Naval Education and Training Command

NAVEDTRA 12204
May 1990
0502-LP-213-1100

Training Manual (TRAMAN)

Machinery Repairman 3 & 2

N.F.R.O.

RETURN TO GOV. DOCS. CLERK

DISTRIBUTION STATEMENT A: Approved for public release; distribution is unlimited.

Nonfederal government personnel wanting a copy of this document must use the purchasing instructions on the inside cover.

S/N 0502-LP-213-1100

The terms training manual (TRAMAN) and nonresident training course (NRTC) are now the terms used to describe Navy nonresident training program materials. Specifically, a TRAMAN includes a rate training manual (RTM), officer text (OT), single subject training manual (SSTM), or modular single or multiple subject training manual (MODULE); and an NRTC includes nonresident career course (NRCC), officer correspondence course (OCC), enlisted correspondence course (ECC), or combination thereof.

Although the words "he," "him," and "his" are used sparingly in this manual to enhance communication, they are not intended to be gender driven nor to affront or discriminate against anyone reading this text.

Naval Education And Training Program Management Support Activity

MACHINERY REPAIRMAN 3 & 2

NAVEDTRA 12204

1990 Edition Prepared by
MRCM Reynaldo R. Romero

PREFACE

This Training Manual (TRAMAN) and Nonresident Training Course (NRTC) 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 NRTC provides the usual way of satisfying the requirements for completing the TRAMAN. The set of assignments in the NRTC includes learning objectives and supporting questions designed to help the student learn the materials in the TRAMAN.

1990 Edition

Stock Ordering No.
0502-LP-213-1100

Published by
NAVAL EDUCATION AND TRAINING PROGRAM
MANAGEMENT SUPPORT ACTIVITY

UNITED STATES
GOVERNMENT PRINTING OFFICE
WASHINGTON, D.C.: 1990

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
Atlas Press Company, Clausing Corporation	11-3
Brown & Sharpe Manufacturing Company	11-8, 11-9, 11-13, 11-14, 11-16, 11-17, 13-20
Cincinnati Milacron Marketing Co.	11-1, 11-2, 11-4, 11-5, 11-12, 11-13, 11-15, 11-18, 11-19, 11-20, 11-21, 11-83, 13-10, 13-11, 13-12, 13-15, 13-23, 13-24, 13-25
Cincinnati Inc.	12-1, 12-3
Devlieg-Sundstrand	10-42, 10-43
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
Kearney & Trecker Corporation	11-11
Lars Machine, Inc.	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, and table 12-2
Monarch Tool Company	7-1
Rockford Line	12-13
SIFCO Selective Plating	14-11, 14-12, 14-13, 14-14, 14-15, 14-16, 14-17, 14-18, 14-19, tables 14-3, 14-4, 14-5, 14-6, 14-7, 14-8, 14-9, 14-10, 14-11, 14-12 and all inserts in Chapter 14
South Bend Lathe Works	7-2, 7-5, 7-6, 7-8, 7-9, 7-10, 7-11, 7-12, 7-13, 7-14, 7-15, 7-16, 7-17, 7-27, 7-29, 7-32, 7-33, 7-34, 7-35, 7-36, 7-37, 7-39, 7-40, 8-2, 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, 9-23, 9-24, 9-25, 9-30
Warner & Swasey Co.	10-3, 10-4, 10-5, 10-6, 10-7, 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, 10-30, 10-31, 10-34, 10-35, 10-36, 10-37, 10-38, 10-39, 10-40, 10-41, 13-3, 13-13

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 that 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 repair part.

One of the important characteristics you will gain 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. In fact, you would be surprised at the depth and range of your knowledge and skill compared to your civilian counterpart, based on a somewhat equal length of experience. The machinist trade in private industry tends to break 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 is 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 in planning the required machining operations. You will find that laying out intricate parts is not so difficult with this knowledge. A third job description is "Setup Man," a job which requires considerable knowledge and skill, all within what you can expect to gain as a Machinery Repairman. A setup 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 are concerned. This person is 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 related to design limitations, shop math and metallurgy. There are many other related fields in which 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 ASSIGNMENT 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 detail 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-motivated toward learning all you can to increase your ability as a Machinery Repairman and that you seek advice from sources off of 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.

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 type 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 other 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 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 instruction provides the theory of basic operating procedures, safety precautions and certain project procedures, while time spent in the shop provides 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 and specific content of the course may vary from time to time to accommodate the needs of the fleet. You will have no difficulty 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.

TRAINING MANUALS AND NONRESIDENT TRAINING COURSES

Navy training manuals and nonresident training 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 training manuals and nonresident training 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 training manuals for the paygrades below yours, 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 efficient 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 or her 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 revision from time to time—some at regular intervals, others as the need arises. When using any publication that is subject to 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 do your work or 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 and *Mathematics*, Vol. 2, NAVEDTRA 10071. These two volumes provide a review of the mathematics you will need in shop work.

2. *Blueprint Reading and Sketching*, NAVEDTRA 10077, provides information on blueprint reading and layout work.

3. *Tools and Their Uses*, NAVEDTRA 10085, 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 the *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 *Basic Military Requirements* training manuals as well as instructions prepared by your command. In this section we shall look at some of the more common safety hazards you will find in a machine shop and some of the precautions 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 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. Wear safety glasses or goggles 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 of protection is safety-toe shoes. Granted, the additional weight of the steel reinforced toe does not make them the most comfortable shoes you can wear, but they do offer outstanding foot protection and 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 it is caught 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 someone bend over and pick up a heavy object by using his or her 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. 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, 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 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 prevent 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 it is subjected to during its intended use. Excessive pressures could cause machine or tool failure followed by injury.

Whenever you are operating a machine, give it your total concentration. Save daydreaming for 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 onboard 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 which 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 you perform maintenance on your equipment.

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 your are going to do before you do it.

PURPOSES, BENEFITS, AND LIMITATIONS OF THE PLANNED MAINTENANCE SYSTEM

You will soon find, if you have not done so already, that the continued operation of machinery depends on systematic and dedicated maintenance. The following paragraphs contain a brief discussion on the purposes, benefits, and limitations of the Navy's formal maintenance system, the Planned Maintenance System. You will be involved in the Planned Maintenance System, to some degree, throughout your career in the Navy.

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

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 they contain.

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.

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 NAVSEA's control. 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.*

MANUFACTURER'S 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 need 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. 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, and so forth. You will find this information helpful in your everyday shop work.

