to the facing material used in rechucking a piece of work in the lathe.

FATIGUE: The tendency of a material to break under repeated strain.

FILE FINISH: Finishing a metal surface with a file.

FILLET: A concave internal corner in a metal component.

FINISH ALLOWANCE: An amount of stock left upon the surface of a casting for the operation of machine finish.

FINISH MARKS: Marks used to indicate the degree of smoothness of finish to be achieved on surfaces to be machined.

GRAIN: The cutting edges of a grinding wheel.

HARDNESS: The ability of a material to resist penetration.

HELIX: A groove which advances longitudinally on a cylindrical object.

HONING: Finishing machine operation using stones vice a tool bit or cutting tool.

INVOLUTE: Usually referred to as a cutter used in gearing.

JIGS: A fixed fixture used in production machining, or to hold a specific job for machining.

KNOOP: Trade name used in hardness testing.

MANDREL: Tool used to mount work usually done in a lathe, or milling machine.

NORMALIZING: Heating iron-base alloys to approximately 100°F above the critical temperature range followed by cooling to below that range in still air at ordinary temperature.

OCCUPATIONAL STANDARDS: Requirements that are directly related to the work of each rating.

PERISCOPE: An instrument used for observing objects from a point below the object lens. It consists of a tube fitted with an object lens at the top, an eyepiece at the bottom and a pair of prisms or mirrors which change the direction of the line of sight. Mounted in such a manner that it may be rotated to cover all or a part of the horizon or sky and fitted with a scale graduated to permit taking of bearings, it is used by submarines to take observations when submerged.

PERPENDICULAR: Vertical lines extending through the outlines of the hull ends and the designer's waterline.

PIG IRON: Cast iron as it comes from the blast furnace in which it was produced from iron ore.

PINHOLE: Small hole under the surface of the casting.

PLAN: A drawing prepared for use in building a ship.

PLASTICITY: That property which enables a material to be excessively and permanently deformed without breaking.

PREHEATING: The application of heat to the base metal before it is welded or cut.

PUNCH, PRICK: A small punch used to transfer the holes from the template to the plate. Also called a CENTER PUNCH.

QUENCHING: Rapid cooling of steels at different rates.

REAMING: Enlarging a hole by the means of revolving it in a cylindrical, slightly tapered tool with cutting edges running along its sides.

RECHUCKING: Reversing of a piece of work upon a faceplate so that the surface that was against the faceplate may be turned to shape.

REFERENCE PLANE: On a drawing, the normal plane from which all information is referenced.

RPM: Revolutions per minute.

SCALE: The ratio between the measurement used on a drawing and the measurement of the object it represents. A measuring device such as a ruler, having special gradations.

SECTOR: A figure bounded by two radii and the included arc of a circle, ellipse, or other central curve.

SPOT FACING: Turning a circular bearing surface about a hole. It does not affect a pattern.

STANDARD CASING: The half of a split casing that is bolted to the foundation, as opposed to the half, or cover, which can be removed with minimum disturbance to other elements of the equipment.

STRAIGHTEDGE: Relatively long piece of material having one or both edges a true plane.

STRENGTH: The ability of a material to resist strain.

STRESS RELIEVING: Heat treatment to remove stresses or casting strains.

STUD: (1) A light vertical structure member, usually of wood or light structural steel, used as part of a wall and for supporting moderate loads. (2) A bolt threaded on both ends, one end of which is screwed into a hole drilled and tapped in the work, and is used where a through bolt cannot be fitted.

SYNTHETIC MATERIAL: A complex chemical compound which is artificially formed by the combining of two or more simpler compounds or elements.

TEMPER: To relieve internal stress by heat treating.

TEMPLATE: A pattern used to reproduce parts.

TOLERANCE: An allowable variation in the dimensions of a machined part.

VICKERS: A scale or test used in metal hardness testing.

VITRIFIED BOND: A man made bond used in grinding wheels.

WAVINESS: Used as a term in the testing of finish machining of parts.

ZINC: An alloy used widely in die casting.

INDEX

C

Q

R

S

T

U

V

W

Z

MACHINERY REPAIRMAN 3&2

NAVEDTRA 10530-E

Prepared by the Naval Education and Training Program Development Center, Pensacola, Florida

Your NRCC contains a set of assignments and perforated answer sheets. The Rate Training Manual, Machinery Repairman 3&2 NAVEDTRA 10530-E, is your textbook for the NRCC. If an errata sheet comes with the NRCC, make all indicated changes or corrections. Do not change or correct the textbook or assignments in any other way.

HOW TO COMPLETE THIS COURSE SUCCESSFULLY

Study the textbook pages given at the beginning of each assignment before trying to answer the items. Pay attention to tables and illustrations as they contain a lot of information. Making your own drawings can help you understand the subject matter. Also, read the learning objectives that precede the sets of items. The learning objectives and items are based on the subject matter or study material in the textbook. The objectives tell you what you should be able to do by studying assigned textual material and answering the items.

At this point you should be ready to answer the items in the assignment. Read each item carefully. Select the BEST ANSWER for each item, consulting your textbook when necessary. Be sure to select the BEST ANSWER from the subject matter in the textbook. You may discuss difficult points in the course with others. However, the answer you select must be your own. Remove a perforated answer sheet from the back of this text, write in the proper assignment number, and enter your answer for each item.

Your NRCC will be administered by your command or, in the case of small commands, by the Naval Education and Training Program Development Center. No matter who administers your course you can complete it successfully by earning a 3.2 for each assignment. The unit breakdown of the course, if any, is shown later under Naval Reserve Retirement Credit.

WHEN YOUR COURSE IS ADMINISTERED BY LOCAL COMMAND

As soon as you have finished an assignment, submit the completed answer sheet to the officer designated to grade it. The graded answer sheet will not be returned to you.

If you are completing this NRCC to become eligible to take the fleetwide advancement examination, follow a schedule that will enable you to complete all assignments in time. Your schedule should call for the completion of at least one assignment per month.

Although you complete the course successfully, the Naval Education and Training Program Development Center will not issue you a letter of satisfactory completion. Your command will make an entry in your service record, giving you credit for your work.

WHEN YOUR COURSE IS ADMINISTERED BY THE NAVAL EDUCATION AND TRAINING PROGRAM DEVELOPMENT CENTER

After finishing an assignment, go on to the next. Retain each completed answer sheet until you finish all the assignments in a unit (or in the course if it is not divided into units). Using the envelopes provided, mail your completed answer sheets to the Naval Education and Training Program Development Center where they will be graded and the score recorded. Make sure all blanks at the top of each answer sheet are filled in. Unless you furnish all the information required, it will be impossible to give you credit for your work. The graded answer sheets will not be returned.

The Naval Education and Training Program Development Center will issue a letter of satisfactory completion to certify successful completion of the course (or a creditable unit of the course). To receive a course-completion letter, follow the directions given on the course-completion form in the back of this NRCC.

You may keep the textbook and assignments for this course. Return them <u>only</u> in the event you disenroll from the course or otherwise fail to complete the course. Directions for returning the textbook and assignments are given on the book-return form in the back of this NRCC.

PREPARING FOR YOUR ADVANCEMENT EXAMINATION

Your examination for advancement is based on the Occupational Standards for your rating as found in the MANUAL OF NAVY ENLISTED MANPOWER AND PERSONNEL CLASSIFICATIONS AND OCCUPATIONAL STANDARDS (NAVPERS 18068). These Occupational Standards define the minimum tasks required of your rating. The sources of questions in your advancement examination are listed in the BIBLIOGRAPHY FOR ADVANCEMENT STUDY (NAVEDTRA 10052). For your convenience, the Occupational Standards and the sources of questions for your rating are combined in a single pamphlet for the series of examinations for each year. These OCCUPATIONAL STANDARDS AND BIBLIOGRAPHY SHEETS (called Bib Sheets), are available from your ESO. Since your textbook and NRCC are among the sources listed in the bibliography, be sure to study both as you take the course. The qualifications for your rating may have changed since your course and textbook were printed, so refer to the latest edition of the Bib Sheets.

NAVAL RESERVE RETIREMENT CREDIT

This course is evaluated at 22 Naval Reserve Retirement points. These points are creditable to personnel eligible to receive them under current directives governing retirement of Naval Reserve personnel. Points will be credited in units as follows:

Unit 1: 12 points upon satisfactory completion of Assignments 1 through 6.

Unit 2: 10 points upon satisfactory completion of Assignments 7 through 11.

COURSE OBJECTIVE

This course provides knowledge that will help you develop your skills in the following areas:

Use and care of precision handtools, managing a toolroom, layout of parts, general bench work, working with metals and plastics and basic metallurgy.

Use and care of shop equipment including power saws, band saws, drill presses, grinders, engine lathes, turret lathes, milling machines, shapers, planers, engravers, metal plating equipment, and tool and cutter grinders.

Shop work, including grinding of lathe and shaper tool bits, cutting spur gears, turning shafts, metal buildup, and machining various parts.

The organization of a shipboard repair department and planning and organizing shop work.

While working on this correspondence course, you may refer freely to the text. You may seek advice and instruction from others on problems arising in the course, but the solutions submitted must be the result of your own work and decisions. You are prohibited from referring to or copying the solutions of others, or giving completed solutions to anyone else taking the same course.

Naval courses may include a variety of questions -- multiple-choice, true-false, matching, etc. The questions are not grouped by type; regardless of type, they are presented in the same general sequence as the textbook material upon which they are based. This presentation is designed to preserve continuity of thought, permitting step-by-step development of ideas. Some courses use many types of questions, others only a few. The student can readily identify the type of each question (and the action required) through inspection of the samples given below.

MULTIPLE-CHOICE QUESTIONS

Each question contains several alternatives, one of which provides the best answer to the question. Select the best alternative, and blacken the appropriate box on the answer sheet.

SAMPLE

s-1. The first person to be appointed Secretary of Defense under the National Security Act of 1947 was
1. George Marshall
2. James Forrestal
3. Chester Nimitz
4. William Halsey

Indicate in this way on the answer sheet:

	1 T	2 F	3	4	
s-1	□	■	□	□	___

TRUE-FALSE QUESTIONS

Mark each statement true or false as indicated below. If any part of the statement is false the statement is to be considered false. Make the decision, and blacken the appropriate box on the answer sheet.

SAMPLE

s-2. Any naval officer is authorized to correspond officially with any systems command of the Department of the Navy without his commanding officer's endorsement.

Indicate in this way on the answer sheet:

	1 T	2 F	3	4	
s-2	□	■	□	□	___

MATCHING QUESTIONS

Each set of questions consists of two columns, each listing words, phrases or sentences. The task is to select the item in column B which is the best match for the item in column A that is being considered. Items in column B may be used once, more than once, or not at all. Specific instructions are given with each set of questions. Select the numbers identifying the answers and blacken the appropriate boxes on the answer sheet.

SAMPLE

In questions s-3 through s-6, match the name of the shipboard officer in column A by selecting from column B the name of the department in which the officer functions.

A	B
s-3. Damage Control Assistant	1. Operations Department
s-4. CIC Officer	2. Engineering Department
s-5. Disbursing Officer	3. Supply Department
s-6. Communications Officer	

Indicate in this way on the answer sheet:

	1 T	2 F	3	4	
s-3	□	■	□	□	___
s-4	■	□	□	□	___
s-5	□	□	■	□	___
s-6	■	□	□	□	___

Assignment 1

Scope of the Machinery Repairman Rating; Toolrooms and Tools; Layout and Benchwork

Textbook Assignment: Pages 1-1 through 3-18

In this course you will demonstrate that learning has taken place by correctly answering training items. The mere physical act of indicating a choice on an answer sheet is not in itself important; it is the mental achievement, in whatever form it may take, prior to the physical act that is important and toward which course learning objectives are directed. The selection of the correct choice for a course training item indicates that you have fulfilled, at least in part, the stated objective(s).

The accomplishment of certain objectives, for example, a physical act such as drafting a memo, cannot readily be determined by means of objective type course items; however, you can demonstrate by means of answers to training items that you have acquired the requisite knowledge to perform the physical act. The accomplishment of certain other learning objectives, for example, the mental acts of comparing, recognizing, evaluating, choosing, selecting, etc., may be readily demonstrated in a course by indicating the correct answers to training items.

The comprehensive objective for this course has already been given. It states the purpose of the course in terms of what you will be able to do as you complete the course.

The detailed objectives in each assignment state what you should accomplish as you progress through the course. They may appear singly or in clusters of closely related objectives, as appropriate; they are followed by items which will enable you to indicate your accomplishment.

All objectives in this course are learning objectives and items are teaching items. They point out important things, they assist in learning, and they should enable you to do a better job for the Navy.

This self-study course is only one part of the total Navy training program; by its very nature it can take you only part of the way to a training goal. Practical experience, schools, selected reading, and the desire to accomplish are also necessary to round out a fully meaningful training program.

Learning Objective: Identify the Machinery Repairman rating, listing the sources of training, information, and overview functions of the rating. Textbook pages 1-1 through 1-8.

1-1. Machinery Repairmen always work in machine shops which are fully equipped and have all the associated machinery.

1-2. Which of the following types of training is/are provided by the machinery repairman class "A" school?
1. Classroom theory
2. Hands on experience
3. Project procedures
4. All of the above

1-3. By what training system is the heat treatment of metals taught?
1. "A" school
2. On-job-training
3. Self-taught
4. "C" school

1-4. The courses that are mandatory to meet advancement requirements are listed in what publication(s)?
1. BUPERSINST 1430.16
2. NAVSEA publications
3. NAVPERS 10530E
4. NAVEDTRA 10194-C

1-5. The Bibliography for Advancement Study, NAVEDTRA 10052, is revised annually.

1-6. Preventive maintenance reduces the need for major corrective maintenance.

1-7. By what procedure is the Naval Ships' Technical Manual kept up-to-date?
1. Annual changes
2. Quarterly changes
3. Monthly changes
4. Changes when revised

1-8. In what order are the drawings listed in the Ship Drawing Index (SDI)?
1. Alphabetical order
2. Numerical order
3. By ship's hull number
4. By compartment numbers

Learning Objective: Identify the duties/responsibilities of a toolroom keeper and the characteristics of a properly organized toolroom. Textbook pages 2-1 through 2-4.

1-9. Normally, which of the following functions is NOT a duty of the toolroom keeper?
1. Controlling the custody of tools
2. Testing metal for hardness
3. Storing measuring instruments properly
4. Replacing broken hammer handles

1-10. Which of the following provisions is NOT normally required in a toolroom ashore, but should be provided in a shipboard toolroom?
1. Cushions for reducing damage to tools
2. Fasteners for holding tools in place
3. Special bins for storing power tools
4. Replacement parts for repairing broken tools

● For items 1-11 through 1-14, assume that you have the following storage spaces in your toolrooms onboard ship:
A. Metal shelves
B. Metal rotating bins
C. Lined wooden boxes
D. Metal cabinet with various sized drawers
E. Wooden wall rack 3 feet wide, extending to the overhead, and equipped with spring-loaded fasteners

1-11. Where should the cutter illustrated in figure 2-12 in the textbook be stored?
1. On A
2. In C
3. In D
4. On E

1-12. Which storage space should you use for a gear tooth vernier?
1. B
2. C
3. D
4. E

1-13. Storage space B is most suitable for
1. wood chisels
2. hammers
3. nuts and bolts
4. small power drills

1-14. Which of the following is a good practice for a toolroom keeper to follow?
1. Check the accuracy of precision tools before issuing them and again after they are returned
2. Repair damaged tools immediately after they are returned to the toolroom
3. Remove all oil and foreign matter from tools returned to the storeroom and store them dry
4. Sharpen all cutting tools before placing them in their storage spaces

1-15. What recommended procedure should a storeroom keeper use as a means of controlling tools loaned to all departments on a ship?
1. Provide a checkout log at the issuing window and require the borrower to sign his name and list the tool borrowed
2. Issue numbered tool checks to all departments; when a department borrows a tool, it surrenders a tool check to the toolroom keeper
3. Maintain a columnar checkout board with a separate column for each department; when a tool is issued, its name is recorded in the proper column on the board
4. Use checkout slips or forms on which are recorded the borrower's name, name of the tool and its location, and the date

1-16. If maintaining a custody record of tools issued fails to provide you with adequate control of the tools, what additional measures should you take to make your control of tools more effective?
1. Take a periodic physical count of each tool and each item of supply
2. Require each department to submit a list of tools and supplies in its possession at the end of each week
3. Require departments to turn in tools at the end of each working day
4. Go to each department at least once a month and check the tools and supplies in its possession against your record

1-17. Aboard ship, who initially signs the custody card for an item listed as "controlled equipage" when it is received from the supply department?
1. User
2. Toolroom keeper
3. Division officer
4. Department head

1-18. Inventory of controlled equipage is conducted at least once each quarter.

Learning Objective: Identify the characteristics and uses of adjustable gages. Textbook pages 2-4 through 2-10.

1-19. Using a lathe, you are to turn a piece of 8-inch stock down to 7 1/2 +1/64 inch. Normally, what is the most accurate procedure to use?
   1. Measure the stock with a 6-inch rule, scribe a circle on the end of the stock, and turn it down until the cutting tool reaches the scribe mark
   2. Set a caliper at a reading slightly greater than 7 1/2 inches and gage the workpiece at intervals during the turning operation
   3. Set the lathe to cut to a fixed depth and repeat the turning operation the required number of times
   4. Measure the workpiece at intervals during the turning operation with any type of gage, and continue to cut until reaching the desired dimension

1-20. For what purpose is a dial indicator most useful?
   1. To locate the center of a piece of round stock
   2. To check a shaft for out-of-roundness
   3. To true the internal surfaces of a cylinder
   4. To measure distances between two closely mated surfaces

1-21. Standard vernier calipers have scales graduated in what increments?
   1. 0.0001 in.
   2. 0.001 in.
   3. 0.01 in.
   4. 0.1 in.

1-22. To check the inside of a cylinder for out-of-roundness, you should use a/an
   1. master ring gage
   2. dial vernier caliper
   3. outside micrometer
   4. dial bore gage

1-23. A gear tooth vernier is able to measure what dimension(s)?
   1. One linear dimension
   2. One linear dimension and one angle simultaneously
   3. Two linear dimensions simultaneously
   4. Two linear dimensions and two arcs simultaneously

1-24. What method is used to determine the height at which an adjustable parallel is locked?
   1. A scale on the diagonal edge of one block
   2. An outside micrometer measurement
   3. A feeler gage placed between the block and the workpiece
   4. A direct comparison with a gage block

Learning Objective: Identify the characteristics and uses of fixed gages. Textbook pages 2-11 through 2-15.

1-25. The steel rule with holder set is used normally for measuring
   1. recesses
   2. gear teeth
   3. thicknesses
   4. cylindrical surfaces

1-26. A one-quarter size scale that is 12 inches long is calibrated to read from zero to
   1. 3 in.
   2. 4 in.
   3. 36 in.
   4. 48 in.

1-27. Acme thread tool gages are used to
   1. measure the pitch of screw or bolt threads
   2. estimate the thread angle of screws or bolts
   3. set up thread-cutting tools
   4. determine clearances between bolt and nut threads

1-28. Suppose that a mechanism has a bushing with a 6-inch diameter and has a clearance of 0.001 inch between it and the mechanism housing. Which tool should you use to check the clearance of the housing?
   1. Feeler gage
   2. 6-inch metal scale
   3. Cutter clearance gage
   4. Micrometer

1-29. A rounded shoulder on a part to be machined is measured with a
   1. Center gage
   2. Feeler gage
   3. Surface gage
   4. Radius gage

1-30. A sine bar is used to locate very small irregularities on a machined surface.

1-31. Which of the following is an ordinary use of parallel blocks?
1. To check the concavity of a curved surface
2. To determine whether the opposite edges of a workpiece are parallel
3. To compare workpiece dimensions with a finished piece
4. To accurately position work for machining

Learning Objective: Identify the proper procedures for using and maintaining micrometers. Textbook pages 2-15 through 2-21.

1-32. Which of these measuring tools is the most precise for checking the accuracy of a micrometer?
1. Gage block
2. Machinist's rule
3. Dial indicator
4. Vernier caliper

1-33. How can you determine if a micrometer needs to be recalibrated?
1. Ask your supervisor
2. Ask the supply clerk
3. Read index B in Tools and Their Uses, NAVEDTRA 10085-B1
4. Check the tool calibration sticker

1-34. When using a micrometer, you should avoid all EXCEPT which of the following practices?
1. Back the spindle away from the anvil when placing it in storage
2. Measure moving parts
3. Open it by spinning the frame
4. Carry it in your pocket

1-35. Which of these conditions can cause a zero adjusted micrometer to read more than zero when its spindle is in contact with the anvil?
1. Thread wear
2. Burrs on the spindle
3. Worn anvil
4. Rusted barrel

A number of steps in adjusting a thimble cap type micrometer are listed below in scrambled order.

A. Rotate the thimble to zero
B. Back the spindle away from the anvil
C. Rotate the spindle counterclockwise
D. Release the thimble cap
E. Tighten the thimble cap
F. Bring the spindle into contact with the anvil

1-36. In what order should these steps be performed?
1. B, D, F, C, A, E
2. B, D, F, A, C, E
3. F, D, B, A, C, E
4. F, D, C, A, B, E

1-37. A cylindrical standard is used to test a micrometer for
1. incorrect positioning of the anvil
2. master gage variations
3. concave wear of the measuring faces
4. distortion of the frame

1-38. If the measuring surfaces of a micrometer are concave, how should they be corrected?
1. By grinding
2. By lapping
3. By filing
4. By facing on a lathe

1-39. Which of the following instruments is required to zero set an inside micrometer?
1. Hollow cylinder standard
2. Micrometer caliper
3. Vernier caliper
4. Cylinder gage

1-40. To lubricate the internal parts of a micrometer, you should use
1. petroleum jelly
2. silicone grease
3. graphite powder
4. light oil

1-41. A sticky or sluggish dial in a vernier dial indicator can be freed by applying a few drops of light machine oil.

Learning Objective: Indicate the proper uses of blueprints and identify basic blueprint symbols. Textbook pages 3-1 through 3-7.

1-42. What is a blueprint?
1. A detailed enlargement of a portion of a drawing
2. A reduced copy of a large drawing
3. An original drawing
4. An exact copy of an original drawing

1-43. You can locate your shipboard duty station by referring to a blueprint called a/an
1. plan view
2. assembly print
3. subassembly print
4. detail print

1-44. The blueprints that you will use most often in shop work are known as
1. plan views
2. assembly prints
3. subassembly prints
4. detail prints

1-45. Which of the following types of view is most commonly used on detail prints?
1. Isometric
2. Perspective
3. Orthographic
4. Auxiliary

1-46. Which section of a blueprint for a part contains the pattern number of the part?
1. Reverse side
2. Upper right-hand corner
3. Upper left-hand corner
4. Title box

- For items 1-47 through 1-49, refer to textbook figures 3-2 through 3-5.

1-47. Which symbol indicates a short break?
1. — - — - —
2. ⋀⋀⋀⋀⋀⋀⋀⋀⋀⋀
3. ------------
4. ┌—— ——┐

1-48. Which symbol indicates parallelism?
1. ———
2. ⊥ ⊥
3. ⊥
4. ═══

1-49. Which symbol indicates concealed edges?
1. — - — - —
2. — - - — - - —
3. Ⓒ
4. -----------------

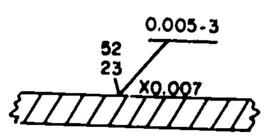


Figure 1A

- In answering items 1-50 and 1-51, refer to figure 1A.

1-50. What type of lay is required by this drawing?
1. Angular in both directions to surface boundary line
2. Parallel to the surface boundary line
3. Multidirectional with respect to the surface boundary line
4. Perpendicular to the surface boundary line

1-51. What is the maximum permissible roughness height rating?
1. X.007
2. 52
3. 3
4. 0.005-3

Learning Objective: Locate and correctly interpret dimensions on blueprints. Textbook pages 3-7 through 3-10.

1-52. What is the decimal equivalent of 1/32 inch?
1. 0.625 in.
2. 0.0625 in.
3. 0.0313 in.
4. 0.313 in.

1-53. One inch is equal to how many millimeters?
1. 1.00 mm
2. 2.54 mm
3. 25.40 mm
4. 100.00 mm

1-54. How many inches are there in 190.5 millimeters?
1. 7.5
2. 8.0
3. 8.5
4. 9.0

1-55. Tolerance is the difference between the allowable minimum and maximum dimensions.