2. Appendix II contains 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.

[redacted]

CHAPTER 2

TOOLROOMS AND TOOLS

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

This training manual will provide information which applies to the tools and instruments used primarily by a Machinery Repairman. You can find additional information on tools that are commonly used by the many different naval ratings in *Tools and Their Uses*, NAVEDTRA 10085.

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) is available.

Operating and maintaining a toolroom is simple if the correct procedures and methods are used to set up the 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 use of space.

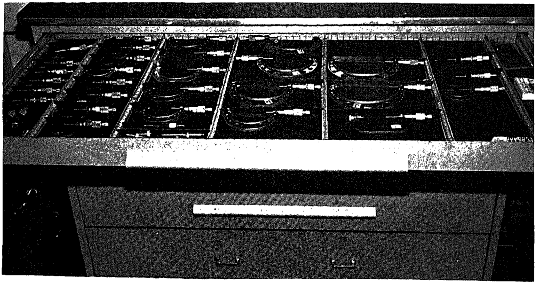
Hammers, wrenches, and other tools that do not have cutting edges may normally be stored in bins. They also may be segregated by size or other designation. Tools with cutting edges require more space to prevent damage to the cutting edges. Usually these tools are stored on shelves lined with wood, on 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 of the boards when the ship is rolling. Precision tools (micrometers, dial indicators and so forth) should be stored 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.

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 storage of 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

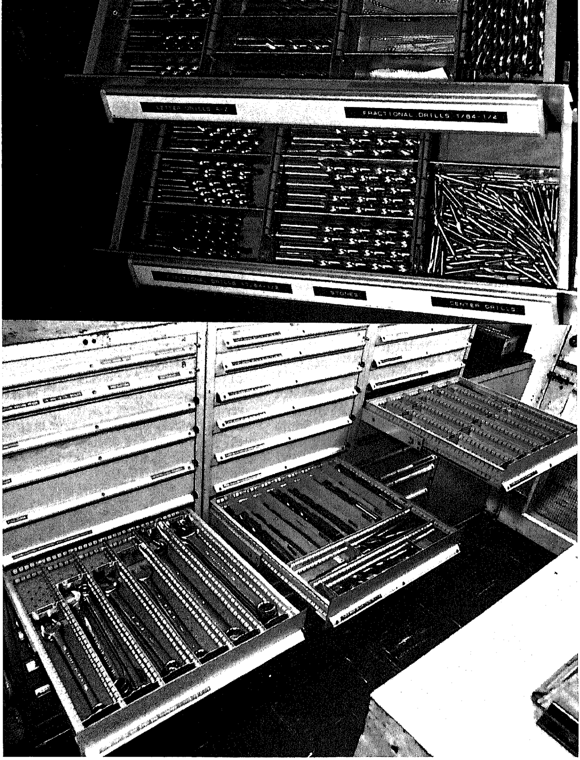
28.333.1

Figure 2-1.—Method of tool storage.



28.334

Figure 2-2.—Method of tool storage.



28.335

Figure 2-3.—Method of tool storage.

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 their metal surfaces a light coat of oil. Check all precision tools upon issue and return to determine if they are accurate. 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 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 custody of the tools. Be sure that a method of identifying a borrower with the tool is established, and that provisions are 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 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. 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 for each item, 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. You can find information on procurement of tools and supplies in *Military Requirements for Petty Officer 3 & 2*, NAVEDTRA 10056.

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. At sea the toolroom can be especially hazardous 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 in or issued from the toolroom, there is a real danger present.

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 GAUGES

Practically all shop jobs require measuring or gauging. You will most likely measure or gauge 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 "gauge," 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 gauges 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 gauging 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 time to measure it, set the caliper at a reading slightly greater than the final dimension desired; then, at intervals during turning operations, gauge, or "size," the workpiece with the locked instrument. After you have reduced the workpiece dimension 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 GAUGES

You can adjust adjustable gauges by moving the scale or by moving the gauging surface to the dimensions of the object being measured or gauged. 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 move the measuring surface to the dimensions of the object being measured.

Dial Indicators

Dial indicators are used by Machinery Repairman 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

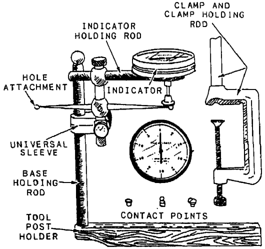


Figure 2-4.—Universal dial indicator.

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 post holder

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

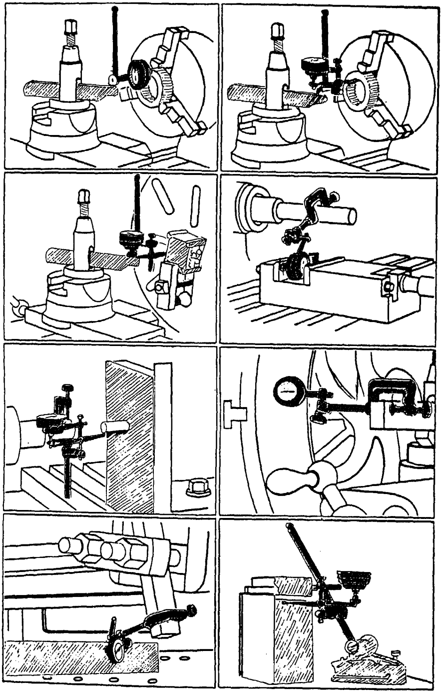
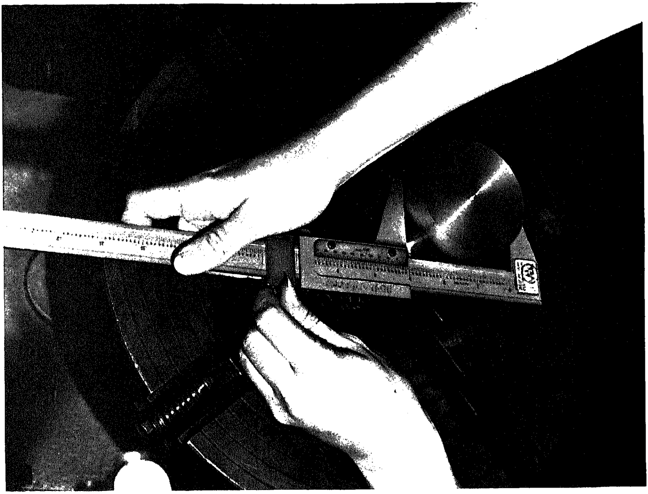


Figure 2-5.—Applications of a dial indicator.

(0.005) of an 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 of an (0.100) inch in graduations on its face and has a total range of two hundred thousandths (0.200) of an 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 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.

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



28.314

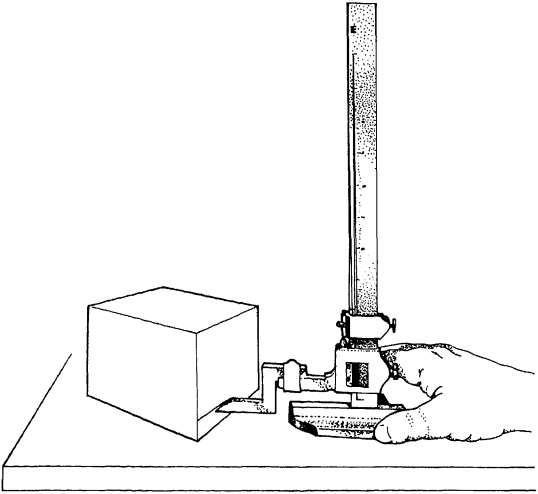
Figure 2-6.—Vernier caliper.

out work for machining operations or to check the dimensions on surfaces which have been machined. Attachments for the gauge include the offset scriber shown attached to the gauge 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 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

as a vernier caliper.

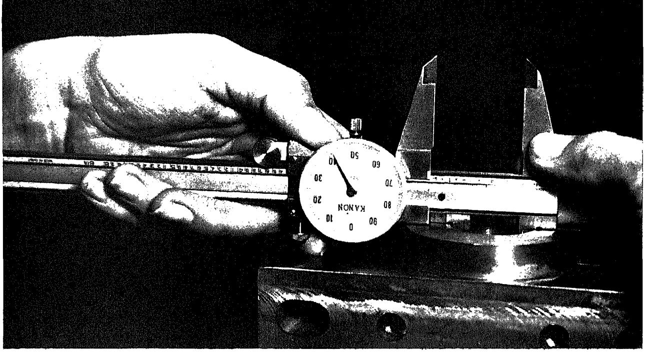
Dial Vernier Caliper

A dial vernier caliper (fig. 2-8) 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.

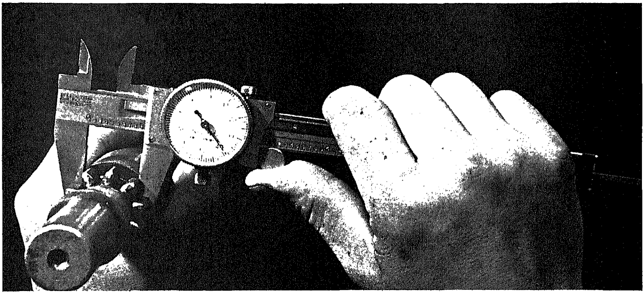


28.4(28D)

Figure 2-7.—Vernier height gauge.



A. MEASURING THE INSIDE

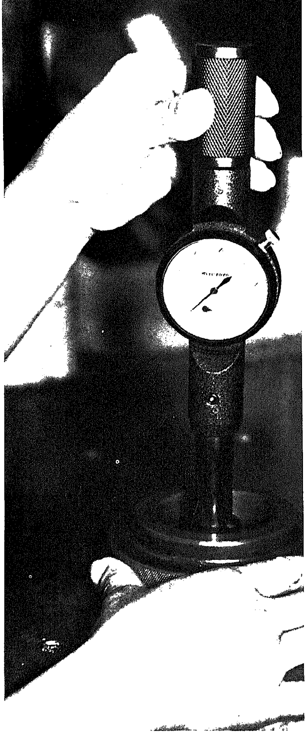


B. MEASURING THE OUTSIDE

28.315

Figure 2-8.—Dial vernier caliper.

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 gauge. The dial bore gauge (fig. 2-9) 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 gauge, outside micrometer, or vernier caliper can be used to preset the gauge. A dial bore gauge 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 gauge.



28.316

Figure 2-9.—Dial bore gauge.

Internal Groove Gauge

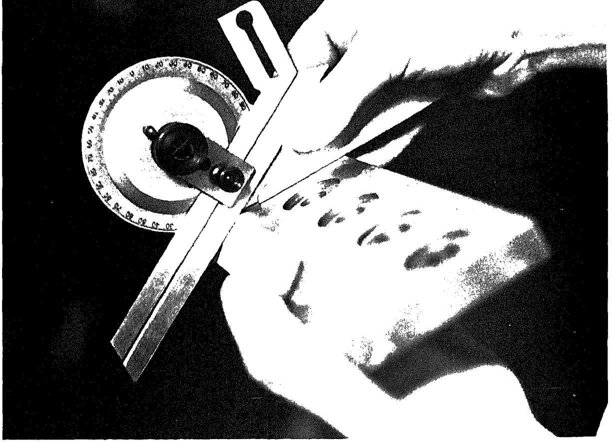
The internal groove gauge is very useful for measuring the depth of an O-ring groove or other recesses inside a bore. This tool lets you measure a deeper recess and one located farther back in the bore than if you were to use an inside caliper. As with the dial bore gauge, this tool must be set with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

Universal Vernier Bevel Protractor

The universal vernier bevel protractor (fig. 2-10) 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.

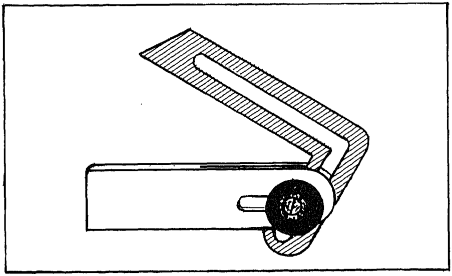
Universal Bevel

The universal bevel (fig. 2-11), 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



28.317

Figure 2-10.—Universal vernier bevel protractor.



28.5

Figure 2-11.—Universal bevel.

... the desired angle.

Gear Tooth Vernier

A gear tooth vernier (fig. 2-12) 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 Gauge

The cutter clearance gauge (fig. 2-13) is one of the simplest to use, yet it is suitable for gauging clearance on all styles of plain milling ... than 8 teeth and a diameter range from 1/2 inch to 8 inches. To gauge a tooth with this instrument, bring the surfaces of the "V" into contact with the cutter and lower the gauge blade to the tooth to be gauged. Rotate the cutter sufficiently to bring the tooth face into contact with the gauge blade. If the angle of clearance on the tooth is correct, it will correspond with the angle of the gauge blade. Cutter clearance gauges that have an adjustable gauge blade for checking clearance angles of 0°-30° on most common cutter styles are also available.