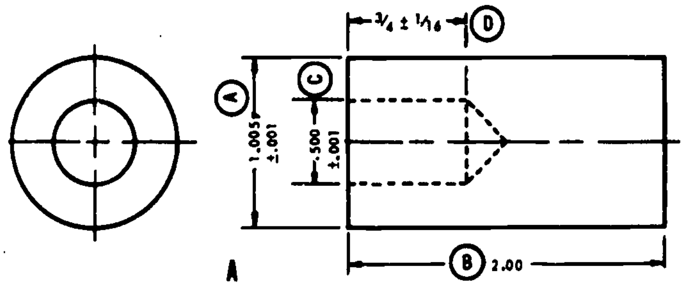


Figure 1B

- In answering items 1-56 through 1-58, refer to figure 1B.

1-56. What is the total tolerance of dimension A?
1. 0.0005 in.
2. 0.001 in.
3. 0.002 in.
4. 0.006 in.

1-57. What is the maximum permissible length of dimension B?
1. 2.0 in.
2. 2.05 in.
3. 2.005 in.
4. 2.0005 in.

1-58. What is the short limit of dimension D?
1. 5/8 in.
2. 11/16 in.
3. 3/4 in.
4. 13/16 in.

1-59. A shaft 2.276 inches in diameter is fitted into a hole 2.275 inches in diameter. The difference in the diameters of the shaft and the hole is called the
1. clearance allowance
2. interference allowance
3. tolerance
4. isometric allowance

1-60. How can you assure that the required clearance allowance will be kept while some tolerance is permitted between a shaft and the hole into which it fits?
1. Give both the shaft and the hole a plus tolerance
2. Give both the shaft and the hole a minus tolerance
3. Give the shaft a plus tolerance and the hole a minus clearance
4. Give the shaft a minus tolerance and the hole a plus tolerance

---

Learning Objective: Identify the terminology and procedures used during layout. Textbook pages 3-10 through 3-18.

1-61. The term layout is used to describe the
1. positioning of parts on the workbench
2. marking of metal surfaces as an outline for machining
3. guide which aids in the disassembling of the equipment to be repaired
4. original drawing used to make a blueprint

1-62. The usual way of improving the visibility of layout lines on the surface of a metal is to
1. coat the metal surface with layout dye before inscribing
2. retrace the lines with chalk
3. rub the surface with chalk before inscribing
4. exert heavy pressure on the scribe

1-63. What devices should you place under layout work to vary its height?
1. Surface plates
2. Parallel blocks
3. Angle plates
4. V-blocks

1-64. What procedure is used in layouts to maintain the best accuracy?
1. Use each preceding layout line as a reference for the next line
2. Establish a baseline to which all other lines are referenced
3. Establish a minimum of two reference lines and refer to both in laying out successive lines
4. Always place layout lines in such a way that a maximum amount of material is removed from one surface

1-65. When inscribing a line or a metal surface, how should you position the handle of the scratch awl you use with reference to a straightedge and the direction of movement?
1. Point it opposite to the direction of movement and over the straightedge
2. Point it in the direction of movement and over the straightedge
3. Point it opposite to the direction of movement and away from the straightedge
4. Point it in the direction of movement and away from the straightedge

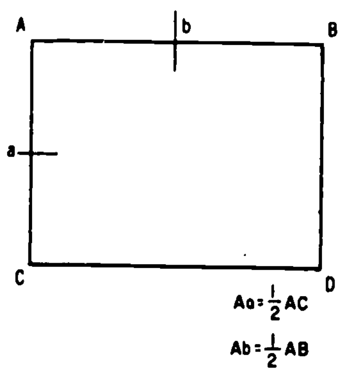


Figure 1C

1-66. What is the best way to find the center of the rectangle ABCD shown in figure 1C?
1. Scribe a line parallel to AC through point b; find the midpoint of this line
2. Scribe lines from points a and b square with the edge until they intersect
3. Scribe a line from A to D and another from B to C
4. Scribe two arcs with radius Aa

1-67. Which of the following tools may be used in conjunction with a surface plate?
1. Sine bar
2. Angle plate
3. Parallels
4. Each of the above

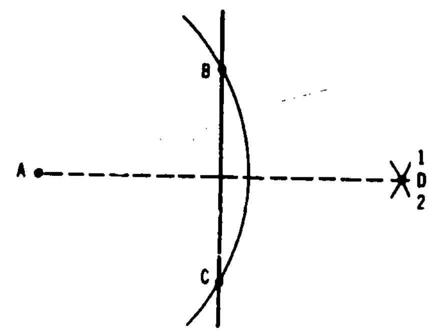


Figure 1D

- In answering items 1-68 and 1-69, refer to figure 1D.

1-68. In drawing a line perpendicular to line BC through point A, you first prick punch point A. Your second step is to
1. draw arc BC
2. prick punch points B and C
3. draw arcs 1 and 2
4. prick punch point D

1-69. Arc 2 is drawn with dividers centered on point
1. A
2. B
3. C
4. D

In answering items 1-70 and 1-71, refer to figure 1E. You are drawing a line perpendicular to line BC through point A.

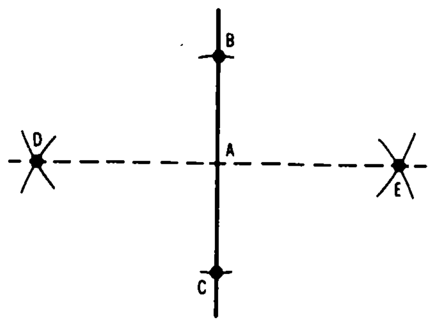


Figure 1E

1-70. Your first step is to
1. prick punch point A
2. draw arcs B and C
3. draw arcs at D and E
4. prick punch points D and E

1-71. Your second step is to
1. prick punch point A
2. draw arcs B and C
3. draw arcs at D and E
4. prick punch points D and E

1-72. When scribing parallel lines 2 inches apart using dividers to establish the points you need, you set the dividers to
1. 1 1/2 in.
2. 2 in.
3. 6 in.
4. any convenient length

Assignment 2

Layout and Benchwork; Metals and Plastic

Textbook Assignment: Pages 3-18 through 4-8

Learning Objective: Identify the terminology and procedures used to layout flange bolt holes. Textbook pages 3-18 through 3-21.

2-1. The straight-line distance between the centers of two adjacent flange bolt holes is called the
1. horizontal bisector
2. vertical bisector
3. pitch circle
4. pitch chord

2-2. The horizontal line across the face of the flange is called the
1. vertical bisector
2. horizontal bisector
3. pitch chord
4. pitch circle

2-3. What is the flange bolt hole circle called?
1. Pitch circle
2. Horizontal bisector
3. Pitch chord
4. Vertical bisector

2-4. What is the name of the vertical line across the face of a flange?
1. Pitch circle
2. Horizontal bisector
3. Pitch chord
4. Vertical bisector

2-5. How many pitch chords are used in laying out a 6-hole flange?
1. Six
2. Two
3. Three
4. Four

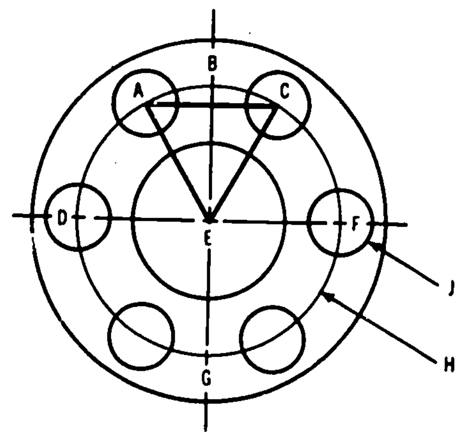


Figure 3A

- In answering Items 2-6 through 2-10, refer to figure 3A.

2-6. The horizontal bisector in figure 3A is indicated by line
1. AC
2. AD
3. BE
4. DF

2-7. The pitch chord in figure 3A is indicated by
1. line AC
2. line AE
3. line BG
4. CF

2-8. What line in figure 3A bisects a pitch chord?
1. Line CE
2. Line DE
3. Line FE
4. Line GB

2-9. What part of the construction shown in figure 3A is located first when a valve flange is laid out for drilling?
1. Line BG
2. Line DF
3. Point A
4. Point E

2-10. When the pipe flange in figure 3A is laid out for drilling, witness marks are placed on the
1. pitch circle
2. pitch chords
3. hole centers
4. hole circumferences

2-11. The following list is some of the steps you take in laying out valve flange bolt holes.
A. Scribe the bolt circle
B. Draw the bisectors
C. Mark off bolt hole centers
D. Plug the flange opening

In what order do you carry out these steps?
1. A, B, C, D
2. B, C, A, D
3. D, A, B, C
4. D, C, B, A

2-12. The pitch chords of a 6-hole pipe flange are equal to the
1. radius of the pitch circle
2. diameter of the pitch circle
3. radius of the flange
4. inside diameter of the pipe

2-13. What is the approximate pitch chord of a flange that has 5 bolt holes and a pitch diameter of 4 inches?
1. 0.59 in.
2. 1.18 in.
3. 2.00 in.
4. 2.35 in.

Learning Objective: Identify basic terminology and procedures associated with Benchwork. Textbook pages 3-21 through 3-23.

2-14. Benchwork includes all EXCEPT which of the following types of work?

1. Machining a part of a piece of equipment
2. Making layouts of parts
3. Fitting mating parts of equipment precisely
4. Assembling and disassembling equipment

2-15. In the benchwork required to restore equipment to operational status, what are the first and last steps respectively
1. Get information about the equipment and test it
2. Disassemble the equipment and reassemble it
3. Inspect the equipment for defects and repair those uncovered
4. Inspect the equipment for defects and reassemble it

2-16. In overhauling equipment, which of the following precautions should you take to protect the surfaces of parts?
1. Use only the force that is needed in disassembling and reassembling
2. Use only hammers or mallets with faces made of lead, plastic, or rawhide
3. Free stuck bolts or nuts with penetrating oil
4. Take all of the above precautions

Learning Objective: Identify terms and procedures associated with precision work. Textbook pages 3-23 through 3-32.

2-17. In metal workings scraping is used primarily to
1. remove sharp edges from castings
2. produce curved surfaces
3. smooth a machined surface
4. prepare castings for machining

2-18. You are scraping a flat surface to achieve a correct clearance. Why do you need to use prussian blue in this work?
1. It sticks to the high spots on the workpiece and shows up the areas that should be scraped
2. It helps prevent "frictional drag" in the scraping operation
3. It helps to maintain longitudinal as well as radial clearance on the flat surface
4. It keeps chips from breaking off the edges when the part is operating agai

2-19. A solid metal surface which has a cross-hatched appearance has been correctly scraped because cuts were made at an angle of 90° to each other.

2-20. One reason for removing burrs and sharp edges from machined pieces is to prevent damage to the equipment you are repairing.

2-21. Machine reaming is more accurate than hand reaming.

2-22. You are going to use a reamer which has a diameter of 1/4 inch. Which of the following is the properly sized rough hole?
1. 0.2500 in.
2. 0.2350 in.
3. 0.2505 in.
4. 0.2600 in.

2-23. Which of the following tools is used to shear a particular shape of hole during the machining process?
1. Reamer
2. Broach
3. Scraper
4. Babbitt

2-24. When you cut threads in a blind hole, what tap should you use first?
1. Taper
2. Bottoming
3. Plug
4. Cutting

2-25. What size tap drill should you select for a 3/8-20 NC tap?
1. 0.3906
2. 0.3680
3. 0.3250
4. 0.3125

2-26. A tap that is NOT perpendicular to the surface being tapped will usually align itself if it is turned into the hole a few additional turns.

2-27. The size of a die is usually found on the
1. trailing face of the die
2. edge of the die
3. leading face of the die
4. case of the die

2-28. The easiest and least damaging method of removing a broken tap is to break the tap into small pieces by using a punch and a hammer.

2-29. Why should you NEVER add water to acid?
1. The acid solution is premixed
2. Acid should be used full strength
3. The acid may react violently
4. Water will cause the tap to rust

Learning Objective: Identify the three types of fits and characteristics and manufacture of each type. Textbook pages 3-32 through 3-34.

2-30. What are three types of fit of plain cylindrical parts?
1. Interference, clearance, transitional
2. Clearance, moderate, transitional
3. Transitional, moderate, interference
4. Clearance, loose, tight

2-31. What is the PRI[illegible]Y reference for dimensional siz[illegible] and maintenance matters?
1. The manufacturers' technical manual
2. The appropriate NAVSHIP's Technical Manual
3. The ship's preventive maintenance program
4. The appropriate Preventive Maintenance System Maintenance Requirement Card

2-32. What class of fit includes locational fits?
1. Clearance fits
2. Transitional fits
3. Interference fits
4. Transitional and interference fits

2-33. Which of the following classes of fits most likely requires the use of heat?
1. Class 1
2. Class 2
3. Class 3
4. Class 4

Learning Objective: Identify the correct uses of hydraulic presses. Textbook pages 3-34 and 3-35.

2-34. The only way to determine the amount of pressure exerted by a hydraulic press is to watch the pressure gage.

2-35. What should you do to prevent mutilation of the surface of a workpiece that is being pressed?
1. Increase ram pressure
2. Lubricate ram pressure
3. Heat the workpiece
4. Insert a soft metal plate between the work and the ram

2-36. What should you do to prevent seizing between mated parts that are being force-fitted by a hydraulic ram?
1. Increase ram pressure
2. Lubricate the parts
3. Heat the parts
4. Insert a soft metal plate between the work

Learning Objective: Identify terms and procedures associated with oxyacetylene welding. Textbook pages 3-35 through 3-39.

2-37. Compressed oxygen should NEVER come into contact with oil or grease because an explosion could occur.

2-38. What color is the oxygen hose?
1. Red
2. Green
3. Blue
4. Yellow

2-39. On an oxyacetylene welding machine, how can the thread direction be used to identify the oxygen and acetylene hose connections?
1. The oxygen hose connection has left-hand threads; the acetylene hose connection has right-hand threads
2. Both acetylene and oxygen hose connections have left-hand threads
3. Both acetylene and oxygen hose connections have right-hand threads
4. The oxygen hose connections have right-hand threads; the acetylene hose connections have left-hand threads

2-40. What is the maximum amount the acetylene cylinder valve should be opened?
1. 1/4 turn
2. 1/2 turn
3. 1 turn
4. 1 1/2 turns

2-41. What is the maximum working pressure of acetylene?
1. 5 psig
2. 10 psig
3. 15 psig
4. 20 psig

2-42. From the flame characteristics listed below, select the characteristics of a neutral flame.

A. Sooty
B. Bluish-white
C. Yellowish
D. Bright inner cone
E. Long and bushy
F. Outer flame envelope

1. A, B, D
2. E, F, B
3. D, A, E
4. B, D, F

2-43. If a flashback occurs, you should immediately shut off the torch oxygen valve and then shut off the acetylene valve.

Learning Objective: Identify types and uses of fastening devices. Textbook pages 3-39 through 3-48.

2-44. Two general groups of fasteners commonly used by a Machinery Repairman are threads and keys. What is the third group?
1. Nuts
2. Pins
3. Bolts
4. Stays

2-45. After a nut which has been completely torqued down, what is the minimum number of full bolt threads allowed to protrude above the nut?
1. One
2. Two
3. Three
4. Four

2-46. What is the designation for a Woodruff key with a diameter of 3/8 inch and a width of 1/16 inch?
1. 203
2. 206
3. 403
4. 606

2-47. Taper pins are usually installed with a small hydraulic press.

2-48. Which of the following types of pins is used to align the end casing and housing of a pump?
1. Cotter pin
2. Taper pin
3. Dowel pin
4. Any of the above

Learning Objective: Identify the basic properties of metals which affect the shaping and uses of the metals. Textbook pages 4-1 through 4-3.

In items 2-49 through 2-51, select from column B the behavior of the metal that is made possible by the property in column A.

	A. Property	B. Behavior
2-49.	Ductility	1. Enables a metal to resist breaking
2-50.	Tensile strength	2. Enables a metal to be rolled or hammered into sheets
2-51.	Toughness	3. Enables a metal to be drawn out or pulled into wire form
		4. Enables a metal to resist pulling apart under stress

2-52. If a part having a factor of safety is designed to withstand a working load of 1150 pounds per square inch (psi), its ultimate strength is at least
1. 1150 psi
2. 1600 psi
3. 1950 psi
4. 2300 psi

2-53. High fatigue resistance is desirable in a material that is subjected to
1. abrasion
2. repetition of stress
3. corrosive substances
4. high temperatures

2-54. A coarse-grained surface over part of a break in a steel shaft indicates the area of
1. abrasion
2. crystallization
3. final failure
4. early failure

2-55. The heat resistance of a metal is indicated by its ability to
1. conduct heat
2. withstand stress at high temperatures
3. maintain a temperature that differs from the surrounding temperature
4. maintain constant dimensions under wide variations in temperature

2-56. Which of the following terms best describes the ease with which a metal may be planed and shaped?
1. Malleability
2. Plasticity
3. Ductility
4. Machinability

Learning Objective: Identify the characteristics and uses of ferrous metals. Textbook pages 4-3 through 4-6.

2-57. Which of the following materials are melted together to make cast iron?
1. Steel and iron ore
2. Iron ore and pig iron
3. Pig iron and scrap iron
4. Scrap iron and steel

2-58. The grade of carbon steel alloys produced in converting pig iron to steel is determined by the
1. amount of carbon burned out in refining
2. amount of nonferrous metal added after refining
3. mixture of coke and limestone used in refining
4. amount of carbon added after refining

2-59. High carbon steel is commonly used for which of the following applications?
1. Ship framing
2. Ship plating
3. Cutting tools
4. Electric wiring

2-60. Which of the following metals is contained in stainless steels?
1. Molybdenum
2. Chromium
3. Tungsten
4. Vanadium

2-61. Which of the following metals is used for valves for high-temperature, high-pressure, piping systems?
1. Tungsten steel
2. Bronze
3. Copper
4. Carbon-molybdenum steel

Learning Objective: Identify the characteristics and uses of non-ferrous metals. Textbook pages 4-6 through 4-8.

2-62. Which metal is commonly used for objects requiring high electrical conductivity?
1. Tin
2. Brass
3. Copper
4. Zinc

2-63. Which metal is the strongest and best to use on-board ship because of its high resistance to the corrosion effect of saltwater?
1. Aluminum
2. Copper
3. Bronze
4. Copper-nickel

2-64. Sherardized steel sheet has a coating of zinc applied by
1. electroplating
2. depositing vaporized zinc
3. rolling
4. hot dipping

2-65. Which of the following metals is used to protect a steel-hulled ship against the effects of electrolysis?
1. Iron
2. Copper
3. Lead
4. Zinc

2-66. Which of the following metals has the greatest weight per unit volume?
1. Bronze
2. Cast iron
3. Lead
4. Copper-nickel alloy

2-67. True brass is produced by alloying
1. copper and zinc
2. copper and nickel
3. bronze and tin
4. copper, aluminum, and phosphorus

2-68. Which of these materials is used in K Monel but not in regular Monel?
1. Iron
2. Aluminum
3. Silicon
4. Cobalt

2-69. K-Monel metal can be made easier to machine by
1. soaking in brine at least 24 hours before machining
2. annealing immediately before machining
3. alloying with copper several days before machining
4. sherardizing with zinc immediately before machining

Assignment 3

Metals and Plastics; Power Saws and Drilling Machines

Textbook Assignment: Pages 4-8 through 5-3

Learning Objective: Identify standard metal marking systems and characteristics of marked metals. Textbook pages 4-8 through 4-14.

3-1. Which of the following systems are used for marking iron and steel?
1. AA, AMS
2. ANSI, MIL-S-XXXX
3. CDA, ASME
4. SAE, AISI

3-2. A 4- or 5-digit SAE number is used to identify
1. nonferrous metals
2. plain steels
3. alloy steels
4. plain and alloy steels

3-3. What AISI number represents an alloy steel containing 5% nickel and 1/5% carbon?
1. 2205
2. 3105
3. 5220
4. 2520

3-4. The SAE designator 1050 indicates a
1. carbon-molybdenum steel
2. nickel steel
3. medium carbon steel
4. copper-nickel alloy

3-5. What is the approximate content of alloying elements in a free cutting steel with the classification SAE 1115?
1. 1% manganese and 0.15% carbon
2. 1% phosphorus, 1% sulfur, and 5% carbon
3. 11% chromium and 5% nickel
4. 11% carbon and 15% chromium

3-6. What are the chemical symbols for the elements in plain bronze?
1. Cu and Ni
2. Cu and Sn
3. Fe and V
4. Sn and Zn

3-7. What marking designation in the Aluminum Association Marking System identifies an aluminum which is more than 99% pure with no control over impurities?
1. 1075
2. 1999
3. 2030
4. 3056

● Items 3-8 and 3-9 refer to an aluminum alloy bearing the Aluminum Association designation 5052-H16.

3-8. The number 5052 in the designation indicates that the major alloy is
1. magnesium
2. silicon
3. manganese
4. copper

3-9. The number H16 in the designation shows that the aluminum is
1. strain hardened, then partially annealed and 1/4 hard
2. strain hardened only and 3/4 hard
3. strain hardened, then stabilized and 1/2 hard
4. artificially aged only and 3/4 hard

3-10. What technique is used in marking a steel bar according to the continuous identification marking system?
1. A continuous strip of specified width is painted along the entire length of the bar
2. The appropriate marking is printed with heavy ink at specified intervals along the bar's length
3. The appropriate marking is punched on with a metal stamper at specified intervals along the bar's length
4. The appropriate symbol of specified color is painted on each end of the bar

3-11. What information is shown on metals marked with the continuous identification marking system?
1. Producer's name or registered trademark and commercial designation of the steel
2. Name and trademark of the producer who finished the steel before it was marketed
3. Military Standards designation and Federal Government job order number
4. U.S. Bureau of Standards quality control number and SAE designation

3-12. Thin copper wire is identified by
1. continuous identification marking
2. spot symbols
3. peripheral symbols
4. tagging

3-13. Which symbol designates the physical condition of a metal?
1. CX
2. CR
3. CQ
4. CO

- In answering items 3-14 through 3-17, refer to textbook table 4-2.

3-14. The characteristic color of a newly filed surface of aluminum is
1. light gray
2. dark gray
3. white
4. dark silver

3-15. Which of the following metals, when it is in the unfinished state, has a smooth surface but is velvety in appearance?
1. Aluminum
2. Lead
3. Gray cast iron
4. White cast iron

3-16. Which of the following metals produces short chips when turned on a lathe?
1. Aluminum
2. Copper-nickel
3. Nickel-copper
4. Bronze

3-17. If a metal coated with green oxide reacts positively to the magnet test, it is probably composed of
1. nickel
2. copper
3. bronze
4. stainless steel

Learning Objective: Identify the steps performed and the results obtained when the spark test is used to identify metals. Textbook pages 4-14 through 4-17.

3-18. The reliability of the spark test in identifying metals depends on what conditions?
1. A properly lighted, draft-free testing location
2. A clean, dressed, 30- to 60-grain grinding wheel
3. The same wheel speed and feed pressure for both the known test specimen and the unknown sample
4. All of the above

3-19. In the spark test, why is it important to apply medium pressure against the grinding wheel?
1. Hard pressure will increase the temperature of the spark stream and the burst, giving an impression of much higher carbon content than that in the metal
2. Hard pressure will cause the spark stream to be diminished or eliminated, giving an impression that there is little or no carbon in the metal
3. Hard pressure will cause the color of the spark stream to change, throwing off identification of the metal entirely
4. Too little pressure can cause the spark stream to be too short and the volume of sparks to be too few for identification

3-20. If the spark stream has shafts and forks only, the metal under test is a
1. steel of high carbon content
2. steel of low carbon content
3. nickel alloy
4. molybdenum alloy

3-21. Which of the following metals gives off tiny blocks of brilliant white light?
1. Nickel alloy steel
2. High carbon steel
3. Molybdenum steel
4. Silicon alloy steel

3-22. The presence of a large quantity of white sparks indicates that the metal is
1. aluminum
2. cast steel
3. wrought iron
4. high carbon steel

Learning Objective: Identify the steps performed and the results obtained when the acid test is used to identify metals. Textbook pages 4-18 and 4-19.