Adjustable Parallel

The adjustable parallel in figure 2-14 consists of two wedges connected on their inclined surfaces

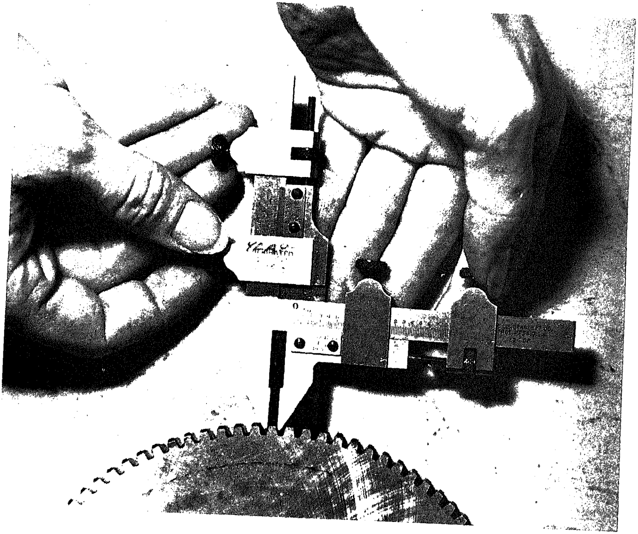
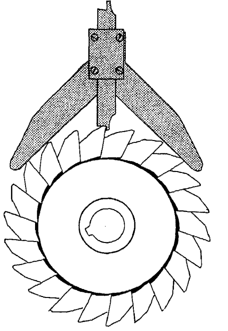


Figure 2-12.—Gear tooth vernier. 28.318

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.



28.7

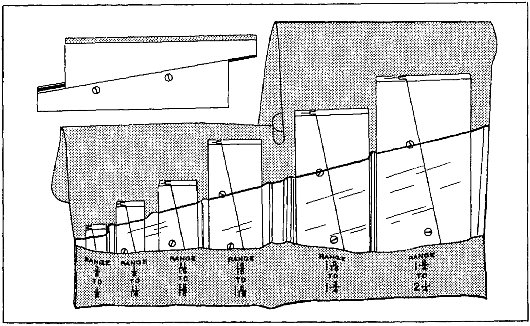
Figure 2-13.—Cutter clearance gauge.

Surface Gauge

A surface gauge (fig. 2-15 is useful in gauging or measuring operations. It is used primarily in layout and alignment work. The surface gauge is commonly used with a scriber to transfer dimensions and layout lines. In some cases a dial indicator is used with the surface gauge to check trueness or alignment.

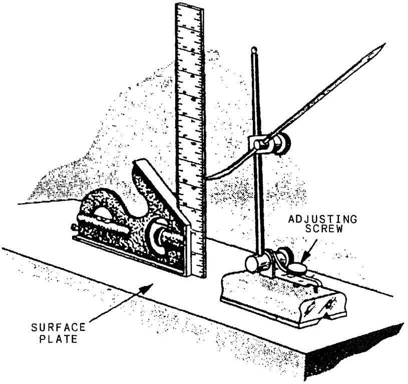
FIXED GAUGES

Fixed gauges cannot be adjusted. They can generally be divided into two categories, graduated and nongraduated. The accuracy of your work, when you use fixed gauges, will depend on your ability to determine the difference between the work and the gauge. For example, a skilled machinist can take a dimension accurately to within 0.005 of an inch or less when



28.6

Figure 2-14.—Adjustable parallel.



28.9

Figure 2-15.—Setting a dimension on a surface gauge.

using a common rule. Practical experience in the use of these gauges will increase your ability to take accurate measurements.

Graduated Gauges

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

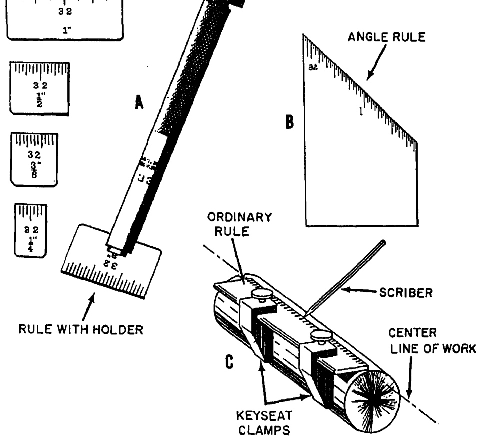
RULES.—The steel rule with holder set (fig. 2-16A) is convenient for measuring recesses. It has a long tubular handle with a split chuck for holding the ruled blade. The chuck can be adjusted by a knurled nut at the top of the holder, allowing the rule to be set at various angles. The set has rules ranging from 1/4 to 1 inch in length.

The angle rule (fig. 2-16B) is useful in measuring small work mounted between centers on a lathe. The long side of the rule (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 keyset rule (fig. 2-16C). 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 so 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

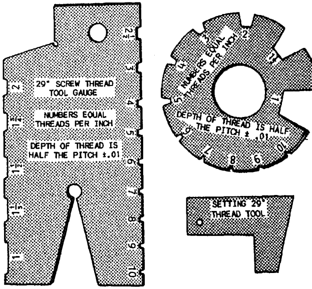


28.10

Figure 2-16.—Special rules for shop use.

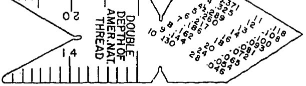
1 inch on the scale is equivalent to an actual measurement of 2 inches; a 12-inch long scale of this type is equivalent to 24 inches. A scale, therefore, gives proportional measurements instead of the actual measurements obtained with a rule. Like rules, scales are made of wood, plastic, or metal, and they generally range from 6 to 24 inches.

ACME THREAD TOOL GAUGE.—This gauge (fig. 2-17) 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 gauge. The width of the flat on the point of the tool varies according to the number of threads per inch. The gauge provides different slots for you to use as a guide when you grind the tool. Setting the tool up in the lathe is simple. First, ensure that the tool is centered on the work as far as height is



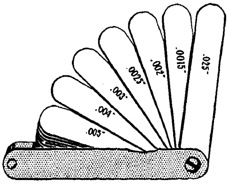
5.16.1

Figure 2-17.—Acme thread gauges.