3-23. How should you mix acid and water during preparations for making an acid test?
1. Pour the acid rapidly into the water
2. Pour the acid slowly into the water
3. Pour the water rapidly into the acid
4. Pour the water slowly into the acid

3-24. The first step that should be taken if acid is splashed on the hand of the man carrying out a nitric acid test is to
1. wash the skin with soap and water
2. send the man to sick bay
3. call a hospital corpsman or the medical officer
4. rinse the skin with a great deal of water

3-25. No reaction to the acid test indicates that the metal is one of the
1. molybdenum steels
2. nickel steels
3. stainless steels
4. tungsten steels

3-26. What metal gives a quick reaction and a blue-green color when a drop of nitric acid is applied to it?
1. Copper
2. Wrought iron
3. Aluminum
4. Nickel

3-27. A slow reaction and a pale green color after nitric acid is placed on a metal indicates that the metal probably contains the element
1. iron
2. copper
3. aluminum
4. nickel

Learning Objective: Identify the purposes and procedures of heat treating metals. Textbook pages 4-19 through 4-21.

3-28. Which effect results from the annealing of a metal?
1. Softening
2. Hardening
3. Improvement of corrosion resistance
4. Reduction of malleability

3-29. Slow cooling to room temperature is required for annealing
1. copper
2. brass
3. Stellite
4. Monel

3-30. What heat treating process is used to help retain the cutting edge on chisels?
1. Annealing
2. Process annealing
3. Normalizing
4. Hardening

3-31. What happens to a steel that is heated to its critical temperature?
1. Nothing if the steel is cooled gradually
2. Nothing if the steel is quenched in water or oil
3. The steel is always made more machinable
4. The grain structure and physical properties of the steel are changed noticeably

3-32. In which of the following heat treatments is the metal always cooled rapidly by quenching?
1. Annealing
2. Normalizing
3. Hardening
4. Tempering

3-33. Why is oil sometimes used as a quenching medium for the hardening of steel?
1. To slow down the rate of cooling
2. To prevent surface corrosion
3. To provide carbon impregnation of the surface
4. To produce greater hardness

3-34. In which of the following heat treatments is the temperature held below the critical temperature of the metal?
1. Normalizing
2. Hardening
3. Tempering
4. Case hardening

3-35. A metal is case hardened to produce greater
1. corrosion resistance
2. surface hardness
3. uniformity of internal stress
4. uniformity of carbon distribution

Learning Objective: Identify the characteristics and uses of standard hardness tests for metals. Textbook pages 4-21 through 4-27.

3-36. Standard hardness tests are based on a metal's resistance to penetration because penetration resistance
1. varies more than other types of hardness
2. is easier to measure accurately than other types of hardness
3. is the most important quality in metals for shipboard use
4. is the only possible definition of hardness

3-37. After you have placed the metal on the anvil of the Rockwell tester at the start of the test, what is the first thing that you should do?
1. Apply the minor load
2. Apply the major load
3. Turn the zero adjuster
4. Bring the test piece into contact with the penetrator

3-38. When you read the hardness number on the Rockwell tester dial, what are the positions of the major and minor loads?
1. The minor load rests on the specimen and the major load is raised
2. The major load rests on the specimen and the minor load is raised
3. Both loads rest on the specimen
4. Both loads are raised

3-39. If the hardness of a hard steel sample were measured in a Brinell tester, how large a load would be applied?
1. 500 kg
2. 720 kg
3. 1,500 kg
4. 3,000 kg

3-40. All EXCEPT which of the following hardness tests are based on the principle of penetration of a soft material by a harder one?
1. Rockwell test
2. Brinell test
3. Scleroscope test
4. Vickers test

3-41. A metal which offers considerable resistance to a file but can be filed by repeated effort is said to be
1. soft
2. hard
3. very hard
4. extremely hard

Learning Objective: Identify the basic characteristics of plastic materials. Textbook pages 4-25 through 4-31.

3-42. Which of the following substances is a basic element of most plastics?
1. Sodium chloride
2. Carbon
3. Aluminum
4. Silicon

3-43. Which of the following properties is a distinct characteristic of thermosettings?
1. Elasticity
2. Fire resistance
3. Tendency to absorb moisture
4. Malleability

3-44. Which of the following properties is a distinct characteristic of thermoplastics?
1. Hardening on exposure to heat
2. Softening on exposure to heat
3. Rapid conduction of heat
4. Effective insulation of heat

3-45. The thermoplastic most resistant to heat distortion is called
1. lucite
2. celluloid
3. nylon
4. polythene

3-46. You can use a fire test to differentiate between thermoplastics and thermosettings because a thermoplastic will burn readily and a thermosetting will resist burning.

3-47. You are working with a thin laminated plastic that breaks when bent sharply. In all likelihood, the layers of this plastic are made of
1. wood
2. glass
3. canvas
4. paper

3-48. Why is it important NOT to feed the work into a saw too fast when you are sawing plastics?
1. Since plastics do not take away heat generated by a saw, it is easy to burn work
2. Since plastics are sawed easily and quickly, it is easy to make an error
3. The saw might become gummy with plastic chips
4. The saw could be broken by the toughness of the plastic

Learning Objective: Identify machine shop power saw operations and safety precautions. Textbook page 5-1.

3-49. All EXCEPT which of the following are examples of machine shop work?
1. Cutting metal stock into pieces with a power saw
2. Drilling holes in flat metal stock with a drill press
3. Grinding cutting edges on cold chisels with a power grinder
4. Cutting metal pipe into pieces with an oxyacetylene torch

3-50. Which of the following is a safe practice to be observed by Machinery Repairmen while operating a power saw?
1. Wearing goggles or a face shield
2. Keeping hands as far from the saw blade as possible
3. Supporting ends of long pieces of work
4. Each of the above

3-51. Adjusting a power saw while it is in operation is an unsafe practice.

3-52. When performing machine work, you should first stress
1. accuracy
2. efficiency
3. safety
4. speed

Learning Objective: Identify the characteristics and operating procedures of power hacksaws. Textbook pages 5-1 through 5-3.

3-53. What is the minimum number of teeth that must be kept in contact with the work when a power hacksaw is used to cut thin steel sheet?
1. Eight
2. Two
3. Ten
4. Four

3-54. Which blade should be used on a power hacksaw to cut high-speed steel?
1. Coarse
2. Regular
3. Medium
4. Fine

3-55. In a power hacksaw, what type of feed mechanism automatically controls the feed when it is cutting through a hard spot in the work?
1. Hydraulic
2. Mechanical
3. Gravity
4. Hydraulic or gravity

3-56. Which of the following materials is cut on a power hacksaw without the use of a coolant?
1. Mild steel
2. Carbon steel
3. Cast iron
4. Brass

3-57. What is the recommended cutting speed when a power hacksaw is used to cut annealed tool steel?
1. 50 strokes per minute
2. 60 strokes per minute
3. 90 strokes per minute
4. 136 strokes per minute

3-58. Which of these practices is likely to cause blade breakage on a power hacksaw?
1. Applying coolant directly to the blade rather than to the work
2. Failing to align the cutting mark on the work with the blade
3. Starting the machine with the blade touching the work
4. Using a coarse blade to cut cast iron

Assignment 4

Power Saws and Drilling Machines; Offhand Grinding Tools

Textbook pages 5-4 through 6-11

Learning Objective: Identify the characteristics and operating procedures of continuous feed cutoff saws (bandsaws). Textbook pages 5-4 through 5-18.

4-1. What will the continuous feed cutoff saw do automatically as it completes each cut?
1. Raise the saw head
2. Stop the drive motor
3. Stop the drive motor and start the coolant motor
4. Stop the drive motor, start the coolant motor, and raise the saw head

4-2. After the saw band is installed and stretched properly, band tension is preserved during operation by the
1. tension handwheel
2. hydraulic system
3. V-belt
4. saw guides

4-3. Which of the following operations can be more easily performed on a tiltable blade bandsaw than on a tiltable table bandsaw?
1. Cutting long pieces
2. Contour cutting
3. Mitering
4. Polishing

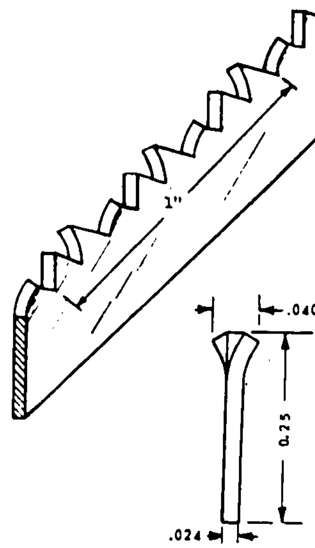


Figure 4A

- In answering Items 4-4 through 4-7, refer to figure 4A.

4-4. The saw segment has a pitch of
1. 7
2. 8
3. 9
4. 10

4-5. The saw segment has a width of
1. 0.024 in.
2. 0.040 in.
3. 0.25 in.
4. 1 in.

4-6. What is the gage of the saw segment?
1. 0.008 in.
2. 0.016 in.
3. 0.024 in.
4. 0.040 in.

4-7. What is the set of the teeth of the saw segment?
1. 0.008 in.
2. 0.020 in.
3. 0.032 in.
4. 0.040 in.

4-8. A sawband having which set pattern is used to cut metal pipes?
1. Raker
2. Straight
3. Wave
4. Any of the above

4-9. Which type of saw band is likely to be used for cutting solid slab stock?
1. Straight set, temper B
2. Wave set, temper B
3. Raker set, temper A
4. Straight set, temper A

4-10. The ends of a file band are joined with the
1. gate clip and tail gate
2. spacer and gate clip
3. segment and spacer
4. tail gate and segment

4-11. A file-band spacer is inserted between the
1. gate clip and tail gate
2. gate clip and back band
3. back band and file segment
4. file segment and tail gate

4-12. The abrasive material on a polishing band is carried on a backing of
1. treated paper
2. fabric
3. steel
4. rubber

4-13. The upper movable band guide of a bandsaw should clear the workpiece by
1. 1/4 in.
2. 1/2 in.
3. 1 1/2 in.
4. 2 in.

4-14. When using a file belt on a metal-cutting bandsaw, you replace the standard bandsaw guides with
1. insert type guides
2. roller type guides
3. lubricant-impregnated, fabric-faced guides
4. metal backup support strips

4-15. Excessive wear between a fabric polishing band and its guide is prevented by
1. cutting oil applied to the work
2. water coolant applied to the work
3. a graphite impregnated fabric-face on the backup strip
4. lubricating oil fed from the pressure system

4-16. What factor limits the gage of the saw band that can be used with a particular band tool machine?
1. Type of feed
2. Size of the band wheels
3. Type of band guide
4. Range of band speeds available

4-17. What factors influence the quality of work produced with a metal cutting bandsaw?
1. Band speed
2. Feed pressure
3. Band type
4. All of the above

4-18. Which bandsaw setup is correct for cutting cast steel with a maximum thickness of 2 inches?
1. Pitch 8, velocity 50 fpm
2. Pitch 10, velocity 50 fpm
3. Pitch 12, velocity 75 fpm
4. Pitch 14, velocity 125 fpm

4-19. Which setup is correct for cutting 3/4 inch cast iron on the metal-cutting bandsaw?
1. Medium feed pressure, velocity 200 fpm
2. Heavy feed pressure, velocity 200 fpm
3. Light feed pressure, velocity 225 fpm
4. Medium feed pressure, velocity 150 fpm

4-20. Which saw band should be used for a 3-inch radius cut in a 1-inch aluminum sheet?
1. 1/2 in. wide, 8-pitch band
2. 1/2 in. wide, 16-pitch band
3. 5/8 in. wide, 16-pitch band
4. 3/4 in. wide, 6-pitch band

4-21. Manual work feed on a bandsaw is generally used only on materials up to
1. 1/4 in. thick
2. 1/2 in. thick
3. 1 in. thick
4. 1 1/2 in. thick

4-22. Inadequate feed pressure on work in a bandsaw is likely to
1. break the teeth of the saw band
2. dull the saw band
3. snag the saw band
4. bind the saw band in the work

4-23. Which cut on a bandsaw requires the use of a butt welder?
1. Inside cut
2. Straight cut
3. Angular cut
4. Disk cut

4-24. The correct length for a replacement band on a two-wheel bandsaw is equal to
1. twice the circumference of one wheel plus the distance between wheel centers
2. twice the distance between wheel centers plus the circumference of one wheel
3. twice the sum of the distance between wheel centers plus the circumference of two wheels
4. three times the distance between wheel centers plus the circumference of one wheel

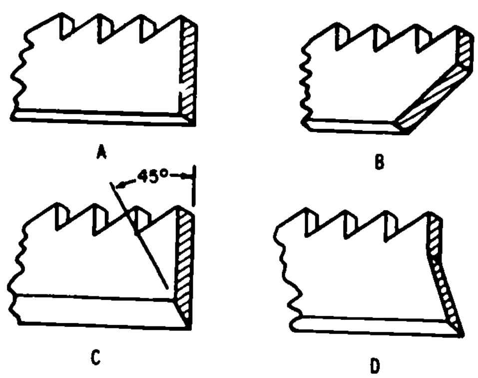


Figure 4B

4-25. Which of the saw band ends in figure 4B is properly prepared for splicing?
1. A
2. B
3. C
4. D

4-26. Which step follows the welding and grinding of a spliced saw band?
1. Annealing
2. Polishing
3. Quenching
4. Installing

4-27. Improper tracking of a newly installed saw band is corrected by adjustment of the
1. band tension
2. band guides
3. backup bearings
4. wheel tilt control

4-28. Before starting to cut metal with a bandsaw, what should you always be sure to adjust?
1. The band guide inserts so they clear the angle of cut
2. The worktable so its level matches the angle of cut
3. The upper band guide so it clears the work by 3/8 to 1/2 inch
4. All of the above

4-29. Which factors determine the tension to be applied to a newly installed saw band?
1. Set and gage of the band
2. Width and gage of the band
3. Width and temper of the band
4. Temper and gage of the band

4-30. Inadequate tension on a saw band is likely to cause
1. wandering
2. overheating
3. dulling
4. snagging

4-31. Corner holes are used on a complex saw workpiece to permit the use of
1. higher cutting speeds
2. a narrower band
3. a thinner gage
4. a smaller pitch

4-32. What type of feed is usually used for filing operations on a band tool machine?
1. Gravity
2. Hydraulic
3. Mechanical
4. Manual

Learning Objective: Identify characteristics of drills and drilling machines and procedures which an MR must follow during their use. Textbook pages 5-18 through 5-33.

4-33. To drill a hole using an upright drill press, you position the workpiece; using a radial drill press you position the drilling head.

4-34. For which of the following types of work is the sensitive drill press particularly useful?
1. Work requiring many holes to be drilled in a large piece of metal
2. Large castings requiring holddown clamps during drilling
3. Work requiring the operator to rely on his sense of touch to determine how the drill is cutting
4. Work requiring high-speed drilling in which vibrations are not harmful

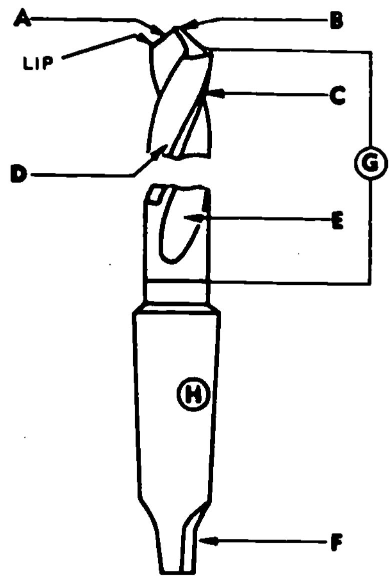


Figure 4C

- In answering items 4-35 through 4-39, refer to the twist drill shown in figure 4C.

4-35. The body is indicated by
1. B
2. F
3. G
4. H

4-36. The land consists of parts
1. A and B
2. C and D
3. E and F
4. G and H

4-37. The difference in the diameters of points CB-AD is to
1. facilitate removal of the drill from the spindle of a drill press
2. provide the body clearance that helps to reduce drilling friction
3. separate the flutes of the drill
4. accentuate the taper at the tip of the drill

4-38. What is the function of part A?
1. To cut the material as the drill is fed to the work
2. To provide side clearance for the drill body
3. To center the drill on the work
4. To provide chip clearance as the drill advances through the work

4-39. Which part is intended to simplify the removal of the drill from its mounting?
1. D
2. E
3. F
4. H

4-40. Which high-speed drill speed is preferred if alloy steel and cast iron are to be cut without changing drills or spindle speeds?
1. 50 fpm
2. 70 fpm
3. 100 fpm
4. 150 fpm

4-41. What is the approximate speed of the drill press with which you are drilling a 1/2-inch hole using a twist drill whose cutting speed is 88 fpm?
1. 90 rpm
2. 200 rpm
3. 670 rpm
4. 750 rpm

4-42. The feed of a drill press is expressed in
1. centimeters per revolution
2. inches per minute
3. thousandths of an inch per minute
4. thousandths of an inch per revolution

4-43. In drilling a hole through a metal plate, you should ease up on the feed pressure as the drill pierces the bottom of the plate.

4-44. To line up the center-punch mark on the workpiece exactly under the spindle when you are getting ready to drill, you will find it useful to
1. use a drill larger than the required size to make the lead hole
2. insert a dead center in the spindle socket
3. chisel a groove away from the punch-mark
4. use a counterbore pilot to guide your drill to the punchmark

4-45. You are to drill a large hole in a curved piece of brass. Your twist drill has a dead center as large as the center-punch mark which you made at the point to be drilled. What should you do to keep the twist drill on the punchmark when you start drilling the large hole?
1. Exert heavy pressure on the twist drill at the beginning and then ease up after you have drilled awhile
2. Insert a center drill in the spindle and make a center hole before you start drilling the large hole
3. Chisel two small grooves on opposite sides of the punchmark
4. Use angle plates to support your work

4-46. Which of the following practices is recommended for bringing a drill back to center after it has started off center?
1. Remove the drill from the hole and start again
2. Put pressure on the work from the side toward which the center is to be drawn
3. Chisel a groove on the side of the hole toward which the center is to be drawn
4. Recenter the work on the table under the drill

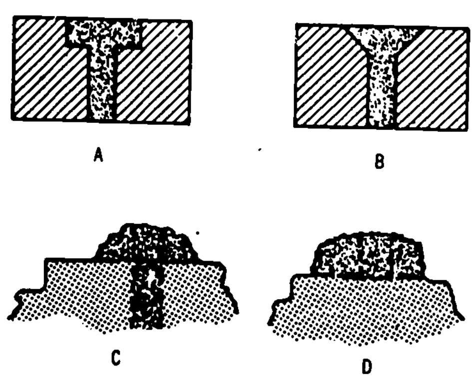


Figure 4D

4-47. The shaded portion of which part of figure 4D is removed by a countersink and drill?
1. A
2. B
3. C
4. D

4-48. What type of shank is found on a reamer designed for use in a drill press or lathe?
1. Straight
2. Tapered
3. Fluted
4. Splined

4-49. What is the rate of taper of a taper pin reamer?
1. 1/8 in. per foot
2. 1/4 in. per foot
3. 1/3 in. per foot
4. 1 in. per foot

4-50. Which of these items is NOT used to make an angular hole by the Watts method?
1. Floating chuck
2. Broach
3. Guide plate
4. Drill press

Learning Objective: Identify the proper procedures associated with the setting up and use of bench and floor mounted grinders. Textbook pages 6-1 and 6-2.

4-51. To become proficient at grinding small handtools and single-edged cutting tools, you must combine experience and practice with a knowledge of which of the following things?
1. The safety precautions to observe
2. The composition of the tool materials
3. How to install grinding wheels
4. All of the above

4-52. Before an operator starts to use a grinder, he should read the safety precautions which are posted on or near the grinder.

4-53. How should the tool rest on a bench or pedestal grinder be set up in relation to the grinding wheel?
1. 1/8 inch from the wheel face and level with the center of the wheel
2. 1/8 inch from the wheel face and level with the bottom of the wheel
3. 1/2 inch from the wheel face and level with the center of the wheel
4. 1/2 inch from the wheel face and level with the bottom of the wheel

4-54. Why does the operator of a grinder stand to one side of the machine while starting it?
1. To prevent getting injured if the wheel explodes
2. To be in position for side grinding
3. To check whether the toolrest is positioned properly
4. To inspect the condition of the grinding wheel

4-55. Normally a grinding wheel installed on a bench grinder does NOT exceed what thickness?
1. 1/4 in.
2. 7/8 in.
3. 1 in.
4. 1 1/2 in.

Learning Objective: Identify the basic characteristics and markings of grinding wheels. Textbook pages 6-2 through 6-11.

4-56. During grinding operations, the abrasive grains on a grinding wheel wear down causing the wheel to become dull.

4-57. The number A 100 D 15 V indicates a grinding wheel with a
1. coarse grit and dense structure
2. fine grit and open structure
3. hard bond and aluminum oxide abrasive
4. soft bond and silicon carbide abrasive

4-58. A hard grade grinding wheel possesses which of the following qualities?
1. A large number of strong abrasive grains
2. A small amount of bond surrounding the abrasive grains
3. Thick bond posts interlocking the abrasive grains
4. Thin bond posts offering greater resistance to grinding pressure

In answering items 4-59 through 4-61, select from column B the grinding wheel bond having characteristics listed in column A.

	A. Characteristic	B. Grinding Wheel Bond
4-59.	When used as cut-off wheel has cool cutting action	1. Rubber
		2. Shellac
		3. Vitrified
4-60.	Strong and shock resistant	4. Resinoid
4-61.	Not affected by oil, acid, or water	

4-62. Assume a diamond grinding wheel is marked D 100 C 50 B 1/8". This designation indicates the wheel
1. is 1/8 inch wide
2. contains manufactured abrasive
3. bond is not modified
4. has a grit size of 50

4-63. Which of the following changes will make a grinding wheel "act softer"?
1. Reduce the wheel diameter
2. Increase the wheel speed
3. Reduce the grain depth of cut
4. Increase the arc of contact of the wheel

4-64. In choosing a wheel for grinding high-speed steel, you should select one with an aluminum oxide abrasive.

4-65. Which type of bond should a wheel have for general purpose grinding?
1. Resinoid
2. Shellac
3. Rubber
4. Vitrified

4-66. What is the recommended grinding wheel for surface grinding a high-speed steel surface?
1. A60K8V
2. C36K5V
3. A46H8V
4. A46K8V

4-67. What can you tell about the condition of a grinding wheel which gives off a dull thud when tapped with a piece of hard wood?
1. It is out of round
2. It is safe to use
3. It has invisible cracks
4. It has been soaked in coolant and baked dry

4-68. A purpose of using flanges and cardboard or rubber washers in installing a grinding wheel on a spindle is to

1. remove play of the wheel on the spindle
2. distribute and even out the pressure on the wheel
3. hold the wheel on the spindle
4. grip the wheel so that it can rotate with the spindle

4-69. Which grinding wheel condition is caused by leaving part of the grinding wheel soaking in coolant?

1. Imbalance
2. Out-of-roundness
3. Grooved grinding surface
4. Cracks

Assignment 5

Offhand Grinding of Tools; Lathes and Attachments

Textbook Assignment: Pages 6-11 through 7-27

Learning Objective: Identify the terms and steps associated with the grinding of single point cutting tools. Textbook pages 6-11 through 6-20.

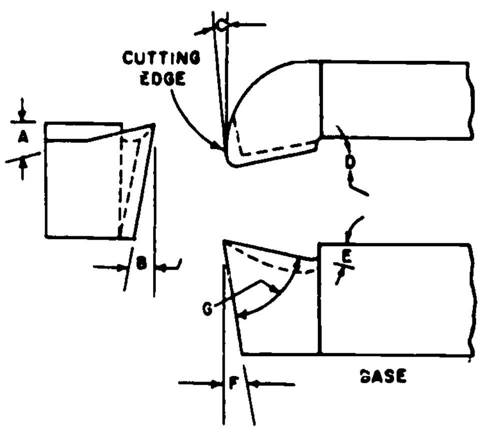


Figure 5A-Cutting tool terminology.

- In answering items 5-1 through 5-4, refer to figure 5A.

5-1. Which of the angles of the facing cutter is the side rake angle?
1. A
2. B
3. C
4. D

5-2. The back rake angle is a positive back rake angle.

5-3. Which angle of the facing cutter is the end relief angle?
1. A
2. B
3. F
4. G

5-4. Which angle of the facing cutter is the back rake angle?
1. D
2. E
3. F
4. G

In items 5-5 and 5-6, select from column B the cutting tool material that is generally used for the purpose given in column A.