5.16.2

Figure 2-18.—Center gauge.



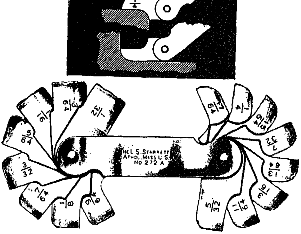
4.19

Figure 2-19.—Feeler (thickness) gauge.

concerned. Then, with the gauge edge laid parallel to the centerline of the work, adjust the side of your tool until it fits the angle on the gauge very closely.

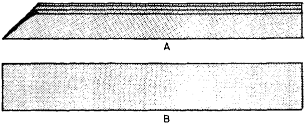
CENTER GAUGE.—The center gauge (fig. 2-18) is used like the Acme thread gauge. Each notch and the point of the gauge has an included angle of 60°. The gauge is used primarily to check and to set the angle of the V-sharp and other 60° standard threading tools. The center gauge is also used to check the 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 GAUGE.—A feeler (thickness) gauge, like the one shown in figure 2-19, is used to determine distances between two closely mating surfaces. This gauge is made like a jackknife with blades of various thicknesses. When you use a combination of blades to get a desired gauge 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 get the "feel" of using a feeler gauge correctly is to practice with the gauge on openings of known dimensions.



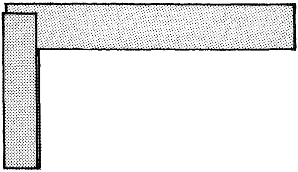
28.338

Figure 2-20.—Fillet or radius gauges.



28.11

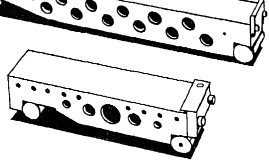
Figure 2-21.—Straightedge.



28.12

Figure 2-22.—Machinist's square.

RADIUS GAUGE.—The radius gauge (fig. 2-20) is often underrated in its usefulness to the machinist. 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



28.339

Figure 2-23.—Sine bars.

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 gauges have both concave (inside curve) and convex (outside curve) radii in the common sizes.

Nongraduated Gauges

Nongraduated gauges 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 2-21. 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.

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 the two balance points sit on the resting pads.

MACHINIST'S SQUARE.—The most common type of machinist's square has a hardened steel blade securely attached to a beam. The steel blade is NOT graduated. (See fig. 2-22.) 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. You should take the same care of machinist's squares, in storage and use, as you do with a micrometer.

SINE BAR.—A sine bar (fig. 2-23) is a precision tool used to establish angles which required extremely close accuracy. When used in conjunction with a surface plate and gauge 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. Be sure to repair any scratches, nicks, or other damage before you use the sine bar, and take care in using and storing the sine bar. Instructions on using the sine bar are included in chapter 3.

PARALLEL BLOCKS.—Parallel blocks (fig. 2-24) 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

28.319

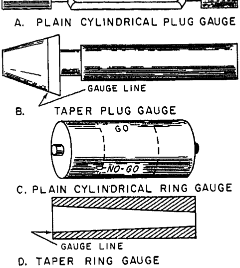
Figure 2-24.—Parallel blocks.

pairs and in standard fractional dimensions. Use care in storing and handling them 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.

GAUGE BLOCKS.—Gauge blocks are used as master gauges to set and check other gauges 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 gauge 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 gauge 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 gauge blocks has enough different size blocks that you can establish 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 gauges 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 GAUGES.—A ring gauge (fig. 2-25) is a cylindrically-shaped disk that has a precisely ground bore. Ring gauges are used to check machined diameters by sliding the gauge over the surface. Straight, tapered, and threaded diameters can be checked by using the appropriate gauge. The ring gauge is also used to set other measuring instruments to the basic dimension required for their operation. Normally, ring gauges are available with a "GO" and a "NO GO" size that represents the tolerance allowed for the particular size or job.



28.340

Figure 2-25.—Ring gauge and plug gauge.

A plug gauge (fig. 2-25) is used for the same types of jobs as a ring gauge 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. The 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.

MICROMETERS

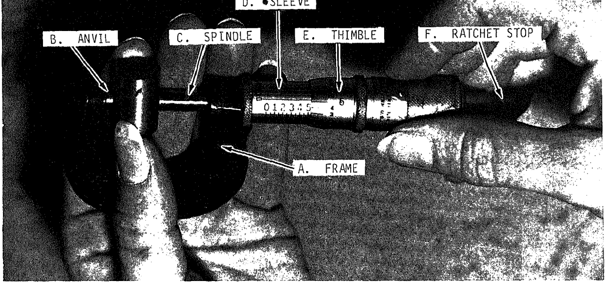
Micrometers are probably the most often used precision measuring instruments in a machine shop. There are many different types, each designed to permit measurement of surfaces for various applications and configurations of workpieces. The degree of accuracy obtainable from a micrometer also varies, with the most common graduations being from one thousandth of an inch (0.001) to one ten-thousandth of an inch (0.0001). Information on the correct

Uses, NAVEDTRA 10085. A brief description of the more common types of micrometers is provided in the following paragraphs.

An outside micrometer (fig. 2-26 and 2-27), often called a micrometer caliper, or mike, is used to measure the thickness or the outside diameter

28.320

Figure 2-26.—Common types of micrometers.



28.321

Figure 2-27.—Nomenclature of an outside micrometer caliper.

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 inside micrometer (fig. 2-26) is used to measure inside diameters or between parallel surfaces. They are available in sizes ranging from 0.200 inch 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 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 you feel a very slight drag on the in and out movement. There are no specific guidelines on the number of positions within a hole that should be measured. If you are checking for taper, you should take measurements as far apart as possible within the hole. If you are checking for roundness or concentricity of a hole, 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. 2-26) is used to measure the depth of holes, slots, counterbores, recesses, and the distance from a surface to some 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. 2-26) 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 below.

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 bought 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 an outside diameter.

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 of a groove micrometer measurement.

CARE AND MAINTENANCE OF GAUGES

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 it 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 you take a measurement. Always use the friction slip ratchet if there is one on the instrument. Too much pressure on the knurled

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 you do not obtain a zero reading after making the correction described above, you will need to adjust the spindle-thimble relationship. 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

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 completing the alignment, back the spindle away from the anvil, and retighten the barrel sleeve locking nut. Recheck for zero setting, to be sure you did not disturb the thimble-sleeve relationship 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 you have loosened the lock screw, 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 you did not move the anvil out of position while you tightened the setscrew.