	A. Purposes	B. Materials
5-5.	General Cutting	1. Stellite
5-6.	Fine finishing at low cutting speed	2. Carbon tool steel
		3. High-speed steel
		4. Tungsten carbide

5-7. In grinding a cutter bit, why should you remove as little of the bit as necessary to obtain the appropriate rake angles?
1. To enable the bit to cut with less chatter
2. To eliminate the use of a coolant during the grinding process
3. To enable the bit to absorb more of the heat that results during use
4. To eliminate the need for honing the bit

5-8. In grinding a tool bit, your last step should be to
1. hone it on an oilstone
2. grind it to obtain rake angles
3. grind it to obtain a radius on the end
4. grind it to obtain clearance angles

Learning Objective: Identify the characteristics and uses of engine lathe bits. Textbook pages 6-17 through 6-21.

5-9. What is the direction of feed of a roundnosed turning tool which is ground flat on top?
1. Right to left
2. Left to right
3. Either 1 or 2 above
4. Away from the lathe center

5-10. A left-hand or right-hand facing tool has a sharp point for the purpose of
1. cutting outside threads
2. cutting inside threads
3. machining square corners
4. machining necks

5-11. Which of these lathe tools is used to machine a groove?
1. Cutoff tool
2. Threading tool
3. Turning tool
4. Facing tool

5-12. Which of these lathe tools is usually ground in the shape of a left-hand turning tool?
1. Threading tool
2. Round-nosed turning tool
3. Cutoff tool
4. Boring tool

5-13. How should you vary the feed and depth of cut of the tool bit in going from roughing to finishing work?
1. Increase feed and depth of cut
2. Decrease feed and depth of cut
3. Increase feed of cut and decrease depth of cut
4. Decrease feed of cut and increase depth of cut

5-14. In an engine lathe tool properly ground from a roughing tool to a finishing tool, the side clearance angle will be
1. 0°
2. 4°
3. 5° to 9°
4. 6° to 10°

Learning Objective: Identify the characteristics of cutting tools used in turret lathes. Textbook pages 6-21 through 6-23.

5-15. The back rake angle for a turret lathe tool used for cutting cast iron should be ground to
1. 5°
2. 8°
3. 10°
4. 12°

5-16. Side and front relief angles on steel turret lathe tools should all be ground to
1. 12°
2. 10°
3. 8°
4. 5°

5-17. The cutters used on turret lathes are usually ______ (larger, smaller) than the cutters used on engine lathes because a turret lathe is designed to remove ______ (large, small) quantities of metal rapidly.
1. Smaller, small
2. Larger, small
3. Smaller, large
4. Larger, large

5-18. The clearance ground on the shank of a carbide-tipped cutter should be slightly larger than the clearance angle to be ground on the tip in order that
1. both shank and tip may be ground simultaneously
2. the grinding wheel can be kept off the shank when the tip clearance angle is being ground
3. better support can be given to the tip by the shank
4. the shank can remain intact if the tip breaks

Learning Objective: Identify the characteristics and uses of shaper and planer tools. Textbook pages 6-23 through 6-25.

5-19. The relief angles of shaper and planer cutting tools are equal to the relief angles of lathe cutting tools.

5-20. What is the main purpose of the side rake angle in a cutting tool?
1. To prevent the tool from roughing the surface of the workpiece
2. To prevent binding of the tool on the workpiece surface
3. To reduce the power needed to make a cut
4. To reduce the probability of the cutting edge breaking

5-21. Which shaper or planer cutting tool is NOT given a back rake angle?
1. Downcutting tool
2. Shovel nose tool
3. Shear tool
4. Gooseneck tool

5-22. Downcutting in either right- or left-hand direction may be accomplished by a
1. roughing tool
2. shovel nose tool
3. shear tool
4. cutting-off tool

5-23. Which cutting tool is normally used as a finishing tool?
1. Shear tool
2. Downcutting tool
3. Cutting-off tool
4. Shovel nose tool

Learning Objective: Identify and select some of the purposes of the principal parts of the engine lathe. Textbook pages 7-1 through 7-15.

5-24. In a routine machining operation with an engine lathe the workpiece is rotated about a horizontal axis.

5-25. The swing of a 16-inch by 8-foot lathe is 8 feet.

5-26. What are the upper surface shapes of the ways of a typical lathe?
1. Square or curved
2. Curved or V-shaped
3. V-shaped or flat
4. Flat or square

5-27. Driving power is applied to the workpiece on a lathe through the
1. tailstock spindle
2. headstock spindle
3. crossfeed screw
4. dead center

5-28. How many spindle speeds can be obtained on a lathe that has a 6-position headstock cone pulley?
1. 6
2. 12
3. 18
4. 24

5-29. Why does a headstock spindle have a hole running through its center?
1. To permit the bars or rods to pass through the spindle
2. To improve the cooling of the gearbox
3. To permit thorough lubrication of the spindle bearings
4. To dissipate heat from the cutting tool

5-30. The distance between centers on an engine lathe may be adjusted by moving the
1. headstock
2. tailstock
3. tailstock and headstock
4. carriage

5-31. Rotation of the tail. handwheel on an engine lathe resu in a movement of the tailstock
1. base
2. spindle
3. top
4. clamp bolt

5-32. Which part of an engine lathe is attached directly to the crossfeed slide of the carriage?
1. Cutting tool
2. Workpiece
3. Tool post
4. Compound rest

5-33. When an engine lathe is used for milling, the workpiece is mounted on the
1. headstock and tailstock centers
2. tailstock spindle
3. carriage
4. face plate

5-34. Which of the following operations on an engine lathe is usually performed with the carriage locked in position?
1. Turning
2. Facing
3. Boring
4. Drilling

5-35. The gear trains in the apron of an engine lathe are driven by the
1. control rod
2. lead screw
3. reverse rod
4. feed rod

5-36. Friction clutches are operated on the apron to engage and disengage the
1. headstock speed
2. tailstock
3. power-feed mechanism
4. hand-feeds

5-37. The half-nuts in the apron are engaged with the lead screw when an engine lathe is used for
1. turning
2. facing
3. boxing
4. threading

5-38. Which of the following parts of an engine lathe receive power through the feed rod?
1. Headstock spindle and longitudinal feed mechanism
2. Longitudinal feed mechanism and crossfeed mechanism
3. Crossfeed mechanism and tailstock spindle
4. Tailstock spindle .nd headstock spindle

5-39. In lathes that have no feed rod, the feedrod function is performed by the
1. rack and pinion
2. splined lead screw
3. worm gear
4. V-ways

5-40. When an engine lathe is used for thread cutting, the number of threads per inch is determined by the relationship between the speeds of the
1. drive motor and spindle
2. spindle and feed rod
3. lead screw and feed rod
4. lead screw and spindle

5-41. The pitch of threads cut by an engine lathe is changed by changing the
1. spindle drive speed
2. pitch of the lead screw
3. lead screw gears
4. size of the rack pinion

5-42. A workpiece in an engine lathe must be threaded with 15 threads per inch. If the lead screw has 10 threads per inch, which of the following sets of gears can be used?
1. 15-tooth spindle gear and 10-tooth screw gear
2. 15-tooth spindle gear and 30-tooth screw gear
3. 15-tooth spindle gear and 30-tooth screw gear
4. 20-tooth spindle gear and 30-tooth screw gear

5-43. Idler gears in a gear train are used to
1. adjust the speed ratio between the driving and driven gears
2. couple the driven gear to the driving gear without affecting the speed ratio
3. stabilize the speed of the driven gear with changes in load
4. permit reversal of the driving gear rotation

5-44. The stud gear of the lead screw gear train of an engine lathe meshes with the
1. screw gear
2. idler gear
3. spindle gear
4. small compound gear

5-45. A rod was threaded with a pitch of 7 on an engine lathe equipped with a lead screw having a pitch of 5. Which of the following pairs of change gears were used to thread this rod?
1. 15-tooth gear and a 10-tooth gear
2. 21-tooth gear and a 18-tooth gear
3. 35-tooth gear and a 24-tooth gear
4. 42-tooth gear and a 30-tooth gear

5-46. Which ratio is commonly used for the compound gears of a lathe?
1. 1:1
2. 2:1
3. 8:3
4. 10:1

5-47. A lathe you are assigned to work on is equipped with a quick-change gear box and a sliding compound gear. One lever on the gearbox has three possible positions while the other lever has six positions. How many screw speeds are available?
1. 3
2. 6
3. 18
4. 36

5-48. Feeds on the quick-change gear box are identified in terms of ten thousandths of an inch per
1. second
2. minute
3. spindle revolution
4. feed rod revolution

Learning Objective: Identify some of the characteristics and uses of major lathe attachments and accessories. Textbook pages 7-15 through 7-27.

5-49. The toolpost is used only to provide rigid support for the toolholder.

5-50. Which type of work is done on a lathe with the help of a knurling tool attachment?
1. Trimming an oversize metal workpiece
2. Roughing the surface of a round metal workpiece
3. Threading the outside of a solid metal workpiece
4. Threading the inside of a solid metal workpiece

5-51. What type of lathe chuck is used to hold workpieces that have irregular cross sections?
1. Scroll chuck
2. 4-jaw chuck
3. Standard collet chuck
4. Hexagonal collet chuck

5-52. What type of lathe chuck can be used to automatically center round workpieces of many sizes?
1. Scroll chuck
2. 4-jaw chuck
3. Standard collet chuck
4. Hexagonal collet chuck

5-53. What type of lathe chuck is preferred for precision turning of small work?
1. Combination chuck
2. Universal chuck
3. Independent chuck
4. Draw-in collet chuck

5-54. What is the angle of the points of Morse-tapered lathe centers?
1. 30°
2. 45°
3. 60°
4. 75°

5-55. A tailstock center differs from a headstock center mainly in
1. shank taper
2. point taper
3. metal hardness
4. diameter

5-56. Which mark is used to identify the tailstock center of an engine lathe?
1. Longitudinal cut
2. Dimple
3. Circular groove
4. Punched depression

5-57. Which workholding devices are used in combination?
1. Tailstock center and clamp dog
2. Lathe dog and driving plate
3. Faceplate and collet chuck
4. Collet chuck and tailstock center

5-58. A function of the center rest used on an engine lathe is to support
1. long workpieces
2. workpieces being machined to a noncircular cross section
3. especially heavy workpieces
4. workpieces that have no indented centers

5-59. A follower rest is used on a lathe to prevent
1. springing of the work
2. improper centering of the work
3. irregular feed pressure in thread cutting
4. out-of-round turning of the work

5-60. The guide bar support of a taper attachment is connected directly to the
1. carriage saddle
2. headstock
3. tailstock
4. lathe bed

5-61. A carriage stop may be used on an engine lathe to remove the need for
1. manual operation at the end of a cut
2. individual measurements on duplicate parts
3. setup measurements made directly on the workpiece
4. variable rates of feed across a workpiece

5-62. The depth of a cut made by a milling attachment on an engine lathe is controlled by the
1. lead screw
2. cross feed
3. tailstock position
4. longitudinal feed

5-63. Which type of lathe is constructed in such a way that a piece can be removed from its bed to accommodate work of large diameter?
1. General purpose screw cutting precision lathe
2. Toolroom lathe
3. Gap lathe
4. Bench lathe

Assignment 6

Basic Engine Lathe Operations; Advanced Engine Lathe Operations

Textbook Assignment: Pages 8-1 through 9-25

Learning Objective: Identify the proper procedures required to set up an engine lathe for basic lathe operations. Textbook pages 8-1 through 8-14.

6-1. What should you use to remove burrs in the tailstock spindle of a lathe?
1. A grinder
2. A tail center coated with lapping compound
3. A 60° taper reamer
4. A Morse taper reamer

6-2. To positively align lathe centers without making a test cut, you should use a
1. test bar and indicator
2. center gage
3. Morse taper gage
4. steel rule between the centers

6-3. The tool used to true a lathe center is fed to the work by means of the
1. longitudinal feed
2. compound-rest feed
3. cross feed
4. rotation of the tool post swivel

6-4. Which of the following lathe operations is performed with the workpiece mounted on the carriage?
1. Turning
2. Facing
3. Threading
4. Milling

6-5. Centers are usually made in finished round workpieces with a
1. lathe
2. punch
3. hand drill
4. drill press

6-6. What should be the large diameter of the center hole in a lathe workpiece that has an outside diameter of 2 1/2 inches?
1. 5/32 in.
2. 3/16 in.
3. 5/16 in.
4. 7/16 in.

6-7. In a properly formed center hole, the lathe centers rest against the
1. bottom of the drilled hole
2. sides of the countersunk hole
3. inner rim of the countersunk hole
4. outer rim of the countersunk hole

6-8. Driving torque is usually applied to a mandrel through
1. the live center
2. a lathe dog
3. a collet chuck
4. a drill chuck

6-9. Which of the following workpieces would probably be turned on a soft mandrel?
1. A plastic gear blank for a 1 1/4-inch shaft
2. A brass pulley for a 1/2-inch shaft
3. A steel collar with a nonstandard inside diameter
4. a steel spindle with a nonstandard outside diameter

6-10. Why must the surface of a standard mandrel be lubricated before the work is mounted?
1. To prevent damage to the work when the mandrel is removed
2. To permit free rotation of the mandrel on the tail center
3. To permit free rotation of the work on the mandrel
4. To prevent tilting of the work on the tapered surface

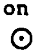
6-11. A hollow workpiece that has two cylindrical surfaces, each with a different axis, may be turned on an engine lathe by mounting the work on
1. a universal chuck
2. a gang mandrel
3. an eccentric mandrel
4. an arbor

6-12. The centering of work in a 4-jaw independent chuck should be checked by
1. taking a light test cut
2. holding a piece of chalk against the rotating work
3. bringing the tail center against the face of the work
4. locating the axis of the cylindrical portion with a combination square

6-13. Which of the following precautions should you take when you chuck a thin-walled cylinder in a lathe?
1. Insert paper or shim stock under the chuck jaws
2. Expand the chuck jaws against the bore of the work
3. Use only enough jaw pressure to prevent slipping
4. Adjust the jaws individually to prevent distortion

6-14. When work is held in a draw-in collet chuck for precise machining, what is the maximum diameter of the collet?
1. 0.00001 in.
2. 0.0001 in.
3. 0.001 in.
4. 0.002 in.

6-15. When a universal chuck is used on the headstock spindle of a lathe, the live center should be replaced with a
1. pipe center
2. collet chuck
3. rag
4. block of wood

6-16. Suppose that after you prick punch the center of a workpiece that you mount it on the faceplate of a lathe. You then use the dead center to determine whether or not the workpiece is on center. If it is on center, which of the following illustrations represents what you would see on the end of the workpiece?
1. 
2.
3. 
4. 

6-17. Which of the following attachments must you use to accurately machine thread the length of a long shaft?
1. Follower rest
2. Center rest
3. Either 1 or 2 above
4. Collet chuck

Learning Objective: Identify the steps involved and measurements taken in basic engine lathe machining. Textbook pages 8-15 through 8-26.

6-18. Cutting speeds on a lathe are stated in units of
1. revolutions per minute
2. feet per minute
3. inches per revolution
4. thousandths per revolution

6-19. Which metal is machined at the highest cutting speed?
1. Bronze
2. Cast iron
3. Aluminum
4. Tool steel

6-20. The time required to perform a rough turning operation can often be shortened by reducing the
1. cutting speed and increasing the feed
2. depth of cut
3. longitudinal feed and increasing cutting speed
4. lubricant flow

6-21. Which of the following lathe operations requires the highest cutting speeds?
1. Rough facing
2. Rough turning
3. Finish turning
4. Thread cutting

6-22. Which lubricant is preferred for threading a cast iron workpiece?
1. Kerosene
2. Lard oil
3. Soapy water
4. Graphite solution

6-23. What is the most common source of chatter in lathe operation?
1. High cutting speed
2. Inadequate lubrication
3. Slow feed
4. Excessive depth of cut

6-24. What is normally the direction of feed of a facing cutter?
1. Toward the headstock
2. Toward the tailstock
3. Toward the lathe center
4. Away from the lathe center

6-25. When a facing cut is made in a workpiece on an engine lathe, which feed moves the tool?
1. Cross feed
2. Longitudinal feed
3. Compound-rest feed
4. Both longitudinal feed and compound-rest feed

6-26. A depth of cut of 0.040 inch reduces the diameter of a lathe workpiece by
1. 0.020 in.
2. 0.040 in.
3. 0.080 in.
4. 0.120 in.

6-27. Shoulders are commonly located with a parting tool to eliminate the need for
1. using a pointed turning tool
2. facing the shoulder
3. cutting a fillet
4. measuring during the rough turning

● For items 6-28 through 6-31, assume that you are operating a lathe to drill a 1-inch diameter hole through a thick metal block. To complete the job, you plan to use a pilot drill, a finish drill, a boring tool, and a reamer. When drilling, you will use a cutting speed of 80 fpm.

6-28. Failure to feed enough pressure to a drill is likely to cause the
1. drill to get too hot
2. drill to chatter
3. workpiece to be spoiled
4. drill to be broken

6-29. The flutes of your finish drill are cleared occasionally by
1. running the pilot drill through the hole
2. flushing with lubricating oil
3. backing the drill out
4. reducing the cutting speed of the drill

6-30. What is the approximate diameter of the hole after you finish the drilling and boring?
1. 3/4 in.
2. 7/8 in.
3. 15/16 in.
4. 63/64 in.

6-31. What should you do to keep the reamer from overheating?
1. Pause for a moment when you sense the reamer is getting too hot, increase the feed, and lubricate the reamer
2. Chill the reamer before starting the operation and use a cutting speed of less than 40 fpm
3. Chill the reamer before you begin, use a cutting speed of 60 fpm, and pause if you sense that it is getting too hot
4. Lubricate the reamer and use a cutting speed of 40 fpm

6-32. Which process is used to bring a hole to finished size when high accuracy is required?
1. Boring
2. Coring
3. Drilling
4. Reaming

6-33. Which feed is used when a boring bar is mounted between centers on a lathe?
1. Cross feed
2. Longitudinal feed
3. Compound-rest feed
4. Longitudinal feed and cross feed

6-34. What is the relationship betwen the spindle speeds for roughing and knurling a particular metal?
1. Roughing speed is twice knurling speed
2. Roughing speed is equal to knurling speed
3. Roughing speed is half knurling speed

6-35. When you refinish a damaged lathe center, at what angle should you set the compound rest in relation to the lathe spindle axis?
1. 15°
2. 30°
3. 60°
4. 90°

Learning Objective: Identify the terminology and procedures associated with the turning and boring of tapers. Textbook pages 9-1 through 9-7.

621

6-36. A 4-inch tapered workpiece has a diameter of 3 inches on one end and 3 3/8 inches on the opposite end. What is the taper of this workpiece?
1. 0.025 in. per in.
2. 1/8 in. per ft
3. 3/8 in. per ft
4. 1 1/8 in. per ft

6-37. What is the taper of a shaft that has an included angle of 3° between opposite surfaces?
1. 1/8 in. per ft
2. 3/8 in. per ft
3. 5/8 in. per ft
4. 1 3/8 in. per ft

6-38. Which taper has the greatest included angle between opposite sides?
1. Brown and Sharpe taper
2. Milling machine taper
3. Morse taper
4. Pipe taper

6-39. Which taper is commonly used on drill shanks?
1. Morse taper
2. Jarno taper
3. Brown and Sharpe taper
4. Pipe taper

6-40. What is the diameter of the small end of a No. 3 Morse taper?
1. 0.4 in.
2. 0.778 in.
3. 0.938 in.
4. 1.25 in.

6-41. What method is normally used to cut large angles of taper on short workpieces?
1. Taper-attachment method
2. Compound-rest method
3. Offset-center method
4. Simultaneous-feed method

6-42. What instrument is used to obtain accurate angular settings of the compound rest?
1. Steel rule
2. Micrometer
3. Vernier bevel protractor
4. Center gage

6-43. When a taper attachment is used with a lathe, the depth of cut is adjusted with the
1. shoe clamp
2. crossfeed screw
3. compound-rest feed screw
4. longitudinal feed screw

6-44. Which of the following devices should you use to check the accuracy of a bored taper?
1. A bevel protractor
2. An inside micrometer
3. A dial-indicator cylinder gage
4. A plug gage

6-45. The boring of a blind tapered hole is usually preceded by
1. taper reaming
2. drilling to the small diameter of the taper
3. drilling to slightly less than the small diameter of the taper
4. drilling to the large diameter of the taper

Learning Objective: Identify the terminology associated with machine threads. Textbook pages 9-8 through 9-10.

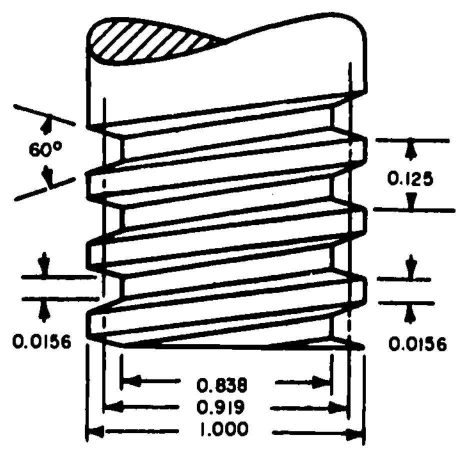


Figure 6A

In answering items 6-46 through 6-48, refer to figure 6A.

6-46. What is the pitch of the screw thread shown in figure 6A?
1. 0.0625 in.
2. 0.125 in.
3. 0.162 in.
4. 0.919 in.

6-47. Which dimensions of the thread shown in figure 6A are equal?
1. Crest width and root width
2. Crest width and half pitch
3. Crest diameter and root diameter
4. Root diameter and pitch diameter

6-48. What is the major diameter of the thread shown in figure 6A?
1. 0.419 in.
2. 0.838 in.
3. 0.919 in.
4. 1.000 in.

- Use the following information in answering item 6-49.

You desire to locate a bolt that has the following characteristics:
A. Nominal thread diameter = .75 inch
B. Threads per inch = 16
C. Series = American National coarse
D. Class = 3

6-49. What is the thread designation?
1. .75-ANC-16-3
2. 16-3-ANC-.75
3. 3-NC-3/4-16
4. 3/4-16-NC-3

Learning Objective: Identify terms and procedures associated with the cutting of V-form threads. Textbook pages 9-10 and 9-11.

6-50. What information do you need to cut a V-form screw thread?
1. Pitch and width of the thread
2. Straight depth of the thread
3. Slant depth of the thread
4. All of the above

6-51. What is the slant depth of an American Standard (external) thread having 10 threads per inch?
1. 0.061343 in.
2. 0.0708 in.
3. 0.1000 in.
4. 0.6250 in.

6-52. Which instrument is usually used to set a threading tool true with the work?
1. Bevel protractor
2. Screw pitch gage
3. Center gage
4. Thread dial indicator

6-53. Failure to set a threading tool perpendicular to the work axis results in an incorrect
1. pitch
2. pitch diameter
3. helix angle
4. angle of thread

Learning Objective: Identify basic characteristics of the following threads: Acme, square, buttress, pipe, tapered pipe, and straight pipe. Textbook pages 9-11 through 9-14.

6-54. What is the depth of an Acme thread that has a pitch of 0.25 inch?
1. 0.125 in.
2. 0.135 in.
3. 0.260 in.
4. 0.510 in.

6-55. What is the cutting edge width of a tool used to cut square threads, 1/4-inch pitch, on a screw?
1. 0.125 in.
2. 0.127 in.
3. 0.500 in.
4. 0.502 in.

6-56. What is the basic depth of a buttress thread which has 10 threads per inch?
1. 0.07140 in.
2. 0.06627 in.
3. 0.03318 in.
4. 0.01631 in.

6-57. Straight pipe threads are usually used to join sections of pipe that will carry water to fire hoses.

Learning Objective: Identify thread measurement tools and their uses. Textbook pages 9-15 and 9-16.

6-58. Which of the following tools is used to measure an internal thread?
1. Thread micrometer
2. Plug gage
3. Go/no-go ring gage
4. Adjustable thread snap gage

6-59. The most accurate test of threads is made by using which of the following tools?
1. Thread micrometer
2. Adjustable thread snap gage
3. Go/no-go ring gage
4. Micrometer and three wires

Learning Objective: Identify the procedures involved in cutting threads on a lathe. Textbook pages 9-16 through 9-25.

6-60. The lead screw on your lathe has a pitch of 10. If you want to cut a screw having 32 threads per inch, what must be the ratio of the headstock spindle to the lead screw?
1. 3.2:1
2. 4.0:1
3. 1:3.2
4. 1:4.0

6-61. A very light cut is usually scribed on the surface of work that is to be threaded to provide a check on the accuracy of the
1. thread angle
2. root diameter
3. pitch
4. crest width

6-62. The best lubricant for thread cutting in steel is lard oil.

6-63. What is the most common chamfer used to finish the thread of a capscrew?
1. 15°
2. 30°
3. 45°
4. 60°

6-64. Threads on a lathe workpiece are most easily terminated at a
1. drilled hole
2. shoulder
3. groove
4. milled slot

Assignment 7

Turret Lathes and Turret Lathe Operations; Milling Machines and Milling Machine Operations

Textbook Assignment: Pages 10-1 through 11-57

Learning Objective: Identify the basic components and workpiece set up procedures for turret lathes. Textbook pages 10-1 through 10-13.

7-1. Due to a system of automatic stops, cutting tools, once adjusted and set in motion, need no further attention from the operator.

7-2. What type of workpiece is usually machined on a bar machine?
1. Large-diameter casting
2. Nonsymmetrical workpiece
3. Long stock
4. Workpiece that requires only one machining operation

7-3. On a ram-type turret lathe, the tools on the hexagonal turret are fed into the work by the movement of the
1. ram
2. cross-slide
3. saddle
4. chuck

7-4. On a turret lathe, the independent carriage-stop adjustments for each of the six positions are located on the
1. dial clips
2. feed rod
3. stop roll
4. stop rod

7-5. Metal chips should be removed often from the working parts of turret lathes with
1. compressed air
2. a vacuum cleaner
3. a handbrush or rag
4. a magnet

7-6. Spring [illegible] collets are used on a [illegible]
1. gripp[illegible]
2. holding [illegible] cylinders
3. feeding bar stock
4. aligning any wor[illegible]

7-7. Which workholding device is commonly used to hold finish-bored cylindrical work for finish turning or facing?
1. Expanding plug arbor
2. Expanding bushing arbor
3. Spring-type collet
4. Independent chuck

7-8. A 4-jaw independent chuck requires separate adjustment of each jaw in order to hold the work.

Learning Objective: Identify the procedures used to grind and install cutting tools. Textbook pages 10-14 through 10-16.

7-9. A coolant is normally used during the grinding of
1. carbide cutters
2. high-speed steel cutters
3. Stellite cutters
4. carbide and Stellite cutters

7-10. Dipping a cutter blade in coolant during dry grinding causes the cutting edge to
1. soften
2. become brittle
3. crack
4. warp

7-11. A groove is sometimes ground along the cutting edge on the top face of a cutter bit to
1. minimize chatter
2. break up chips
3. produce a double chip
4. reduce the strain on the point

7-12. What is the preferred method of controlling chip run-off from a cutter bit?
1. Adjust the rake angles and feed
2. Groove the top face of the cutter
3. Groove the front face of the cutter
4. Adjust the clearance angles

7-13. The tip angles of a cutter must be changed during regrinding if the cutter is used in a
1. rear tool post
2. square turret
3. rocker type tool holder
4. shim type tool holder

7-14. On the hexagonal turret of a lathe, overhead turning cutters are centered on the work with the aid of a
1. center gage
2. spirit level
3. scale
4. plumb bob

Learning Objective: Identify the lathe attachments and procedures used for turning bar stock. Textbook pages 10-17 through 10-21.

7-15. The rollers that back up a bar turner cutter are mounted on the
1. square turret
2. hexagonal turret
3. reach-over tool post
4. carriage

7-16. Back rests without rollers are usually used to back up bar turner cutters that are used on
1. cast iron
2. mild steel
3. brass
4. aluminum

7-17. On stock machined in a bar turner, which of the following conditions usually produces the smoothest finish?
1. Light cut with the rolls ahead of the cutter
2. Light cut with the rolls behind the cutter
3. Heavy cut with the rolls ahead of the cutter
4. Heavy cut with the rolls behind the cutter

7-18. When work is machined on a lathe, the stream of coolant should be applied to the
1. tip of the tool
2. surface of the work below the tool
3. surface of the work above the tool
4. shank of the tool

7-19. Which of the following forming cutters can be used to quickly produce an odd- or curved-shape?
1. Forged
2. Dovetail
3. Circular
4. All of the above

7-20. On a turret lathe, the holder of a dovetail forming cutter is mounted on the
1. saddle or ram
2. hexagonal turret
3. cross-slide
4. square turret

Learning Objective: Identify steps involved in cutting tapers and threads. Textbook pages 10-21 through 10-25.

7-21. Solid adjustable and self-opening dies are usually used for threading in preference to solid tools unless
1. high cutting speeds are required
2. machining time must be minimized
3. smoothly finished threads are required
4. very coarse cuts are taken

7-22. When a turret lathe is used for threading with a single-point tool, angular cutter feed is usually used for
1. tapered threading cuts
2. roughing cuts
3. finishing cuts
4. roughing and finishing cuts on straight or tapered work

7-23. On a turret lathe, the taper produced by a taper attachment is determined by the setting of the
1. automatic cross feed
2. guide plate
3. automatic longitudinal feed
4. spindle speed

7-24. Assume that you are tooling a shoulder stud on a turret lathe. The diameters of the shoulder stud have been turned. What is the next step in the operation?
1. Cutting off the workpiece
2. Cutting the thread
3. Machining the radius on the end of the workpiece
4. Forming the front diameter

7-25. The tooling setup for small and medium lot production of taper studs is almost identical. However, there is a difference in the way the taper is produced between small and medium lot production jobs. What is the difference?
1. In small lot production, the taper is usually formed with a standard wide cutter, and the taper is later ground to make the matched cuts accurate; in medium lot production, the taper is turned with the cross-slide taper attachment giving more accuracy to the taper
2. In small lot production, the taper is turned with the cross-slide taper attachment; in medium lot production, the taper is usually formed with the standard wide cutter, and the taper is later ground to make the matched cuts accurate
3. In small lot production, the taper is usually formed with a standard wide cutter, and the taper is later turned with a cross-slide taper attachment; in medium lot production, the taper is formed with the cross-slide taper attachment alone
4. In small lot production, the taper is formed with the cross-slide taper attachment, and the taper is later ground to make the matched cuts accurate; in medium lot production, the taper is turned with the cross slide taper attachment, and the taper is later formed with a standard wide cutter to make the cross cuts more accurate

---

Learning Objective: Identify basic terms and procedures associated with vertical turret lathes. Textbook pages 10-25 through 10-32.

7-26. Vertical turret lathes are particularly suitable for machining
1. several workpieces simultaneously
2. large, heavy workpieces
3. nonsymmetrical workpieces
4. small precision parts

7-27. The main rails of a vertical turret lathe are perpendicular to the
1. side head
2. feed rods
3. upright bedways
4. lead screw

7-28. The power feed to the main head of a vertical turret lathe permits
1. vertical movement of the main head
2. vertical and horizontal movements of the main head
3. vertical and diagonal movements of the main head
4. vertical, horizontal, and diagonal movements of the main head

● In answering items 7-29 through 7-31, assume that you are preparing to cut tapers using a 30-inch Bullard vertical turret lathe.

7-29. How many degrees in the direction of the angle being turned do you swivel the main turret head in cutting a 53° taper?
1. 16°
2. 32°
3. 47°
4. 53°

7-30. In cutting a 30° to 45° taper, you swivel the main turret head in a direction opposite that of the angle being turned.

7-31. How many degrees do you swivel the main turret head in turning a 40° taper?
1. 5°
2. 10°
3. 15°
4. 20°

Learning Objective: Describe the construction, uses, and limitations of the three common types of knee and column milling machines used by the Navy. Textbook pages 11-1 through 11-6.

7-32. When only one type of knee and column milling machine can be installed, what type will it usually be?
1. Bed type
2. Vertical spindle type
3. Universal type
4. Plain type

7-33. What feature distinguishes the universal milling machine from the vertical spindle and plain type?
1. Three directions of table movement
2. The table swivels on the saddle
3. More rigid construction
4. Horizontal spindle axis

7-34. Which milling machine is most efficient for taking deep cuts at rapid ratea of feed and speed?
1. Plain milling machine
2. Universal milling machine
3. Universal milling machine with high-speed vertical milling attachment
4. Vertical spindle milling machine

7-35. Which movement is possible on a vertical spindle milling machine but not on other knee and column machines without the aid of an attachment?
1. Horizontal movement of the vertical spindle
2. Vertical movement of the vertical spindle
3. Vertical movement of the table
4. Horizontal movement of the table

7-36. To do a face milling job quickly, you should select
1. a plain milling machine
2. a universal milling machine
3. a vertical spindle milling machine
4. either a plain or universal milling machine

Match the component in column A with the description in column B

	A. Component	B. Description
7-37.	Column	1. The support for the table and saddle
7-38.	Knee	
7-39.	Saddle	2. A movable support to which the workpiece is fastened
7-40.	Table	
		3. Used to position the work closer to, or farther from the column
		4. The principal support to which the knee is attached

7-41. The cutting tool of a milling machine is attached to what component(s)?
1. A workholder
2. The overarm dovetails
3. Aroor supports
4. The arbor

Learning Objective: Describe operations and applications of the principal milling machine attachments. Textbook pages 11-6 through 11-11.

7-42. Which vise may be used to mill work at an angle in a horizontal plane?
1. Plain vise or universal vise
2. Swivel vise or universal vise
3. Swivel vise or plain vise
4. Universal vise, plain vise, or swivel vise

7-43. Which milling machine attachment would be good for holding stock which is being made into a gear?
1. Swivel vise
2. Universal vise
3. Dividing head
4. Center rest

7-44. How many turns of the crank of the worm shaft of a dividing head are required to rotate the spindle one revolution?
1. 20
2. 40
3. 60
4. 75

7-45. Which milling machine attachment allows you to machine the smallest lead on a worm?
1. Enclosed driving mechanism
2. Long and short lead attachment
3. Wide range divider
4. Low lead driving mechanism

7-46. Which milling machine attachment permits the greatest variation in the angle at which the cutter can be set?
1. Universal milling attachment
2. Compound vertical milling attachment
3. High-speed vertical milling attachment
4. Circular milling attachment

7-47. Which milling machine attachment is used in cutting keyways?
1. Compound vertical milling attachment
2. Circular milling attachment
3. Slotting attachment
4. Rack milling attachment

Learning Objective: Perform plain indexing. Textbook pages 11-12 through 11-16.

7-48. Rapid indexing is accomplished by using a/an
1. index head sector
2. direct index plate
3. universal dividing head
4. compound index plate

7-49. Eight complete turns of the index crank will move a universal dividing head spindle
1. 1/40 of a revolution
2. 1/8 of a revolution
3. 1/5 of a revolution
4. 1/3 of a revolution

7-50. You are given a piece of work that is to be divided into 12 parts and you have a 24-hole index plate available. How far should you revolve the index crank on the index plate to make each division?
1. 1 complete turn and 3 holes
2. 1 complete turn and 8 holes
3. 3 complete turns and 3 holes
4. 3 complete turns and 8 holes

7-51. You are given a piece of work that is to be divided into 960 parts and you have a 24-hole index plate available. How far should you revolve the index crank on the index plate to make each division?
1. 1 hole
2. 2 holes
3. 6 holes
4. 12 holes

7-52. You are given a piece of work that is to be divided into 48 parts and you have an 18-hole index plate available. How far should you revolve the index crank on the index plate to make each division?
1. 3 holes
2. 5 holes
3. 13 holes
4. 15 holes

7-53. How far should you turn the index crank on a 27-hole index plate to move the circumference of the work 11°?
1. 1 turn and 2 holes
2. 1 turn and 4 holes
3. 1 turn and 6 holes
4. 1 turn and 8 holes

7-54. Assume that you are to divide a piece of work into 37 parts and your index angle is given in minutes. Which index plate should be used if approximate divisions are acceptable?
1. 11-hole plate
2. 15-hole plate
3. 18-hole plate
4. 23-hole plate

7-55. How far should you turn the index crank on a 21-hole index plate to move the circumference of the work 180 minutes?
1. 3 holes
2. 6 holes
3. 7 holes
4. 9 holes

Learning Objective: Perform differential indexing. Textbook pages 11-16 through 11-20.

● Information for items 7-56 and 7-57: You are differential indexing for 72 divisions. Your selected number is 70.

7-56. How many holes in a 21-hole circle of the index plate do you turn the index crank?
1. 2
2. 4
3. 8
4. 12

7-57. What is the ratio of the required spindle gear to the required driven gear?
1. 1:2
2. 2:1
3. 7:8
4. 8:7

7-58. Within the wide range divider, the revolution ratio of the small index crank superimposed on that of the worm wheel is such that the resulting ratio causes the dividing head spindle to rotate at the rate of
1. one turn to every one hundred turns of the large index crank
2. one turn to every four thousand turns of the small index crank
3. one hundred turns for each turn of the large index crank
4. four thousand turns for each turn of the small index crank

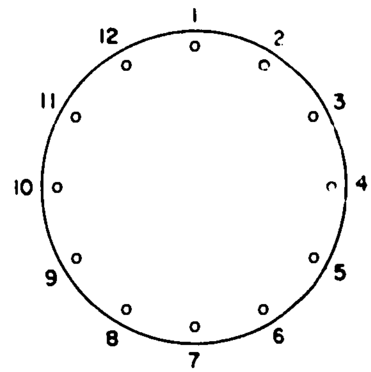


Figure 7A.-12-hole circle.

- In answering item 7-59, refer to figure 7A.

7-59. What hole in the 12-hole circle should be covered by the right-hand arm of the index head sector if the required number of holes for indexing is 5 and the index pin is inserted into hole 1?
1. Hole 5
2. Hole 6
3. Hole 7
4. Hole 8

Learning Objective: Describe and explain the applications of the principal types of cutters. Textbook pages 11-20 through 11-35.

7-60. To what parts of a milling machine are cutters attached?
1. A plain arbor or arbor yoke
2. A screw arbor
3. A collet or adapter
4. An arbor or directly to the spindle

7-61. The principal characteristic of plain milling cutters is that they
1. have teeth on the circumference only
2. have only straight teeth
3. always have helical teeth
4. have teeth on the end only

7-62. Cutters with helical teeth are superior to cutters with straight teeth in which of the following ways?
1. They can take a wider cut
2. They produce a smooth, shaving action
3. They reduce the shock produced when a cutter begins operation
4. They have all of the above advantages

7-63. What type of cutter can be used in pairs to mill slots?
1. Half-side milling cutt
2. Side milling cutter with inter-locking teeth
3. Metal slitting saw
4. Half-side milling c h a metal slitting saw between

7-64. What kind of cutter is ground thinner toward its center to provide clearance between the cutter and the work?
1. Plain milling cutter
2. Side milling cutter
3. Metal slitting saw
4. Shell and mill

7-65. Which of the following statements is most accurate about the use of an arbor with a solid end mill?
1. Use of an arbor is necessary, since a solid end mill has no shank
2. Use of an arbor is not necessary, even though a solid end mill has no shank, since it is mounted on a small central stud for use
3. Use of an arbor is not necessary, since the cutter head can be attached to a shank
4. Use of an arbor is not necessary, since the body and shank of a solid end mill are one piece

7-66. What kind of cutter is used for milling an enclosed space inside a slot?
1. End mill
2. Metal slitting saw
3. T-slot cutter
4. Dovetail cutter

7-67. Woodruff keyseat cutters less than 1 1/2 inches in diameter are usually mounted on a/an
1. arbor
2. permanent shank
3. stub arbor
4. arbor support

7-68. What type of cutter can be manufactured locally and can be made to cut a variety of forms?
1. Fly cutter
2. Corner-rounding cutter
3. Convex cutter
4. Gear hob

7-69. The fastest cutting is done with cutters having
1. fine teeth and minimum diameter
2. coarse teeth and minimum diameter
3. fine teeth and maximum diameter
4. coarse teeth and maximum diameter

7-70. Fine-tooth cutters are best for
1. eliminating chatter when cutting heavy stock
2. milling at high speed
3. milling thin stock
4. producing a continuous chip

Match the cutter or accessory in column A with the type of arbor in column B.

	A. Accessory		B. Arbor
7-71.	Drill chuck	1.	Taper adapter
7-72.	Straight shank drill	2.	Fly cutter arbor
7-73.	Locally manufactured single-point cutter	3.	Stub arbor
		4.	Cutter adapter

621

Assignment 8

Milling Machines and Milling Operations (continued) and Shapers, Planers and Engravers

Textbook Assignment: Pages 11-35 through 12-22

Learning Objective: Describe the principle of milling operations. Textbook pages 11-35 through 11-57.

8-1. In mounting a cutter on an arbor, you replace the arbor nut on the arbor after the
1. draw-in bolt locking nut is inserted
2. cutter and arbor bearing are in place
3. saddle is locked
4. arbor support is in place

8-2. In setting up a milling machine, how do you gage that the surface of the work is just touching the teeth of the cutter?
1. Insert a thin feeler gage between the teeth of the cutter and the work
2. Raise the work until the cutter stops it, then back off slightly
3. With the cutter turning slowly, bring the work up until it stops the cutter
4. With a piece of paper on the work surface, raise it until the rotating cutter pulls the paper

8-3. Face milling must be done with
1. shell end mills only
2. face milling cutters only
3. the surface being milled perpendicular to the cutter axis
4. the feed of the work in the vertical direction

8-4. When cutting a square on the end of a piece of stock, what type of milling is used?
1. Climb milling
2. Up milling
3. Climb milling when the stock is vertical and up milling when the stock is horizontal
4. Up milling when the stock is vertical and climb milling when the stock is horizontal

8-5. How large a square can be cut from a piece of 1-inch round stock?
1. 1.414 inches
2. 1.0 inch
3. 0.800 inch
4. 0.707 inch

8-6. How large a hexagon can be cut from 1-inch round stock?
1. 0.707 inch
2. 0.866 inch
3. 1.0 inch
4. 1.555 inches

8-7. In milling a 3-inch hexagon, what is the minimum diameter cylinder that you can use?
1. 2.60 inches
2. 3.24 inches
3. 3.47 inches
4. 4.67 inches

8-8. To mill a hexagon on the end of round stock, the work is positioned by using an index head.

8-9. What attachment changes the rotary motion of the spindle to a reciprocating motion?
1. Index head attachment
2. Slitting saw attachment
3. Slotting attachment
4. Sawing attachment

Match the material in column A with the type of slitting saw in column B

	A. Material	B. Slitting Saw
8-10.	Fine tooth	1. Brass
8-11.	Coarse tooth	2. Plastic
8-12.	Staggered tooth	3. Thick steel
		4. Thin steel sheet

8-13. To obtain a hooked cutting face on a tap, the flutes are cut with a
1. concave cutter
2. convex cutter
3. slitting saw
4. fluting cutter

8-14. Some straight reamer flutes are not cut at equal angles apart. Why is this done?
1. It gives a better cutting edge
2. The angle of the flutes is not important
3. It prevents chatter
4. For all of the above reasons

8-15. When a fly cutter is being used as a formed cutter, why should you rough out the surface with ordinary cutters?
1. To obtain a smoother surface
2. To save time since the fly cutter is not as fast
3. To reduce the load on the fly cutter arbor
4. For all of the above reasons

8-16. The cutting tools used for boring on a milling machine resemble what other type of cutter?
1. End mill
2. Shell end mill
3. Slot cutter
4. Fly cutter

8-17. The size of the hole being bored is most easily and accurately adjusted in what way?
1. By changing the bed position
2. By changing the boring bar
3. By offsetting the boring bar
4. By moving the cutter in the boring bar

Learning Objective: Identify common milling machine attachments. Textbook pages 11-58 through 11-60.

Match the function in Column B with the type of attachment in Column A.

	A. Attachment	B. Function
8-18.	Tool maker's knee	1. Increases the speed of rotation of the cutter
8-19.	High speed universal attachment	2. Changes the direction of the cutter axis to the vertical position
8-20.	Vertical milling attachment	3. Permits milling of arcs and circles
8-21.	Circular milling attachment	4. Permits milling compound angles

---

Learning Objective: Apply appropriate factors to determine cutting speeds, feeds, and coolants. Textbook pages 11-60 through 11-64.

8-22. What will be the most immediate result of overheating a cutter?
1. A clogged cutter
2. Worn machine gear
3. Excessive vibration
4. Dulled blades

8-23. High-speed steel cutters can be operated at higher speeds than carbon steel cutters because they have greater
1. corrosion resistance
2. heat resistance
3. tensile strength
4. compression strength

8-24. What is the proper spindle speed for a 3/4-inch cutter that is to be run at 65 fpm?
1. 257 rpm
2. 306 rpm
3. 331 rpm
4. 341 rpm

8-25. What is the proper spindle speed for a 1/2-inch cutter that is to be run at 85 fpm?
1. 611 rpm
2. 649 rpm
3. 679 rpm
4. 688 rpm

8-26. What is the cutting speed of a 1 1/4-inch cutter running at 140 rpm?
1. 43.2 fpm
2. 44.3 fpm
3. 45.8 fpm
4. 46.9 fpm

8-27. While a narrow slot was being milled, the cutter of the end mill was broken. What is the most likely reason for the breakage?
1. A deep cut was being taken with a high rate of feed
2. The cutter was overheated
3. The work was improperly supported
4. A high rate of speed was being used together with a low rate of feed

8-28. To mill a metal to a fine finish, you should use a
1. high cutter speed and a slow feed rate
2. low cutter speed and a high feed rate
3. low cutter speed and a low feed rate
4. high cutter speed and a high feed rate

8-29. What rate of speed and feed are best for roughing?
1. Low speed and low feed
2. Low speed and high feed
3. High speed and high feed
4. High speed and low feed

8-30. Which metal should be machined without a liquid coolant?
1. Brass
2. Aluminum
3. Cast iron
4. Nickel

8-31. In milling with a periphery cutter, the flow of the coolant should be directed onto the
1. work
2. cutter
3. work and cutter
4. work, cutter, and spindle

Learning Objective: Describe the operation of the horizontal boring mill. Textbook pages 11-64 through 11-72.

8-32. Which of the following factors may be used to determine the size of a horizontal boring mill?
1. Maximum thickness of work the table will hold
2. Maximum size boring and facing head that will fit on the spindle sleeve
3. Largest diameter flycutter it can accommodate
4. Largest diameter boring bar that the machine is designed to receive

8-33. When you set up a boring and facing head, which of the following is a safety precaution to be followed before shifting the spindle back-gear to neutral?
1. Engage the spindle over-running clutch
2. Shift the spindle back-gear to neutral and adjust the spindle back-gear
3. Extend the spindle of the boring and facing head from the sleeve
4. Rotate the sleeve by jogging until the heavy end of the facing head is down

8-34. When you are using a horizontal boring mill equipped with a boring and facing head, a continuous ratcheting sound will warn you when
1. the feed is engaged and the slide is clamped
2. either the feed is engaged or the slide is clamped
3. the feed is disengaged
4. the slide is unlocked

8-35. What is the fastest slide feed rate that can be set on a combination boring and facing head?
1. 0.005 inch per revolution
2. 0.010 inch per revolution
3. 0.050 inch per revolution
4. 0.100 inch per revolution

8-36. Machinery Repairmen aboard Navy repair ships perform drilling and reaming operations differently on the radial drill than on the horizontal boring mill.

8-37. Which bearing area is labeled for you to use as the reference point when aligning a split-sleeve bearing on a horizontal boring mill?
1. Horizontal flange
2. Front face
3. Circumferential groove
4. Lining

8-38. You desire to cut a pipe thread to certain design specifications. What mechanism on a horizontal boring mill should you adjust to achieve the desired lead?
1. Cutter advance
2. Gear train
3. Driving gear lever
4. Feed clutch

8-39. During a threading operation on a horizontal boring mill, what happens when you shift the thread lead lever to standard position while making a pass?
1. The feed stops
2. The feed clutch is disengaged
3. Thread timing is lost
4. All of the above happen

Learning Objective: Identify the basic parts and setup procedures for the shaper. Textbook pages 12-1 through 12-5.