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 the 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 gauge blocks or other accurate standards. If 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 you complete the gauge block test, 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. You can correct this condition 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 Gauges

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

1. Always loosen a gauge into position. Forcing, besides causing an inaccurate reading, is likely to force the arms out of alignment.

Heavy pressure will force the two scales out of parallel.

3. Prior to putting a vernier gauge 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. Never oil a sluggish dial. Oil will compound the problems. Use a suitable cleaning solvent 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 of information that you should study for details on specific equipment include the *NAVSHIPS' Technical Manual*, manufacturers' technical manuals, and 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 of 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 concerning 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 training manual, *Blueprint Reading and Sketching*, NAVEDTRA 10077.

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 a 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

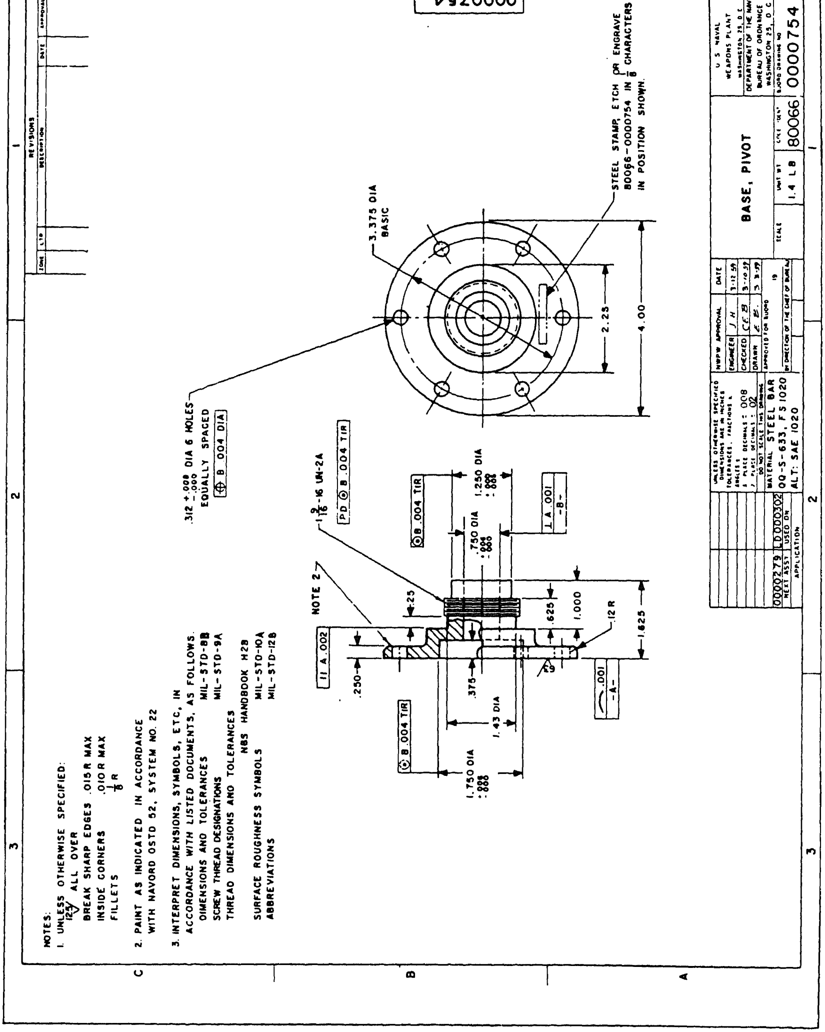


Figure 3-1.—Detail drawing of base pivot.

projection without dimension lines. An isometric projection shows how the part will look when it is 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, scale used, pattern number, material required, assembly or subassembly print number to which the part belongs, and name 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 of its data.
- 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 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 establish the predominant surface pattern. The 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. As the tracer arm travels across the surface 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 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

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 ARROW-HEAD OR DOT AT ONE END USED TO INDICATE A PART, DIMENSION OR OTHER REFERENCE	¼ X 20 THD.
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 ARROW-HEADS TO SHOW DIRECTION VIEWED	

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

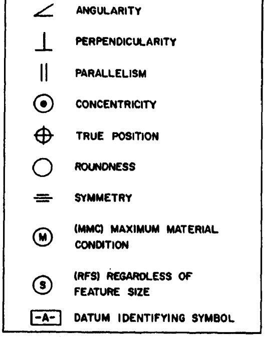


Figure 3-3.—Geometric characteristic symbols.

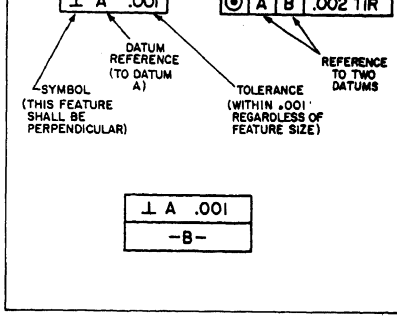


Figure 3-4.—Feature control symbol incorporating datum reference.

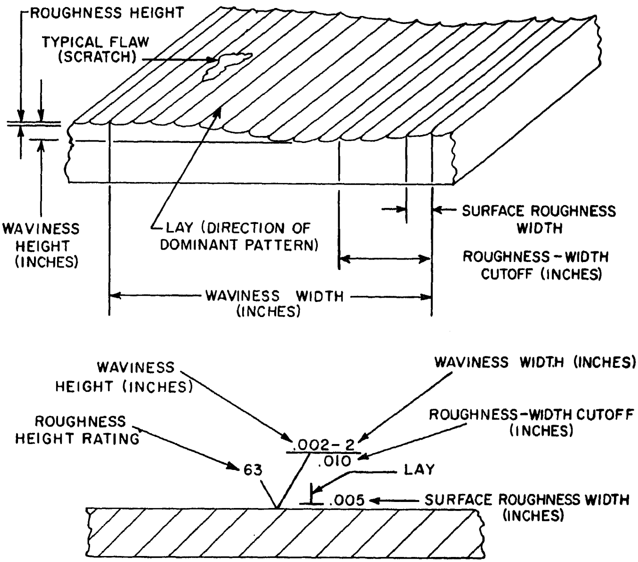


Figure 3-5.—Relation of symbols to surface characteristics.