8-40. The size of a shaper is determined by the
1. size of the cutter
2. depth of the stroke
3. size of the motor
4. length of stroke

8-41. Which mechanism is used to drive the ram of a crank shaper?
1. Hydraulic cylinder
2. Rack and spur gear
3. Rocker arm assembly
4. Pushrod and camshaft

8-42. What happens when you set the crankpin of a shaper offcenter?
1. The rocker arm is prevented from moving when you turn the crank gear
2. The rocker arm moves when you turn the crank gear
3. The ram is positioned
4. The table is positioned horizontally

8-43. How should you change the direction of feed on a shaper?
1. Adjust the feed connecting link
2. Change the nut on the saddle
3. Lift and rotate the pawl on the table feed mechanism 1/2 turn
4. Turn the feed screw

8-44. What should you do to prevent the tool bit from digging into the work on the return stroke of a vertical cut?
1. Fix a hooked tool in the toolhead
2. Lower the work on the return stroke
3. Raise the tool bit on the return stroke
4. Swivel the clapper box to the side

8-45. What is the function of shaper hold-downs?
1. To hold small work in the vise
2. To hold the ram in position
3. To lock the table in place
4. To hold the tool bit in the tool-slide

Learning Objective: Perform computations for and identify steps required in performing basic shaper operations. Textbook pages 12-5 through 12-13.

8-46. You have to cut a large deep channel in a piece of steel and the channel will require machining on the bottom and both sides. Which toolholder should you use?
1. Straight toolholder
2. Swivel head toolholder
3. Spring toolholder
4. Extension toolholder

8-47. To make an internal cut you use a/an
1. left-hand toolholder
2. gang toolholder
3. swivel head toolholder
4. extension toolholder

8-48. Under which circumstance(s) is a shaper likely to chatter?
1. When the tool bit is not set properly
2. When the work is loose in the vise
3. When a formed cutter is in a left-hand holder
4. In all of the above situations

8-49. What is the purpose of the gear shift lever?
1. To change the rate of feed
2. To change the speed of the motor
3. To raise the worktable
4. To change the speed of the shaper

8-50. What percentage of the total machining time is accounted for by the cutting stroke?
1. 50%
2. 60%
3. 70%
4. 80%

8-51. If the cutting speed in a shaping operation is 40 feet per minute and the length of stroke is 12 inches, what is the approximate number of strokes per minute?
1. 16
2. 20
3. 24
4. 30

8-52. When you change from a carbon steel tool bit to a high-speed steel tool bit, you multiply the cutting speed by a factor of
1. 1.5
2. 2.0
3. 2.5
4. 3.0

8-53. When a rectangular block is being shaped, which edge or surface is shaped last?
1. A face surface
2. A side edge
3. An end edge
4. Any of the above

8-54. In shaping a rectangular block, you insert paper strips between each corner of the block and the parallels to
1. test for firmness of seating
2. prevent scratching surfaces
3. prevent chipping corners
4. establish a clearance

8-55. You are cutting a keyway that extends part way up a shaft. What do you do to prevent the tool from binding at the end of the cutting stroke?
1. Use a special tool bit
2. Drill a hole at the end of the slot
3. Allow the tool bit to free itself on the return stroke
4. Raise the tool bit at the end of the cutting stroke

8-56. What device is used to line up an internal keyway in a gear with the axis of the gear?
1. Micrometer
2. Feeler gage
3. Calipers
4. Dial indicator

8-57. When you are machining an irregular surface on a shaper, how do you move the cutting tool?
1. Horizontally, by hand or power feed
2. Horizontally, by hand
3. Vertically, by hand or power feed
4. Vertically, by hand

8-58. In rough cutting a rack tooth, when do you lock the graduated collar on the toolslide feed screw of the shaper?
1. After you start the shaper
2. After the tool has been brought into contact with the work
3. Before the work is clamped to the vice or table
4. Before you set the graduated dial on the crossfeed screw to zero

8-59. What should you do first to rough out a piece that will have concave radii?
1. Control by hand the feed on all the rough cuts
2. Use automatic feed to make a series of horizontal cuts
3. Use automatic feed to make a series of vertical cuts
4. Set the horizontal and vertical feed to give the proper contour

---

Learning Objective: Identify parts of planers and steps involved in planer operation. Textbook pages 12-14 through 12-17.

8-60. The size of a planer is designated according to the
1. length of stroke
2. size of the work which it will accommodate
3. size of the motor
4. length of the bed

8-61. The columns on a planer are designed to support the
1. work
2. table
3. gear train
4. crossrail

8-62. The crossrail on a planer is designed to hold the
1. table
2. vise
3. toolheads
4. column

8-63. How should you compensate for normal wear on a planer?
1. Use heavier oil
2. Install shims
3. Adjust the gibs
4. Install new lead screws

8-64. What motions are controlled by the crossrail feed box?
1. Vertical movement of the crossrail
2. Horizontal movement of the toolhead
3. Vertical movement of the toolhead
4. Both 2 and 3 above

8-65. A planer is making a series of horizontal cuts on a piece of metal. What would you do to stop temporarily only the crossfeed of the tool bit?
1. Shut off the electricity
2. Move the start-stop range selector to the off position
3. Move the directional feed control to the neutral position
4. Disengage the horizontal feed clutch

Learning Objective: Identify parts and operating procedures of the pantograph. Textbook pages 12-18 through 12-[illegible].

8-66. What point should you check to determine if a pantograph is level?
1. Spindle
2. Copyholder
3. Worktable
4. Support base

8-67. On which arm of the pantograph is the tracing stylus attached?
1. Tracer arm
2. Upper bar
3. Lower bar
4. Connecting link

8-68. The blocks are set on the pantograph bars according to the
1. spindle speed desired
2. size of the copy
3. size of the finished work
4. reduction desired

8-69. What is the highest spindle speed obtainable on a model 3-U pantograph?
1. 5,000 rpm
2. 10,000 rpm
3. 15,000 rpm
4. 20,000 rpm

8-70. On a model 3-U pantograph, how is the spindle speed adjusted?
1. By varying the voltage
2. By using the gear shift
3. By adjusting the drive belts
4. By using the flow control lever

8-71. On the copyholder of a pantograph, what is the main purpose of the stop screws?
1. To keep the pantograph arms from going too far
2. To keep the cutter from going too far
3. To hold the work in the copyholder
4. To align the copy with the table

8-72. What is the reduction if length of copy is 1.8 inches and the length of the finished job is 1.2 inches?
1. 0.6
2. 1.5
3. 2.2
4. 3.0

8-73. Cutter speed on a pantograph is affected by which of the following factors?
1. Depth of cut
2. Rate of feed
3. Type of material being cut
4. All of the above

8-74. In general, which of the following materials requires the slowest operation of the cutter on a pantograph?
1. Hardwood
2. Cast iron
3. Hard bronze
4. Hardened alloy steel

Assignment 9

Shapers, Planers and Engravers (continued) and Precision Grinding Machines and Metal Buildup

Textbook Assignment: Pages 12-22 through 14-12

Learning Objective: Identify the technology associated with, and the steps involved in grinding pantograph cutters. Textbook pages 12-22 through 12-29.

9-1. When do you rotate a conical point against a grinding wheel?
1. When rough grinding
2. When grinding chip clearance
3. When grinding flat to center
4. When grinding away stock on the clearance angle

9-2. What is the main purpose in tipping off a cutter point?
1. To increase the life of the cutter's sharpness
2. To obtain special effects when engraving
3. To remove the burr
4. To save grinding any clearance

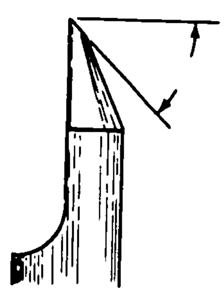


Figure 9A.

9-3. What name is usually given to the indicated angle in figure 9A?
1. Clearance angle
2. Rake angle
3. Cutting angle
4. Relief angle

9-4. What determines the size of the indicated angle in figure 9A?
1. Spindle speed
2. Size of the cutter
3. Material to be cut
4. Size of the copy

9-5. Three- or four-sided cutters are normally used for engraving on
1. large, heavy work
2. very soft work
3. small steel pieces
4. wood

Learning Objective: Identify special purpose accessories for the pantograph and how those accessories are used. Textbook pages 12-30 through 12-33.

9-6. A concave forming guide may be used for engraving on which of the following shapes of workpieces?
1. Flat
2. Concave
3. Convex
4. All of the above

9-7. To what part of a pantograph is the forming guide attached?
1. Copyholder
2. Worktable
3. Forming bar
4. Vertical feed control

9-8. When you use a circular copy plate to engrave various single letters on a number of similar workpieces, you must align worktable, copyholder, and aligning stops for each workpiece.

9-9. When engraving a dial face that has 10 graduations, how many degrees will you rotate the rotary table between cuts?
1. 36°
2. 45°
3. 60°
4. 72°

Learning Objective: Select or determine the appropriate grinding wheel speeds, feeds, and coolants for specified situations. Textbook pages 13-1 through 13-4.

9-10. Which wheel speed is most common in precision grinding?
1. 1,000 to 2,000 fpm
2. 3,000 to 4,000 fpm
3. 5,500 to 9,500 fpm
4. 9,500 to 10,500 fpm

9-11. What is the surface speed of a grinding wheel that has a circumference of 18 inches and is set on a spindle whose speed is 4,000 rpm?
1. 4,500 fpm
2. 6,000 fpm
3. 7,500 fpm
4. 10,000 fpm

9-12. In most cases, a speed less than the speed indicated on a grinding wheel is the most effective cutting speed.

9-13. Which of the following effects indicates that the grinding wheel is rotating too slowly?
1. Wheel surface wears too fast
2. Wheel slides over the work
3. Work overheats
4. All of the above effects

9-14. Rust formation on machine parts prohibits the use of plain water as a grinding coolant.

9-15. Which of the following characteristics is not desirable in a cutting fluid?
1. High cooling capacity
2. High viscosity
3. Low flammability
4. Noncorrosiveness

Learning Objective: Identify the major parts of the surface grinder and state the use(s) of each part. Textbook pages 13-4 through 13-8.

9-16. The wheelhead motor of the surface grinder shown in figure 13-3 of your textbook is located in the base of the grinder.

9-17. In operating the surface grinder shown in figure 13-3 of your textbook, you have to stop the hydraulic motor in order to stop the traverse tables.

9-18. How does the cross traverse table on the grinding machine shown in figure 13-3 of your textbook move in relation to the wheel spindle for each stroke of the sliding table?
1. It moves in a direction parallel to the spindle with a feed slightly less than the thickness of the grinding wheel
2. It moves in a direction perpendicular to the spindle with a feed equal to the diameter of the grinding wheel
3. It moves in a direction parallel to the spindle with a feed equal to the thickness of the workpiece
4. It moves in a direction perpendicular to the spindle with a feed equal to the thickness of the grinding wheel

9-19. How does the sliding table on the grinding machine shown in figure 13-3 of your textbook move in relation to the wheel spindle?
1. It moves at right angles to the spindle with a feed short enough to prevent the workpiece from clearing the grinding wheel
2. It moves at right angles to the spindle with a feed long enough to allow the workpiece to clear the grinding wheel
3. It moves in a direction parallel to the spindle with a feed short enough to prevent the workpiece from clearing the grinding wheel
4. It moves in a direction parallel to the spindle with a feed long enough to allow the workpiece to clear the grinding wheel

9-20. Both the cross traverse and the sliding table on the grinding machine shown in figure 13-3 of your textbook may have the distances of their strokes adjusted and may be manually operated.

9-21. To alter the depth of cut on a workpiece being ground on the grinding machine shown in figure 13-3 of your textbook, you have to adjust the height of the position of the workpiece.

9-22. Which condition is NOT essential to holding work in place in a grinding machine equipped with an electromagnetic chuck?
1. The control lever must be in the ON position
2. Work must be made of ferrous metal
3. Work must be in contact with two poles of the chuck
4. The chuck must be equipped with a demagnetizer

9-23. To finish grind a piece of hardened steel that you have just rough ground, the longitudinal traverse speed of the worktable should be
1. increased from 10 fpm to 25 fpm
2. increased from 25 fpm to 40 fpm
3. decreased from 40 fpm to 25 fpm
4. decreased from 60 fpm to 40 fpm

9-24. What is the purpose of rough grinding a single-point tool bit on a bench grinder before setting the tool bit in a jig for finish grinding?
1. To save time
2. To increase cutting accuracy
3. To lessen chances of heat distortion
4. To lessen stress on finishing wheel

Learning Objective: Identify basic parts and operating procedures associated with cylindrical grinders. Textbook pages 13-8 through 13-10.

9-25. Which component of a surface grinder is NOT found on a cylindrical grinder?
1. Coolant system
2. Hydraulic power unit
3. Cross traverse table
4. Wheelhead

9-26. The platen on the cylindrical grinder permits an operator to
1. feed the grinding wheel into the work
2. slide the table back and forth across the face of the grinding wheel
3. move the grinding wheel above or below the center of the work
4. swivel the grinding wheel

9-27. On a cylindrical grinder, the distance of longitudinal traverse per revolution of the workpiece is equal to
1. a distance slightly less than wheel thickness
2. the length of the workpiece
3. the width of the workpiece
4. the diameter of the grinding wheel

9-28. In setting up work on a cylindrical grinder, when do you start the coolant flow?
1. When you begin grinding the work
2. Just before you dress the wheel
3. After you have made a cleanup cut
4. As soon as you start the spindle motor and hydraulic pump

Learning Objective: Identify the basic parts and set up procedures of the tool and cutter grinder. Textbook pages 13-10 through 13-15.

9-29. Grinding wheels can be mounted on both ends of the spindle on the wheelhead of a tool and cutter grinder.

9-30. Suppose that you are to grind a milling cutter with the tool and cutter grinder shown in figure 13-8. Which of the following combinations of devices do you use to hold the cutter on the grinder?
1. Workhead and footstock
2. Sliding table and taper table
3. Taper table and tooth rest
4. Wheelhead and tooth rest

9-31. Which features of a tool and cutter grinder make it possible for a Machinery Repairman to run the machine while standing at the most favorable position for seeing the work?
1. A left-hand footstock and a right-hand footstock
2. A control handwheel at the front of the grinder and another at the back
3. A column-mounted wheelhead and a double-ended spindle
4. A taper table and a sliding table, used together

9-32. A cutter is considered dull and in need of sharpening when
1. the primary lands are worn and rounded
2. the primary lands become more than 0.035 inch wide
3. the wear lands become less than 0.010 inch wide
4. the wear lands become more than 0.010 inch wide

9-33. What can you do to correct a situation in which the grinding wheel begins to burn the cutting edge of a cutter you are grinding?
1. Have the grinding wheel rotate toward the cutting edge
2. Reduce the grinding speed
3. Increase the grinding speed
4. Decrease the feed

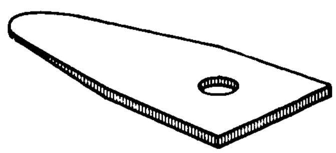


Figure 9B.

9-34. Which of the following types of cutters should be sharpened by using the blade shown in figure 9B?
1. Straight-fluted reamer
2. Helical tooth cutters
3. Side milling cutters
4. Straight tooth plain milling cutter

9-35. To grind a cutter requiring a 5° clearance angle, how much should you raise a 6-inch diameter straight wheel head?
1. 0.002 inch
2. 0.026 inch
3. 0.250 inch
4. 0.261 inch

Learning Objective: Identify the terminology and procedures associated with the grinding of cutters, mills and taps. Textbook pages 13-15 through 13-22.

9-36. What occurs when you sharpen a plain milling cutter with helical teeth using a cup type wheel head when the wheel head is swiveled to 89°?
1. Maximum compensation for wheel wear is accomplished
2. The wheel head axis is positioned in the same horizontal plane as the axis of the footstock centers
3. The end of the cutter clears the opposite cutting face
4. The end of the cutter does not clear the opposite cutting face

9-37. While you are sharpening a milling cutter on a grinding wheel, the gradual wearing down of the wheel grinding surface will cause inconsistencies among the various cutting teeth. How can you prevent this effect?
1. Proportionately increase grinding pressure from one tooth to the next
2. Use a cutting fluid or paste compound
3. Alternate the grinding process by rotating the cutter 180° after completing each revolution
4. Rotate the cutter 90° for each complete revolution

9-38. Assume that you are grinding a stagger tooth milling cutter. Which of the following steps should you perform first?
1. Align the wheelhead axis with the footstock centers
2. Bring one cutter tooth in contact with the tooth rest blade
3. Bring one cutter tooth in contact with the wheel
4. Swing the master electrical switch to ON

9-39. When you sharpen the peripheral teeth on an end mill using a grinder, it will be necessary to change or remove the cutter's size markings.

9-40. To prevent rapid dulling of cutting edges, the corners of the teeth on an end mill are ground with a 45° corner angle.

9-41. Which part of an involute gear cutter's tooth must be ground before you can sharpen the actual cutting surface?
1. Face
2. Back
3. Corner
4. Periphery

9-42. A formed cutter may be sharpened while set up between centers on a mandrel in the same manner as for sharpening a/an
1. plain milling cutter
2. stagger tooth cutter
3. end mill cutter
4. angular cutter

Learning Objective: Identify characteristics of proper honing operations. Textbook pages 13-22 through 13-24.

9-43. Which of the following polished surfaces is/are the result of honing?
1. The inside of a cylinder
2. The cutting edges of a reamer
3. The surface of a magnetic chuck
4. The teeth of an end cutting mill

9-44. What pattern should you see on a properly honed surface?
1. Ovals
2. Circles
3. Straight lines
4. Crosshatch lines

Learning Objective: Identify the capabilities of the spray metalizing process. Textbook pages 14-1 through 14-4.

9-45. What is the basic principle of spray metalizing?
1. Spraying powdered metal on a melted surface
2. Spraying melted droplets of metal on a surface
3. Coating the metallic surface with a metallic spray paint
4. Wiping a heated metal surface with metallic alloys having a low melting point

9-46. Which of the following is a common purpose of spray metalizing to form a hard surface?
1. Plating objects to improve weather resistance
2. Increasing the tensile strength of a machine part
3. Creating a wear-resistant bearing surface
4. Improving the appearance of cast metal objects

9-47. What two thermal spray processes are most likely to be used by an MR 3 or MR 2?
1. Wire oxygen-fuel spray and wire-consumable electrode spray
2. Plasma-arc spray and powder oxygen-fuel spray
3. Wire-consumable electrode spray and plasma-arc spray
4. Powder oxygen-fuel spray and wire oxygen-fuel spray

9-48. Which of the following safety precautions is applicable to the use of the cleaning solvents used in preparing metal surfaces for spray metalizing?
1. Provision for adequate ventilation
2. Wearing of an approved respirator
3. Use of a face shield
4. Wearing of gloves and protective clothing

9-49. Exothermic coatings can be applied by which of these processes?
1. Powder oxygen-fuel spray
2. Wire oxygen-fuel spray

Learning Objective: Identify the procedures to be followed in preparing metal surfaces for spray metalizing. Textbook pages 14-4 through 14-7.

9-50. What are the major steps in preparing a surface to receive a metal spray coating?
1. Cleaning and surface roughening
2. Cleaning and undercoating
3. Cleaning, undercutting and surface roughening
4. Surface roughening, undercutting, and polishing

9-51. A shaft is to be restored by undercutting a worn section and building up with sprayed metal. Before it is roughened by abrasive blasting, which of the following operations must be done?
1. Washing with solvent
2. Undercutting
3. Cleaning by vapor degreasing
4. Both cleaning and undercutting

9-52. When heat cleaning is used on porous steel castings which have been contaminated with oily deposits, they should be heated to what temperature?
1. 650°F
2. 550°F
3. 300°F
4. 200°F

9-53. When preparing a shaft for restoration, you should finish the ends of the undercut section in what way(s)?
1. Weld them
2. Braze them
3. Chamfer or straight cut them
4. Fillet them

9-54. When machining an undercut on a shaft, how do you determine what length is to be undercut?
1. It is a multiple of the diameter of the shaft
2. It is no more than the length of the work area
3. It depends on the amount of wear anticipated
4. It depends on the length of the bearing or packing in which that section of the shaft will run

9-55. In preparing a surface for spray metalizing, you should undercut to a depth at least equal to the recommended minimum thickness for the particular coating plus the wear tolerance.

9-56. The abrasive blasting used for surface preparation should produce what desired degree of roughness?
1. 100-200 microinches
2. 200-300 microinches
3. 300-400 microinches
4. 400-500 microinches

9-57. When using suction equipment during abrasive blasting operations, you should place the nozzle how far from the work?
1. 1 to 2 inches
2. 2 to 4 inches
3. 3 to 6 inches
4. 6 to 12 inches

9-58. Having started abrasive blasting a surface, you pause and place a drop of cleaning solvent on the freshly blasted surface. What are you looking for?
1. A dark ring which indicates oil in the compressed air
2. An oily smudge that indicates the part has not been completely degreased
3. Traces of the blasting grit being embedded in the surface
4. All of the above

9-59. In abrasive blasting, what masking material will work best?
1. Aluminum
2. Paper tape
3. Electrical tape
4. Rubber

9-60. After you have completed preparation of a surface, you should begin the spraying operation within
1. 1 hour
2. 2 hours
3. 3 hours
4. 4 hours

Learning Objective: Describe the procedures for spray metalizing using the fuel-oxygen methods. Textbook pages 14-7 and 14-8.

9-61. Prior to applying metal spray, you should ensure the work is free of moisture by preheating it to
1. 60° to 80°F
2. 100° to 150°F
3. 200° to 225°F
4. 250° to 300°F

9-62. If you are using a gas flame to preheat work, you should avoid directing the flame directly onto that part of the work which is to be sprayed.

9-63. If the surface being sprayed is too cool, the sprayed metal may crack or not bond. If the surface is too hot, either the temper of the base metal will be lost or the deposit will be unsatisfactory. Within what temperature range should the base metal be while being sprayed?
1. 50° to 250°F
2. 60° to 350°F
3. 70° to 450°F
4. 200° to 500°F

9-64. If macroroughening (the cutting of fine grooves in the work with a lathe) has been used to prepare the surface, the stream of spray metal for the first few layers should be directed at an angle of
1. 45° to the work axis
2. 50° to the work axis
3. 55° to the work axis
4. 60° to the work axis

9-65. When the work is to be machined to its final dimension, what minimum amount of excess spray should be applied?
1. 0.003 inch
2. 0.004 inch
3. 0.005 inch
4. 0.008 inch

9-66. On completion of spraying the shaft, how should you cool the work?
1. Quench it with water
2. Quench it with oil
3. Quench it with brine
4. Allow it to set at room temperature

9-67. If severe conditions are anticipated during the service life of the surface, when are sealants applied to the metalized surface?
1. Between the spraying and machining operations
2. After the machining operation
3. At either or both of the above times
4. Before the last layer of spray is deposited

9-68. What material(s) should be used to mask against sprayed metal?
1. Common masking tape
2. A high temperature resisting masking tape
3. A special liquid masking compound
4. Both 2 and 3 above

Learning Objective: Identify special problems involved in machining and finishing thermal sprayed metal deposits. Textbook pages 14-8 through 14-12.

9-69. Which of the following equipment is NOT suitable for finishing sprayed metals?
1. Soft, coarse grinding wheels
2. Dry grinding wheels
3. Carbide tools
4. Clean coolants

9-70. Which of the following techniques is NOT suitable for finishing sprayed metal surfacea?
1. High work speed
2. Light cuts
3. High grinding wheel speed
4. Slow feed or traverse of the cut

9-71. What is achieved by applying sealer to a sprayed surface of a journal bearing before grinding it?
1. The metal will not clog the wheel
2. The metal will cut more cleanly
3. Abrasive is kept out of the pores of the sprayed metal
4. Both 2 and 3 above

Assignment 10

Metal Buildup (Continued)

Textbook Assignment: Pages 14-12 through 14-67

Learning Objective: Identify the equipment and materials used in contact electroplating. Textbook pages 14-12 through 14-16.

10-1. Since it provides corrosion and wear resistance and can be used to build up worn areas on machine parts, contact electroplating has replaced the older system of bath plating.

10-2. The most commonly used electroplating power packs have what output?
1. 60 to 100 amperes a.c.
2. 25 to 100 amperes a.c. or d.c.
3. 25 to 100 amperes d.c.
4. 60 to 100 amperes a.c. or d.c.