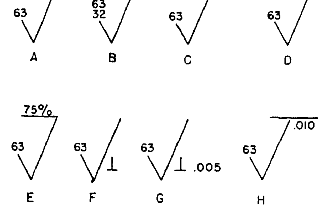


Figure 3-6.—Symbols used to indicate surface roughness, waviness, and lay.

mum permissible roughness height rating, if two are 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

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.	
P[3]	LAY PARTICULATE, NON-DIRECTIONAL, OR PROTUBERANT	

[3] The "P" symbol is not currently shown in ISO Standards. American National Standards Committee B46 (Surface Texture) has proposed its inclusion in ISO 1302-"Methods of indicating surface texture on drawings."

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

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 seven symbols that indicate the direction of lay.)

The roughness width value is shown just to the right of and parallel to the lay symbol. The roughness width cutoff is placed immediately below the extension line and to the right of the 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.

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.

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.

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 MEASUREMENTS

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 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 = 0.0625 inch. A chart giving the decimal equivalents of the most common fractions is shown in Appendix I.

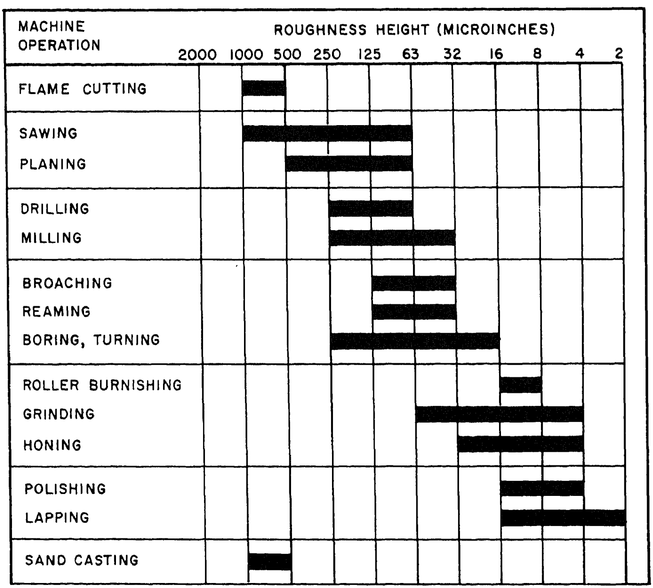


Figure 3-8.—Roughness height values for machine operations.

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 subdivided 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 × 2.540 = 3.4925 cm. Further, 0.0008 inch converted to millimeters is 0.0008 inch × 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 distinction between them.

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 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 expected, by 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 they are 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.

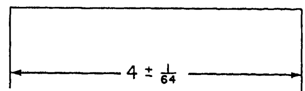


Figure 3-9.—Basic dimension and tolerance.

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 they are 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 were 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 $+0.005$, -0, or $+0$, -0.005, but with the required value not necessarily 0.005. One way to get a better understanding of a clearance allowance, or an interference allowance, is to make a rough sketch of the piece and add dimensions to the sketch where they apply.

Layout is the term used for 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 benefit you 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*, NAVEDTRA 10069; *Mathematics, Volume II,* NAVEDTRA 10071; *Tools and Their Uses*, NAVEDTRA 10085, and *Blueprint Reading and*

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 commonly used layout dyes 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 gauge, 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 to establish angles for layout and inspection work. The sine bar must be used in conjunction with a surface plate and gauge 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 gauge blocks.

Many other special tools, which you may make, 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.

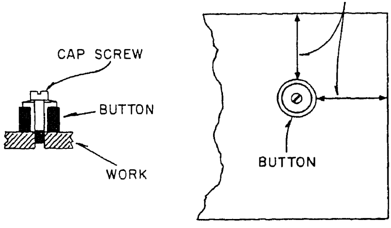


Figure 3-10.—Toolmaker's buttons and their application.

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 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 creating 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 be 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 cored holes in the casting. The layout lines then must be within the machining allowance on all surfaces.

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.

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 scribe a line on stock with a combination square, place the squaring head on the edge of

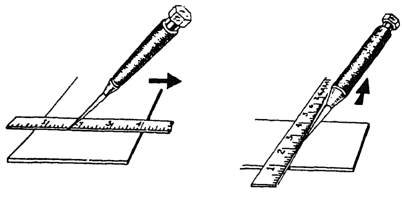


Figure 3-11.—Using a scribe.

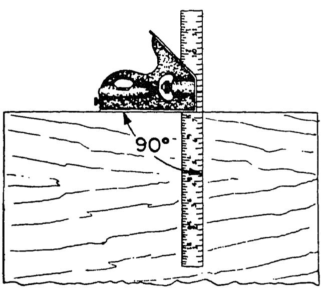


Figure 3-12.—Using the combination square.

along either edge of the stock. 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.

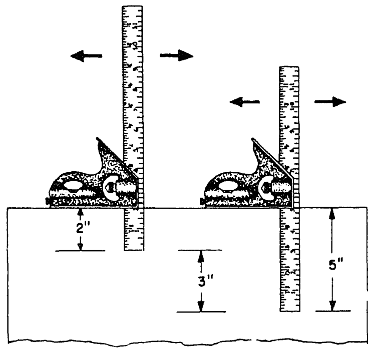


Figure 3-13.—Laying out parallel lines with a combination square.

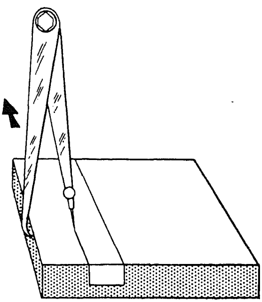


Figure 3-14.—Laying out a parallel line with a hermaphrodite caliper.

in figure 3-14, so the curved leg maintains contact with the edge while the other leg scribes the line. Hold the caliper so 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 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 the desired angle is 60°. Tighten the screw to hold the setting.

Hold the body of the protractor head in contact with the 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.

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.

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 point to keep the point of the dividers from slipping out of position.

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

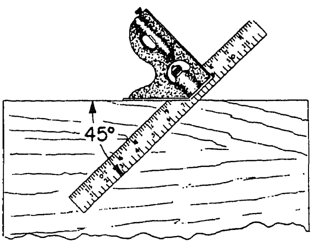


Figure 3-15.—Laying out a 45° angle.

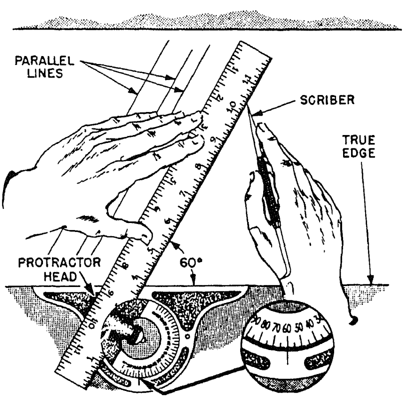


Figure 3-16.—Laying out angular lines.