10-3. A contact plating tool consists of an insulated handle with a conductive core and an anode made of
1. an iridium-platinum alloy
2. graphite or high quality carbon
3. either a high quality graphite or an iridium-platinum alloy
4. conductive graphite with cooling fins

10-4. What three types of solutions are used for contact plating and related operations?
1. Cleaning, activating, and plating
2. Preparatory, plating, and stripping
3. Plating, stripping, and activating
4. Stripping, cleaning, and activating

10-5. In what way is contact plating solution applied to the work?
1. Dipped from a shallow dish
2. Brushed on the work
3. Poured on the work
4. The work is dipped in the solution

10-6. What is/are the function(s) of the covering on the contact plating tool?
1. To prolong tool life
2. To facilitate plating flat areas
3. To soak up and dispense the plating solution
4. To do all of the above

10-7. Which of the following is NOT an important operator function of the contact plating process?
1. Preparing the surface to be plated
2. Maintaining the power pack unit
3. Determining proper power settings for the job to be done
4. Using proper techniques to apply the plating

Learning Objective: Identify the hazards associated with plating solutions. Textbook pages 14-16 and 14-17.

10-8. Contact plating solutions are hazardous in which, if any, of the following ways?
1. Irritating to the eyes
2. Toxic and poisonous
3. Both 1 and 2 above
4. They are not toxic, but may stain the skin

10-9. Contact plating solutions and the vapors produced during the plating operations are irritating to the eyes and skin but are not dangerous to the respiratory system.

Learning Objective: Interpret and apply technical terms used in describing the plating process. Textbook pages 14-17 through 14-21.

In items 10-10 and 10-11, match the operation in column A with the condition in column B.

A. Operation	B. Condition
10-10. Activating	1. Forward current; the work is the cathode
10-11. Plating	2. Forward current; the anode is the cathode
	3. Reverse current; the work is the cathode
	4. Reverse current; the anode is the cathode

10-12. In the electrical circuit used for plating the ions of the plating solution flow away from the anode and toward the cathode.

10-13. Anodic corrosion protection is provided by a deposit of reactive metal on noble metal.

10-14. Sacrificial corrosion protection is provided by the same means as cathodic corrosion protection.

10-15. Carburizing, hydrogen embrittlement, and nitriding are all similar processes used to surface-harden light alloys.

In items 10-16 and 10-17, match the electrical term in column A with the characteristic of water flowing in a pipe in column B.

A. Electrical Term	B. Characteristic of water flowing in a pipe
10-16. Ampere	1. Velocity of the flow
10-17. Volt	2. Volume flowing
	3. Pressure of the water
	4. Internal resistance of the pipe

Learning Objective: Cite typical applications of contact plating. Textbook pages 14-21 and 14-22.

10-18. What application of contact plating has been the most successful and can be done with the least restriction?
1. Class I
2. Class II
3. Class III
4. Class IV

10-19. Which of the following products of a plating operation would be Class V?
1. A rubbing-contact part for galley equipment
2. A rubbing-contact part for a main propulsion turbine
3. Chrome-plated shell casings to decorate the quarterdeck
4. A component of nuclear power propulsion machinery

10-20. Restoring surfaces by contact plating must be approved by NAVSEA in which of the following cases?
1. Repairing steam cuts in a steam turbine casing flange
2. Repairing a steam turbine rotor journal bearing
3. Improving sealing surfaces on wave guides
4. All of the above

Learning Objective: Identify applications which are not suitable for contact plating or which require special techniques. Textbook pages 14-22 and 14-23.

10-21. Plating is an effective way to repair small cracks (not in excess of 1/2 inch in length).

10-22. How can additional layers of chromium be plated over an existing bath chromium deposit?
1. By completely removing the bath chromium deposit
2. By first acid cleaning the surface of the bath
3. By first plating the bath chromium with nickel
4. By using normal surface preparation of the bath chromium

10-23. Why is contact plating with chromium to build up worn parts NOT recommended?
1. Other metals will do the job better
2. Thick chromium layers do not work well
3. Contact plated chromium is not as hard as bath plated chromium
4. For all of the above reasons

10-24. To restore a wear-resistant surface where an extensive buildup is needed, you should deposit a layer of hard metal such as cobalt on a base of
1. nickel
2. cobalt-tungsten
3. copper
4. chromium

Learning Objective: Identify documentation associated with contact plating operations. Textbook pages 14-23 and 14-24.

10-25. If a plating solution does not perform according to its specifications, you should notify the Naval Ship's Engineering Center, and they will notify the manufacturer.

10-26. As a quality control measure, liquid penetrant testing is used to check
1. surfaces before plating
2. flange surfaces
3. all hard surfaces
4. sliding contact bearing surfaces

10-27. Which of the following information about a plating job is NOT part of the engineering and job planning function?
1. Method of surface finishing
2. Name of the ship that ordered the job
3. Types of metals in the base and used for plating
4. Finished thickness of the deposit

Learning Objective: Describe the principal inspections and tests used to check the quality of contact plating. Textbook page 14-25.

10-28. A burned portion of plating has what appearance?
1. Blistered
2. Pitted
3. Coarse grained
4. Dark blue and pitted

10-29. Inspection of the quality of a deposit intended for rubbing contact service includes which of the following tests?
1. Liquid penetrant
2. Visual
3. Adhesion
4. All of the above

10-30. When plating is inspected using Group I liquid penetrant, the indication shall NOT exceed what amount?
1. 1/64 in.
2. 1/32 in.
3. 1/16 in.
4. 1/8 in.

Learning Objective: Identify the components and describe the general operation of the contact plating power pack. Textbook pages 14-25 through 14-27.

10-31. What is the principal function of the ammeter on the power pack?
1. To indicate plating rate
2. To avoid overloading the pack
3. To measure the thickness of deposit
4. To set the proper plating rate

10-32. To determine how much of a total plating operation has been completed, you should observe the
1. ammeter
2. voltmeter
3. volts control knob
4. ampere-hour meter

10-33. The rate at which plating will occur is controlled by the
1. reset button
2. volts control knob
3. ampere-hour meter
4. circuit breakers

10-34. Which of the following guidelines applies to the connecting of the plating tool leads?
1. Always connected to a terminal of the opposite color
2. Large size leads are always connected to the black terminal
3. Usually connected to the black terminal
4. Color and size of leads must be compatible with the terminal to which attached

10-35. You are going to plate a part with nickel solution 2080 and desire a contact area of 16 square inches for the tool. What rating (from Table 14-3) power pack should you select?
1. 30-25
2. 60-35
3. 100-40
4. 140-40

61

Learning Objective: Select and prepare plating tools. Textbook pages 14-27 through 14-33.

10-36. Tools used in the cleaning, deoxidizing, and etching steps do not have to meet the requirements of plating tools. However, they should meet what minimum contact area requirements?
1. Contact 10% of the area
2. Contact 30% of the area
3. Contact 10% of the length of the area
4. Contact 30% of the length of the area

10-37. The importance of selecting the proper plating tool is greatest when which of the following factors is present?
1. Thin plating on a small area
2. Thick plating on a small area
3. Thick plating on a large area
4. Thin plating on a large area

10-38. You are using solution 2080 with a 100-40 power pack. What is the optimum (maximum) contact area? (Use Table 14-3.)
1. 5 sq in.
2. 10 sq in.
3. 17 sq in.
4. 25 sq in.

10-39. You must design a special plating tool using Formula 3 on page 14-66 of the text. Using solution 2080, you find in Table 14-7 that the average current density (ACD) is 6. Your power pack is rated at 100 amps. What is the optimum contact area?
1. 16 2/3 sq in.
2. 18 1/2 sq in.
3. 25 sq in.
4. 60 sq in.

10-40. Which of the following is NOT an advantage of a solution-fed tool?
1. Cooler anodes
2. Faster plating rate
3. Less solution required
4. Ensures more fresh solution available on the plating surface

- In answering item 10-41, use Formula 1 on page 14-65.

10-41. To plate an area of 40 sq in. to a depth of 0.003 in. with a solution of Nickel code 2080, how many ampere-hours are required?
1. 12.6 amp-hr
2. 25.2 amp-hr
3. 29 amp-hr
4. 36.2 amp-hr

- Use the following information in answering question 10-42.

You are using nickel 2080 to plate an area of 40 sq in. From Table 14-7 you determine that ACD is 6. Your power pack is rated at 100 amperes. The depth of plating is to be 0.003 in. and 25.2 amp-hr plating current is required. Your plating tool has a contact area of 17 sq in.

10-42. About how much time can be saved by using a solution-fed tool?
1. 0.105 hr
2. 0.240 hr
3. 0.247 hr
4. 0.252 hr

10-43. If a 150-40 power pack is used to plate the bearing area of a shaft that has a total surface area of 40 sq in., what is the maximum practical contact area?
1. 40 sq in.
2. 20 sq in.
3. 17 sq in.
4. It depends on the length of the contact area

10-44. You are to plate a cylindrical outside diameter surface that has a length of 4 inches and a circumference of 10 inches. You will use plating solution and equipment requiring an optimum contact area of 17 sq in. What should be the dimensions of the tool's (A) length and (B) internal arc?
1. (A) 4 in. (B) 10 in.
2. (A) 4 in. (B) 5 in.
3. (A) 4 in. (B) 4 1/4 in.
4. (A) 3 1/4 in. (B) 5 in.

Learning Objective: Identify procedures for ensuring the use of clean anodes for plating operations, and describe procedures for covering anodes. Textbook pages 14-34 through 14-40.

10-45. What is the best way to be sure an anode is NOT contaminated with one solution before being used with another?
1. Use one tool for preparatory operations and another for plating
2. Scrape and wash all anodes after use
3. Scrape and wash plating anodes before use
4. Identify specific anodes for use with specific solutions

10-46. What is the relationship between initial covers and final covers?
1. The final covers protect the initial covers
2. Final covers are used only for finishing work
3. Initial covers can be used by themselves, but final covers must be used with initial covers
4. The final cover is the one that distributes the solution over the contact area

10-47. What is the most commonly used final cover?
1. Dacron batting
2. Cotton batting
3. Dacron tubegauze
4. Cotton tubegauze

10-48. What is the most commonly used initial cover?
1. Dacron batting
2. Cotton batting
3. Dacron tubegauze
4. Cotton tubegauze

10-49. What covering material is subject to having the plating material deposited inside the cover?
1. Carbon felt
2. Orlon
3. Gray Scotchbrite
4. Dacron felt

10-50. What should you do with a covered plating tool that has been slightly used today and could be used to continue a similar plating operation tomorrow?
1. It must be discarded and a new tool made up the next day
2. The cover must be discarded, but the tool can be cleaned and used again
3. Clean the tool and recover it today
4. Squeeze the cover dry, wrap it in plastic, and use it the next day

10-51. When wrapping the anodes for plating inside or outside diameters, about what total thickness of cotton batting is desired?
1. 1/16 in.
2. 3/32 in.
3. 2/16 in.
4. 3/16 in.

Learning Objective: Identify general procedures for storing plating solutions. Textbook page 14-41.

10-52. Contact plating solutions should be stored at room temperature and away from light.

10-53. If solid crystals precipitate out of a plating solution and form on the bottom of the container, the solution always must be discarded.

10-54. What information is kept in a log to indicate how heavily the solution has been used?
1. The specific gravity of the solution
2. The number of times the solution has been used
3. The amount of electrical current that has passed through the solution
4. The percentage of dilution of the solution

Learning Objective: Recognize factors and techniques involved in masking work to be contact plated. Textbook page 14-42.

10-55. Which of the following materials should NOT be used for masking in contact plating operations?
1. Common masking tape
2. Copper tape
3. Vinyl tape
4. Polyenter tape

10-56. A small margin of aluminum tape exposed at the edge of the plating area will provide what beneficial factor?
1. It will make a good surface for another layer of vinyl tape
2. It will confine the plating to the desired area since aluminum does not plate
3. It will give an early and harmless indication of burning conditions
4. It will add some aluminum to the plating metal

Learning Objective: Plan a typical plating job. Textbook pages 14-43 through 14-53.

10-57. In Example #1 in the text, considering the purpose of the deposit and the information in Tables 14-8 and 14-9, why was Cobalt 2043 chosen instead of Nickel 2080 as the plating solution?
1. Greater maximum buildup is possible with cobalt
2. The cobalt solution is cheaper and easier to use
3. The cobalt solution provides a better color match with steel
4. For the reasons given in both 2 and 3 above

10-58. If the plating thickness on the bore in Example #1 in the text were 0.002 inches, it would reduce the 3.5015 in. bore to
1. 3.4965 in.
2. 3.4975 in.
3. 3.4985 in.
4. 3.4995 in.

10-59. If the plating solution were Nickel 2080, what would be the plating amperage in step #7?
1. 38.5
2. 77.0
3. [illegible]
4. [illegible]

10-60. Which of the formulas on page 14-66 is used to determine the amount of plating solution required?
1. Formula 4
2. Formula 5
3. Formula [illegible]
4. Formula 7

10-61. [illegible] cotton tubegauze selected as [illegible] final cover in Example #1?
1. Greater durability
2. Less expensive
3. High absorbancy
4. Easier to wrap

10-62. In Example #2, what is the purpose of using the Nickel 2085 solution?
1. Faster buildup
2. Easier to machine
3. Color match with the base metal
4. Lower cost than Copper 2050

Learning Objective: Cite practices which will develop and improve your proficiency in contact plating. Textbook pages 14-53 and 14-54.

10-63. You are familiarizing yourself with a new solution by using very high and very low voltages while plating a scrap part. Why?
1. To see the effect on the controls
2. To see the appearances of bad deposits
3. To test the quality of the solution
4. To determine the optimum contact area

10-64. What is the principal reason you should ensure your working position is comfortable before starting a plating job?
1. To maintain maximum efficiency and concentration for as long as the job takes
2. To make the work more enjoyable
3. To avoid boredom
4. To decrease the overall preparation and plating time

Learning Objective: Review procedures for preparing surfaces for plating. Textbook pages 14-54 through 14-[illegible].

10-65. The operations performed in the preparation of a part for plating will depend on
1. the type of plating solution to be used
2. the available preparation solutions
3. the requirements for producing a perfectly clean surface on the base metal
4. the ultimate use of the plated part

10-66. What factor must be considered in performing a cleaning and deoxidizing operation on ultra-high strength steel?
1. The hardness of the oxides that form on these steels
2. Avoiding hydrogen embrittlement of the surfaces being cleaned
3. These steels can be cleaned only by abrasion
4. Deoxidizing of these steels is not necessary

10-67. What is the indication of a successful cleaning and deoxidizing operation?

1. A uniform, dull, grainy appearance
2. A light carbon film on the surface
3. Water flows off the surface without forming droplets
4. The tool surface remains cool

10-68. What are the results of etching?

1. A uniform, dull, grainy appearance
2. At least 0.006 in. of material has been removed
3. Water "breaks" on the surface
4. All of the above

10-69. Immediately before plating, stainless steels must undergo what treatment?

1. Desmutting
2. Etching
3. Drying
4. Activating

10-70. You are preparing stainless steel for plating with Copper 2055. Immediately after activating the part you should

1. etch it
2. preplate it
3. [illegible]
4. [illegible]

Learning Objective: Test plating deposits. Textbook pages 14-63 through 14-67.

10-71. In using the chisel, knife, or saw test, you should cut in what direction?

1. Straight down into the plating
2. Across the plating
3. From the base metal to the plating
4. From the plating to the base metal

10-72. Which of the following problems will cause separation between preplate and plate layers?

1. The surface was not properly etched
2. The base is an improperly sprayed deposit
3. The base metal was not identified correctly
4. An incorrect plating solution was used

10-73. When a deposit is not as thick as it should be, underestimating the area to be plated or underestimating the amp-hr factor are possible causes.

10-74. Properly deposited metals can be successfully machined with which of the following devices?

1. Carbide and tool steel cutting tools
2. Dry grinding and high cutting speeds
3. Carbide tools and grinding wheels
4. Heavy cuts and soft wheels

Assignment 11

The Repair Department and Repair Work

Textbook assignment: Pages 15-1 through 15-45

Learning Objective: Describe the organization of a typical tender's repair department. Textbook pages 15-1 through 15-8.

11-1. Which type of repair ship provides general and specific repairs to all types of ships?
1. AR
2. ARG
3. AS
4. AD

In items 11-2 through 11-5, match the designation of the division in column A with the associated repair department shop in column B.

	A. Division	B. Shop
11-2.	R-1	IC Repair
11-3.	R-2	Machine
11-4.	R-3	Calibration
11-5.	R-4	Shipfitter

11-6. The repair officer aboard a destroyer tender performs all EXCEPT which of the following functions?
1. Issuing and enforcing repair department orders
2. Scheduling and assigning ships for availabilities
3. Accepting or rejecting work requests
4. Initiating requests for additional funds

11-7. Aboard a typical repair ship, who is responsible for the detailed training of repair department personnel?
1. Senior division PO
2. Assistant repair officer
3. Repair officer
4. Educational services officer

11-8. In the repair department of a repair ship, which officer(s), in addition to the assistant repair officer, may be assigned to help the repair officer carry out his functions?
1. Production engineering assistant
2. Radiological control officer
3. Department training officer
4. Any or all of the above

11-9. In a ship's repair department, who may be assigned as the R-2 division officer?
1. Commissioned officer
2. Warrant officer
3. Chief petty officer
4. Any one of the above

11-10. Shop layout and arrangement may vary from one repair ship to another.

Learning Objective: Identify principal dimensions of a spur gear. Textbook pages 15-9 through 15-13.

11-11. The distance from one side of the root circle to the opposite side passing through the center of the gear is called the
1. whole depth
2. root diameter
3. circular pitch
4. working depth

11-12. What is the abbreviation symbol for chordal thickness?
1. CT
2. c_t
3. t_c
4. TC

11-13. In spur gear terminology, what is the dedendum (DED)?
1. The circle formed by the top of gear teeth
2. The whole depth minus the clearance
3. The circle formed by the bottoms of the gear teeth
4. The depth of the tooth inside the pitch circle

11-14. Backlash (B) is the difference between the tooth thickness and the tooth space of engaged gear teeth at the pitch circle.

11-15. What is the most important calculation of a spur gear?
1. Pitch circle
2. Outside diameter
3. Diametral pitch
4. Chordal thickness

● In answering items 11-16 through 11-19, refer to figure 11A which follows.

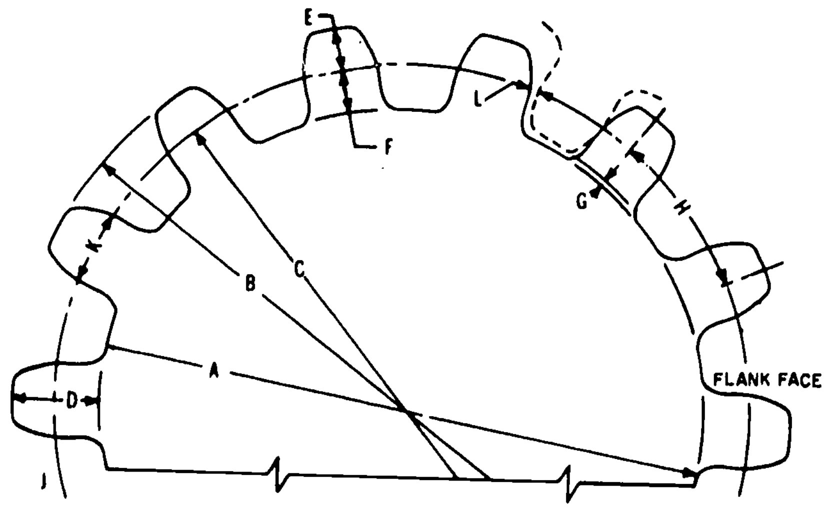


Figure 11A

11-16. The pitch diameter is represented by line
1. A
2. B
3. C
4. D

11-17. The circular pitch of a gear is indicated by
1. E
2. H
3. J
4. K

11-18. The whole depth of a gear tooth is equal to
1. E + G
2. E + F
3. B - A
4. E + F + G

11-19. If B is 4 inches and D is 5/8 inch, what is the root diameter?
1. 2 3/4 in.
2. 3 in.
3. 3 1/4 in.
4. 3 1/2 in.

11-20. Computation involving gear dimensions is simpler if measurements are based on the diameter of the pitch circle instead of the circumference of the pitch circle.

● Items 11-21 through 11-26, refer to a 32-tooth gear whose outside diameter is 4.25 inches.

11-21. What is the whole depth of tooth of this gear?
1. 0.2696 in.
2. 0.4044 in.
3. 0.4348 in.
4. 0.5285 in.

11-22. What number cutter should be used to cut this gear?
1. No. 3
2. No. 4
3. No. 7
4. No. 8

11-23. To check the accuracy of gear tooth thicknesses, what should you use?
1. A micrometer
2. A vernier caliper
3. An indexing device
4. Formulas based on the diametral pitch system

11-24. What is the chordal addendum of a tooth on this gear?
1. 0.0386 in.
2. 0.1274 in.
3. 0.1964 in.
4. 0.2696 in.

11-25. The chordal tooth thickness of a tooth on this gear is equal to 4 times the sine of
1. 1°51'
2. 2°49'
3. 3°45'
4. 4°

11-26. What is the diameter of the stock you should use for making a gear of this size?
1. 4 1/8 in.
2. 4 1/4 in.
3. 4 3/8 in.
4. 4 1/2 in.

Learning Objective: Perform typical shop calculations relating to fabricating parts. Textbook pages 15-14 through 15-16.

11-27. To make a tapered shaft 20 inches long whose maximum diameter is 2 1/2 inches and minimum diameter is 2 inches, what size of round stock should you use?
1. Length 19 15/16 in., diameter 2 3/8 in.
2. Length 19 7/8 in., diameter 2 5/8 in.
3. Length 20 1/16 in., diameter 2 5/8 in.
4. Length 20 1/8 in., diameter 2 9/16 in.

11-28. When machining steps 11 and 12 as noted in figure 15-10 of the textbook are performed, the groove should be cut last.

11-29. What device is used to measure a shaft runout?
1. Dial indicator
2. Micrometer
3. Vernier caliper
4. Steel rule

11-30. In bending a shaft to eliminate a runout 0.3 inch, you should release ram pressure when the dial indicator indicates a reverse bend of
1. 0.003 in.
2. 0.03 in.
3. 0.30 in.
4. 0.303 in.

11-31. You are to stub a damaged shaft 2 inches in diameter. After you cut off the damaged part, you should drill a hole at the end of the undamaged shaft of what diameter?
1. 5/8 in.
2. 1 1/4 in.
3. 2 in.
4. 2 1/4 in.

11-32. The stub you make for a shaft 2 inches in diameter should have a diameter of
1. 3/8 in.
2. 1 1/4 in.
3. 2 in.
4. 2 1/4 in.

11-33. When stubbing a damaged shaft, you should chamfer the shoulder of the machined end of the stub as much as the end of the undamaged part of the shaft has been chamfered.

Learning Objective: Identify key points of common valve repair operations. Textbook pages 15-16 through 15-28.

11-34. The bodies of high-temperature, high-pressure valves are usually made of
1. Monel
2. alloy steel
3. Stellite
4. bronze

11-35. Which valve seat material has the greatest saltwater corrosion resistance?
1. Alloy steel
2. Monel
3. Bronze
4. Brass

11-36. In which situation will a globe valve likely be removed from the line for repair of the valve seat?
1. When the valve seat is surfaced with Stellite
2. When the valve seat has been deeply cut
3. When the valve seat has been threaded into the valve line
4. When the size of the valve is between 1/4 in. and 12 in.

11-37. Which operation includes coating a valve seat with prussian blue?
1. Grinding
2. Spotting-in
3. Lapping
4. Facing

11-38. In what valve-seating operation is abrasive placed between the disk and seat of a globe valve?
1. Grinding
2. Lapping
3. Refacing
4. Machining

11-39. On a globe valve seat, how should small pits be removed?
1. By lapping
2. By grinding
3. By refacing
4. By spotting-in

11-40. Which practice is likely to produce a poor lapping job on a valve?
1. Rotating the working position of the lap on the seat
2. Using light pressure on the lap
3. Using side pressure on the lap
4. Frequently renewing the lapping compound

11-41. Which of the following materials is best suited for manufacturing a lapping tool?
1. Monel
2. Alloy steel
3. Brass
4. Cast Iron

11-42. On a globe valve, what repair operations require the use of an abrasive compound?
1. Grinding and refacing
2. Grinding and lapping
3. Lapping and spotting-in
4. Spotting-in and grinding

11-43. Most ball valves are of the quick-acting type, requiring only a 90° turn to completely open or close the valve.

11-44. Gate valves are not suitable for throttling purposes because the flow is difficult to control.

11-45. To lap slight defects in gate valve seat rings, you should use the wedge gate coated with a medium grade lapping compound.

11-46. How are gate valves classified?
1. Rising stem valves
2. Nonrising stem valves
3. Both 1 and 2 above
4. Quick-acting stem valves

11-47. To repair gate valves for defects such as light pitting or scoring, what is the correct procedure?
1. Machine the gate on a lathe
2. Replace the gate
3. Lap the gate
4. Grind the gate

11-48. In repairing a Leslie constant-pressure governor for a main feed pump, you should keep lapping of the damaged areas to a minimum.

11-49. The letter A stamped on a valve bolt or nut indicates a special-purpose characteristic of its
1. thread design
2. thread gage
3. metal
4. surface

11-50. Before tightening bonnet nuts on a high-pressure steam valve, you should first back the disk away from the seat.

11-51. What instrument is used to determine the nut tension in a high-pressure steam valve assembly?
1. Micrometer
2. Ratchet wrench
3. Spring scale
4. Compressible washer

11-52. To test overhauled steam valves in the shop, you should apply
1. high-pressure steam
2. high-pressure water
3. compressed air
4. prussian blue solution

Learning Objective: Identify key points applicable to repairing pumps. Textbook pages 15-29 through 15-31.

11-53. Which of the following parts of a centrifugal pump generally has a renewable surface?
1. Impeller passage
2. Balance passage
3. Rotor shaft
4. Discharge chamber

11-54. Pressed-on shaft sleeves are removed from a pump with the aid of a
1. gear puller
2. sleeve puller
3. lathe
4. torch

11-55. What tool is used to remove worn impeller wearing rings from a centrifugal pump?
1. Hydraulic press
2. Lathe
3. Drill press
4. Hex key

11-56. When replacement wearing rings for a centrifugal pump have inaccurate concentricity, how can you usually correct the diameters?
1. By grinding
2. By lapping
3. By machining
4. By polishing with emery paper

11-57. When chucking a casing ring in a 4-jaw chuck, you should exercise caution to prevent
1. cutting the ring
2. distorting the ring
3. enlarging the ring
4. shrinking the ring

Learning Objective: Cite principal considerations in maintaining machine shop equipment. Textbook pages 15-31 and 15-32.

11-58. As designed in the 3-M system, implementing systematic planned maintenance should improve the level of operational readiness of maintainable equipment and preserve this level of readiness at all times.

11-59. In the tropics especially, rust prevention should be a part of the lubricating and cleanup routine. How often should the bare metal surfaces of machines be gone over with a rust-preventive compound?
1. Daily
2. Weekly
3. Biweekly
4. Monthly

11-60. In securing a machine shop for sea, which of the following procedures should you follow?
1. Pack small gear in locked drawers and cabinets
2. Lower, block, and secure all top-heavy machinery
3. Secure suspended equipment such as chain falls, trolleys, and overhead cranes
4. Do all of the above

Learning Objective: Describe how to remove broken bolts and studs. Textbook pages 15-32 through 15-36.

11-61. In the removal of broken studs with a screw extractor, what is the first step?
1. Center punch the center of the bolt
2. File the broken portion of the bolt
3. Drill the hole to accept the screw extractor
4. Hit the bolt with a hammer to loosen the thread

11-62. When drilling a hole in a stud that has been broken below the surface of the piece it was holding, what should you use to center the drill?
1. Drill index
2. Tap drill
3. Drill guide
4. Drill extractor

Learning Objective: Describe how to make a piston ring. Textbook pages 15-36 through 15-45.

11-63. In manufacturing a piston ring, how should you machine the inside bore of the billet?
1. Machine 0.010 inch oversize
2. Machine 0.010 inch undersize
3. Machine to finished size
4. Rough machine

11-64. In manufacturing a piston ring, what should you use for finish machining?
1. A 4-jaw chuck
2. A mandrel
3. A faceplate
4. A grinder

U.S. GOVERNMENT PRINTING OFFICE: 1981-740-086/'30

COURSE DISENROLLMENT

All study materials must be returned. On disenrolling, fill out only the upper part of this page and attach it to the inside front cover of the textbook for this course. Mail your study materials to the Naval Education and Training Program Development Center.

PRINT CLEARLY

NAVEDTRA NUMBER	COURSE TITLE
10530-E	NRCC Machinery Repairman 3&2

Name Last First Middle

Rank/Rate Des[illegible]tor Social Security Number

COURSE COMPLETION

Letters of satisfactory completion are issued only to personnel whose courses are administered by the Naval Education and Training Program Development Center. On completing the course, fill out the lower part of this page and enclose it with your last set of answer sheets. Be sure mailing addresses are complete. Mail to the Naval Education and Training Program Development Center.

PRINT CLEARLY

NAVEDTRA NUMBER	COURSE TITLE
10530-E	NRCC Machinery Repairman 3&2

Name

ZIP CODE

MY SERVICE RECORD IS HELD BY:

Activity

Address ZIP CODE

Signature of enrollee

A FINAL QUESTION: What did you think of this course? Of the text material used with the course? Comments and recommendations received from enrollees have been a major source of course improvement. You and your command are urged to submit your constructive criticisms and your recommendations. This tear-out form letter is provided for your convenience. Typewrite if possible, but legible handwriting is acceptable.

Date ____________________

From: ________________________________
(RANK, RATE, CIVILIAN)

____

________________________________ ZIP CODE __________

To: Naval Education and Training Program Development Center
Pensacola, Florida 32509

Subj: RTM/NRCC Machinery Repairman 3&2, NAVEDTRA 10530-E

1. The following comments are hereby submitted:

(Fold along dotted line and staple or tape)

(Fold along dotted line and staple or tape)

DEPARTMENT OF THE NAVY
NAVAL EDUCATION AND TRAINING PROGRAM
DEVELOPMENT CENTER (PD 4)
PENSACOLA, FLORIDA 32509

OFFICIAL BUSINESS
PENALTY FOR PRIVATE USE, $300

POSTAGE AND FEES PAID
NAVY DEPARTMENT
DoD-316

U.S.MAIL

NAVAL EDUCATION AND TRAINING PROGRAM DEVELOPMENT CENTER
BUILDING 2435 (PD 4)
PENSACOLA, FLORIDA 32509

6...

PRINT OR TYPE

NAME ______________________ (Last) (First) (Middle) ADDRESS ______________________ (Street/Ship/Unit/Division, etc.)

(City or FPO) (State) (Zip)

RANK/RATE ______ SOC. SEC. NO. ______________ DESIGNATOR ______ ASSIGNMENT NO. ______

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ______________ DATE MAILED ______

SCORE

	1 T	2 F	3	4			1 T	2 F	3	4			1 T	2 F	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

PRINT OR TYPE

NAME ______________________________ (Last / First / Middle) ADDRESS ______________________________ (Street/Ship/Unit/Division, etc.)

______________________________ (City or FPO / State / Zip)

RANK/RATE ________ SOC. SEC. NO. ______________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ______________ DATE MAILED ________

SCORE

	1 T	2 F	3	4			1 T	2 F	3	4			1 T	2 F	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinerv Repairman 2
NAVEDTRA 10530-E

NAME ______________________________ ADDRESS ______________________________

Last First Middle — Street/Ship/Unit/Division, etc.

____

City or FPO State Zip

RANK/RATE ______ SOC. SEC. NO. ______________ DESIGNATOR ______ ASSIGNMENT NO. ______

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ______________ DATE MAILED ______

SCORE

No.	1 T	2 F	3	4	
1	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____
26	☐	☐	☐	☐	_____
27	☐	☐	☐	☐	_____
28	☐	☐	☐	☐	_____
29	☐	☐	☐	☐	_____
30	☐	☐	☐	☐	_____
31	☐	☐	☐	☐	_____
32	☐	☐	☐	☐	_____
33	☐	☐	☐	☐	_____
34	☐	☐	☐	☐	_____
35	☐	☐	☐	☐	_____
36	☐	☐	☐	☐	_____
37	☐	☐	☐	☐	_____
38	☐	☐	☐	☐	_____
39	☐	☐	☐	☐	_____
40	☐	☐	☐	☐	_____
41	☐	☐	☐	☐	_____
42	☐	☐	☐	☐	_____
43	☐	☐	☐	☐	_____
44	☐	☐	☐	☐	_____
45	☐	☐	☐	☐	_____
46	☐	☐	☐	☐	_____
47	☐	☐	☐	☐	_____
48	☐	☐	☐	☐	_____
49	☐	☐	☐	☐	_____
50	☐	☐	☐	☐	_____
51	☐	☐	☐	☐	_____
52	☐	☐	☐	☐	_____
53	☐	☐	☐	☐	_____
54	☐	☐	☐	☐	_____
55	☐	☐	☐	☐	_____
56	☐	☐	☐	☐	_____
57	☐	☐	☐	☐	_____
58	☐	☐	☐	☐	_____
59	☐	☐	☐	☐	_____
60	☐	☐	☐	☐	_____
61	☐	☐	☐	☐	_____
62	☐	☐	☐	☐	_____
63	☐	☐	☐	☐	_____
64	☐	☐	☐	☐	_____
65	☐	☐	☐	☐	_____
66	☐	☐	☐	☐	_____
67	☐	☐	☐	☐	_____
68	☐	☐	☐	☐	_____
69	☐	☐	☐	☐	_____
70	☐	☐	☐	☐	_____
71	☐	☐	☐	☐	_____
72	☐	☐	☐	☐	_____
73	☐	☐	☐	☐	_____
74	☐	☐	☐	☐	_____
75	☐	☐	☐	☐	_____

PRINT OR TYPE

NAME ______________________________ ADDRESS ______________________________
Last First Middle Street/Ship/Unit/Division, etc.

City or FPO State Zip

RANK/RATE ________ SOC. SEC. NO. ____________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ________________ DATE MAILED ________

SCORE

	1 T	2 F	3	4	
1	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____
26	☐	☐	☐	☐	_____
27	☐	☐	☐	☐	_____
28	☐	☐	☐	☐	_____
29	☐	☐	☐	☐	_____
30	☐	☐	☐	☐	_____
31	☐	☐	☐	☐	_____
32	☐	☐	☐	☐	_____
33	☐	☐	☐	☐	_____
34	☐	☐	☐	☐	_____
35	☐	☐	☐	☐	_____
36	☐	☐	☐	☐	_____
37	☐	☐	☐	☐	_____
38	☐	☐	☐	☐	_____
39	☐	☐	☐	☐	_____
40	☐	☐	☐	☐	_____
41	☐	☐	☐	☐	_____
42	☐	☐	☐	☐	_____
43	☐	☐	☐	☐	_____
44	☐	☐	☐	☐	_____
45	☐	☐	☐	☐	_____
46	☐	☐	☐	☐	_____
47	☐	☐	☐	☐	_____
48	☐	☐	☐	☐	_____
49	☐	☐	☐	☐	_____
50	☐	☐	☐	☐	_____
51	☐	☐	☐	☐	_____
52	☐	☐	☐	☐	_____
53	☐	☐	☐	☐	_____
54	☐	☐	☐	☐	_____
55	☐	☐	☐	☐	_____
56	☐	☐	☐	☐	_____
57	☐	☐	☐	☐	_____
58	☐	☐	☐	☐	_____
59	☐	☐	☐	☐	_____
60	☐	☐	☐	☐	_____
61	☐	☐	☐	☐	_____
62	☐	☐	☐	☐	_____
63	☐	☐	☐	☐	_____
64	☐	☐	☐	☐	_____
65	☐	☐	☐	☐	_____
66	☐	☐	☐	☐	_____
67	☐	☐	☐	☐	_____
68	☐	☐	☐	☐	_____
69	☐	☐	☐	☐	_____
70	☐	☐	☐	☐	_____
71	☐	☐	☐	☐	_____
72	☐	☐	☐	☐	_____
73	☐	☐	☐	☐	_____
74	☐	☐	☐	☐	_____
75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________________ (Last) ______ (First) ______ (Middle) ADDRESS ______________________ (Street/Ship/Unit/Division, etc.) ______ (City or FPO) ______ (State) ______ (Zip)

RANK/RATE ______ SOC. SEC. NO. ______________ DESIGNATOR ______ ASSIGNMENT NO. ______

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ______________ DATE MAILED ______

SCORE

	1 T	2 F	3	4			1 T	2 F	3	4			1 T	2 F	3	4	
1	☐	☐	☐	☐	_ _ _ _ _	26	☐	☐	☐	☐	_ _ _ _ _	51	☐	☐	☐	☐	_ _ _ _ _
2	☐	☐	☐	☐	_ _ _ _ _	27	☐	☐	☐	☐	_ _ _ _ _	52	☐	☐	☐	☐	_ _ _ _ _
3	☐	☐	☐	☐	_ _ _ _ _	28	☐	☐	☐	☐	_ _ _ _ _	53	☐	☐	☐	☐	_ _ _ _ _
4	☐	☐	☐	☐	_ _ _ _ _	29	☐	☐	☐	☐	_ _ _ _ _	54	☐	☐	☐	☐	_ _ _ _ _
5	☐	☐	☐	☐	_ _ _ _ _	30	☐	☐	☐	☐	_ _ _ _ _	55	☐	☐	☐	☐	_ _ _ _ _
6	☐	☐	☐	☐	_ _ _ _ _	31	☐	☐	☐	☐	_ _ _ _ _	56	☐	☐	☐	☐	_ _ _ _ _
7	☐	☐	☐	☐	_ _ _ _ _	32	☐	☐	☐	☐	_ _ _ _ _	57	☐	☐	☐	☐	_ _ _ _ _
8	☐	☐	☐	☐	_ _ _ _ _	33	☐	☐	☐	☐	_ _ _ _ _	58	☐	☐	☐	☐	_ _ _ _ _
9	☐	☐	☐	☐	_ _ _ _ _	34	☐	☐	☐	☐	_ _ _ _ _	59	☐	☐	☐	☐	_ _ _ _ _
10	☐	☐	☐	☐	_ _ _ _ _	35	☐	☐	☐	☐	_ _ _ _ _	60	☐	☐	☐	☐	_ _ _ _ _
11	☐	☐	☐	☐	_ _ _ _ _	36	☐	☐	☐	☐	_ _ _ _ _	61	☐	☐	☐	☐	_ _ _ _ _
12	☐	☐	☐	☐	_ _ _ _ _	37	☐	☐	☐	☐	_ _ _ _ _	62	☐	☐	☐	☐	_ _ _ _ _
13	☐	☐	☐	☐	_ _ _ _ _	38	☐	☐	☐	☐	_ _ _ _ _	63	☐	☐	☐	☐	_ _ _ _ _
14	☐	☐	☐	☐	_ _ _ _ _	39	☐	☐	☐	☐	_ _ _ _ _	64	☐	☐	☐	☐	_ _ _ _ _
15	☐	☐	☐	☐	_ _ _ _ _	40	☐	☐	☐	☐	_ _ _ _ _	65	☐	☐	☐	☐	_ _ _ _ _
16	☐	☐	☐	☐	_ _ _ _ _	41	☐	☐	☐	☐	_ _ _ _ _	66	☐	☐	☐	☐	_ _ _ _ _
17	☐	☐	☐	☐	_ _ _ _ _	42	☐	☐	☐	☐	_ _ _ _ _	67	☐	☐	☐	☐	_ _ _ _ _
18	☐	☐	☐	☐	_ _ _ _ _	43	☐	☐	☐	☐	_ _ _ _ _	68	☐	☐	☐	☐	_ _ _ _ _
19	☐	☐	☐	☐	_ _ _ _ _	44	☐	☐	☐	☐	_ _ _ _ _	69	☐	☐	☐	☐	_ _ _ _ _
20	☐	☐	☐	☐	_ _ _ _ _	45	☐	☐	☐	☐	_ _ _ _ _	70	☐	☐	☐	☐	_ _ _ _ _
21	☐	☐	☐	☐	_ _ _ _ _	46	☐	☐	☐	☐	_ _ _ _ _	71	☐	☐	☐	☐	_ _ _ _ _
22	☐	☐	☐	☐	_ _ _ _ _	47	☐	☐	☐	☐	_ _ _ _ _	72	☐	☐	☐	☐	_ _ _ _ _
23	☐	☐	☐	☐	_ _ _ _ _	48	☐	☐	☐	☐	_ _ _ _ _	73	☐	☐	☐	☐	_ _ _ _ _
24	☐	☐	☐	☐	_ _ _ _ _	49	☐	☐	☐	☐	_ _ _ _ _	74	☐	☐	☐	☐	_ _ _ _ _
25	☐	☐	☐	☐	_ _ _ _ _	50	☐	☐	☐	☐	_ _ _ _ _	75	☐	☐	☐	☐	_ _ _ _ _

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________ (Last) ______________ (First) ______________ (Middle) ADDRESS ______________ (Street/Ship/Unit/Division, etc.)

______________ (City or FPO) (State) (Zip)

RANK/RATE ______ SOC. SEC. NO. ______________ DESIGNATOR ______ ASSIGNMENT NO. ______

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ______________ DATE MAILED ______

SCORE

	1 T	2 F	3	4	
1	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____

	1 T	2 F	3	4	
26	☐	☐	☐	☐	_____
27	☐	☐	☐	☐	_____
28	☐	☐	☐	☐	_____
29	☐	☐	☐	☐	_____
30	☐	☐	☐	☐	_____
31	☐	☐	☐	☐	_____
32	☐	☐	☐	☐	_____
33	☐	☐	☐	☐	_____
34	☐	☐	☐	☐	_____
35	☐	☐	☐	☐	_____
36	☐	☐	☐	☐	_____
37	☐	☐	☐	☐	_____
38	☐	☐	☐	☐	_____
39	☐	☐	☐	☐	_____
40	☐	☐	☐	☐	_____
41	☐	☐	☐	☐	_____
42	☐	☐	☐	☐	_____
43	☐	☐	☐	☐	_____
44	☐	☐	☐	☐	_____
45	☐	☐	☐	☐	_____
46	☐	☐	☐	☐	_____
47	☐	☐	☐	☐	_____
48	☐	☐	☐	☐	_____
49	☐	☐	☐	☐	_____
50	☐	☐	☐	☐	_____

	1 T	2 F	3	4	
51	☐	☐	☐	☐	_____
52	☐	☐	☐	☐	_____
53	☐	☐	☐	☐	_____
54	☐	☐	☐	☐	_____
55	☐	☐	☐	☐	_____
56	☐	☐	☐	☐	_____
57	☐	☐	☐	☐	_____
58	☐	☐	☐	☐	_____
59	☐	☐	☐	☐	_____
60	☐	☐	☐	☐	_____
61	☐	☐	☐	☐	_____
62	☐	☐	☐	☐	_____
63	☐	☐	☐	☐	_____
64	☐	☐	☐	☐	_____
65	☐	☐	☐	☐	_____
66	☐	☐	☐	☐	_____
67	☐	☐	☐	☐	_____
68	☐	☐	☐	☐	_____
69	☐	☐	☐	☐	_____
70	☐	☐	☐	☐	_____
71	☐	☐	☐	☐	_____
72	☐	☐	☐	☐	_____
73	☐	☐	☐	☐	_____
74	☐	☐	☐	☐	_____
75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________________________ (Last, First, Middle) ADDRESS ______________________________ (Street/Ship/Unit/Division, etc.) ______________________________ (City or FPO, State, Zip)

RANK/RATE ________ SOC. SEC. NO. ______________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) __________________ DATE MAILED ________

SCORE

	1 T	2 F	3	4			1 T	2 F	3	4			1 T	2 F	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________________________ ADDRESS ______________________________
Last First Middle Street/Ship/Unit/Division, etc.

____
City or FPO State Zip

RANK/RATE ________ SOC. SEC. NO. ________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ________________ DATE MAILED ________

SCORE

	1 T	2 F	3	4			1 T	2 F	3	4			1 T	2 F	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________________________ ADDRESS ______________________________
Last First Middle — Street/Ship/Unit/Division, etc.

City or FPO State Zip

RANK/RATE ________ SOC. SEC. NO. ______________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ______________ DATE MAILED ________

SCORE

	1 T	2 F	3	4			1 T	2 F	3	4			1 T	2 F	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________________________ ADDRESS ______________________________
Last   First   Middle   Street/Ship/Unit/Division, etc.

City or FPO   State   Zip

RANK/RATE ________ SOC. SEC. NO. ____________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ________________ DATE MAILED ________

SCORE

No.	1 T	2 F	3	4		No.	1 T	2 F	3	4		No.	1 T	2 F	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

PRINT OR TYPE

NRCC Machinery Repairman 3&2
NAVEDTRA 10530-E

NAME ______________________________ ADDRESS ______________________________
Last First Middle Street/Ship/Unit/Division, etc.

City or FPO State Zip

RANK/RATE ________ SOC. SEC. NO. ______________________ DESIGNATOR ________ ASSIGNMENT NO. ________

☐ USN ☐ USNR ☐ ACTIVE ☐ INACTIVE OTHER (Specify) ________________ DATE MAILED ________

SCORE

	1 (T)	2 (F)	3	4			1 (T)	2 (F)	3	4			1 (T)	2 (F)	3	4	
1	☐	☐	☐	☐	_____	26	☐	☐	☐	☐	_____	51	☐	☐	☐	☐	_____
2	☐	☐	☐	☐	_____	27	☐	☐	☐	☐	_____	52	☐	☐	☐	☐	_____
3	☐	☐	☐	☐	_____	28	☐	☐	☐	☐	_____	53	☐	☐	☐	☐	_____
4	☐	☐	☐	☐	_____	29	☐	☐	☐	☐	_____	54	☐	☐	☐	☐	_____
5	☐	☐	☐	☐	_____	30	☐	☐	☐	☐	_____	55	☐	☐	☐	☐	_____
6	☐	☐	☐	☐	_____	31	☐	☐	☐	☐	_____	56	☐	☐	☐	☐	_____
7	☐	☐	☐	☐	_____	32	☐	☐	☐	☐	_____	57	☐	☐	☐	☐	_____
8	☐	☐	☐	☐	_____	33	☐	☐	☐	☐	_____	58	☐	☐	☐	☐	_____
9	☐	☐	☐	☐	_____	34	☐	☐	☐	☐	_____	59	☐	☐	☐	☐	_____
10	☐	☐	☐	☐	_____	35	☐	☐	☐	☐	_____	60	☐	☐	☐	☐	_____
11	☐	☐	☐	☐	_____	36	☐	☐	☐	☐	_____	61	☐	☐	☐	☐	_____
12	☐	☐	☐	☐	_____	37	☐	☐	☐	☐	_____	62	☐	☐	☐	☐	_____
13	☐	☐	☐	☐	_____	38	☐	☐	☐	☐	_____	63	☐	☐	☐	☐	_____
14	☐	☐	☐	☐	_____	39	☐	☐	☐	☐	_____	64	☐	☐	☐	☐	_____
15	☐	☐	☐	☐	_____	40	☐	☐	☐	☐	_____	65	☐	☐	☐	☐	_____
16	☐	☐	☐	☐	_____	41	☐	☐	☐	☐	_____	66	☐	☐	☐	☐	_____
17	☐	☐	☐	☐	_____	42	☐	☐	☐	☐	_____	67	☐	☐	☐	☐	_____
18	☐	☐	☐	☐	_____	43	☐	☐	☐	☐	_____	68	☐	☐	☐	☐	_____
19	☐	☐	☐	☐	_____	44	☐	☐	☐	☐	_____	69	☐	☐	☐	☐	_____
20	☐	☐	☐	☐	_____	45	☐	☐	☐	☐	_____	70	☐	☐	☐	☐	_____
21	☐	☐	☐	☐	_____	46	☐	☐	☐	☐	_____	71	☐	☐	☐	☐	_____
22	☐	☐	☐	☐	_____	47	☐	☐	☐	☐	_____	72	☐	☐	☐	☐	_____
23	☐	☐	☐	☐	_____	48	☐	☐	☐	☐	_____	73	☐	☐	☐	☐	_____
24	☐	☐	☐	☐	_____	49	☐	☐	☐	☐	_____	74	☐	☐	☐	☐	_____
25	☐	☐	☐	☐	_____	50	☐	☐	☐	☐	_____	75	☐	☐	☐	☐	_____

"END OF DOCUMENT"