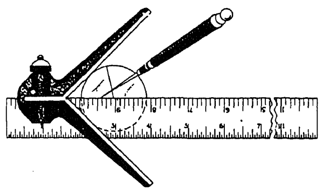


Figure 3-17.—Locating the center of round stock.

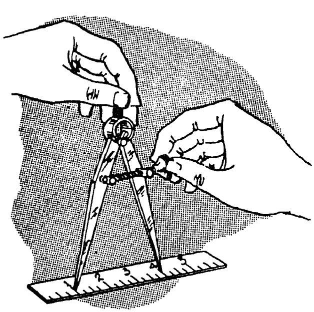


Figure 3-18.—Setting a divider to a dimension.

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 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.)

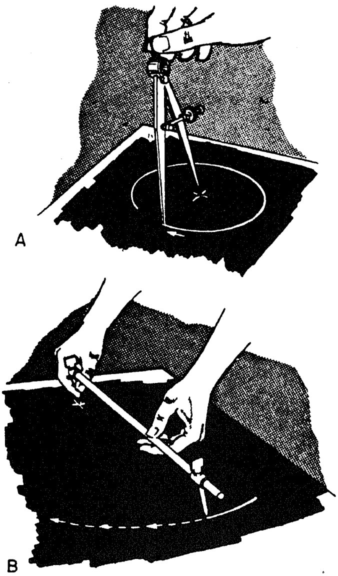


Figure 3-19.—Laying out circles.

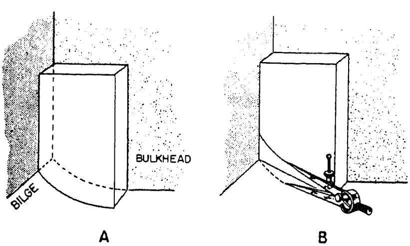


Figure 3-20.—Laying out an irregular line from a surface.

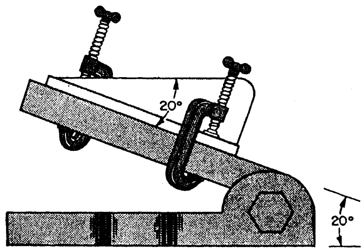


Figure 3-21.—Angle plate.

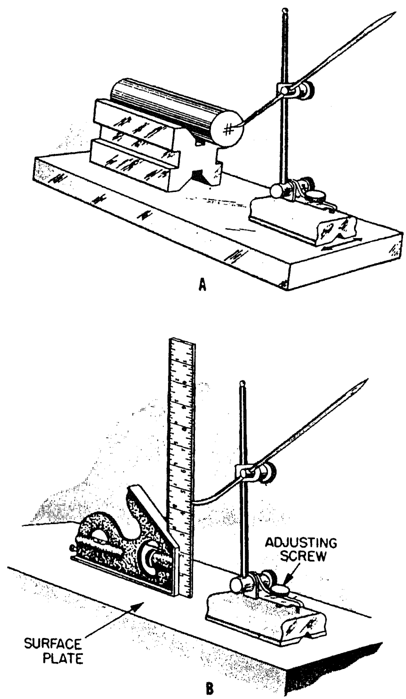


Figure 3-22.—Setting and using a surface gauge.

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 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.)

USING THE SURFACE PLATE.—The surface plate is used with such tools as parallels, squares, V-blocks, surface gauges, 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 gauge V-block combination used in laying out a piece of stock. To set a surface gauge for height, first of a combination square, as shown in B of figure 3-22. Secure the scale so the end is in contact with the surface of the plate. Move the surface gauge 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 side opposite is made up of the gauge 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 gauge 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

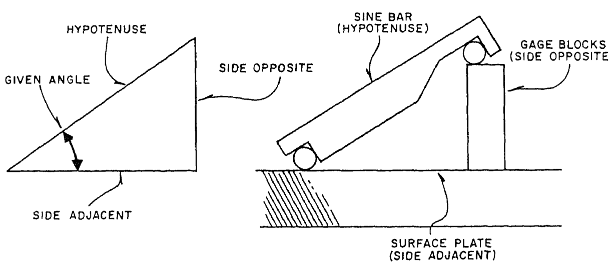


Figure 3-23.—Setup of the sine bar.

to a table of natural trigonometric functions and find the sine of 30°5′. Then multiply the sine value by 10 inches: 0.50126 × 10 = 5.0126, to find the height of the gauge 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 gauge 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 and sweat and other corrosive agents. Regular inspection of the sine bar will locate such defects before they are able to affect the accuracy of the bar. 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

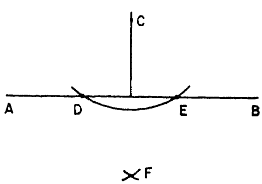


Figure 3-24.—Layout of a perpendicular from a point to a line.

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.

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. 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.

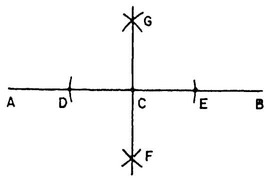


Figure 3-25.—Layout of a perpendicular from a point on a line.

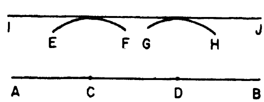


Figure 3-26.—Layout of a parallel line.

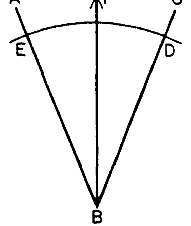


Figure 3-27.—Bisecting an angle.

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.

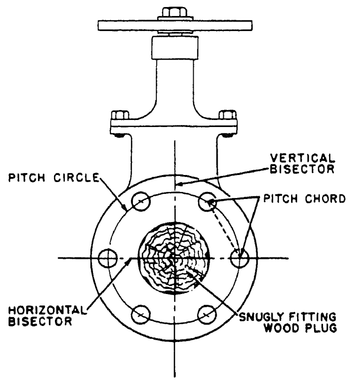


Figure 3-28.—Flange layout terminology.

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 the standard position. If you are working on a "per sample" job, follow the layout of the sample.

Assuming that you have been given information concerning 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 valve flanges 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 make the drawn lines clearly visible.

3. Locate the center of each flange with a surface gauge, 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 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 a right angle 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 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 you must correct 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.

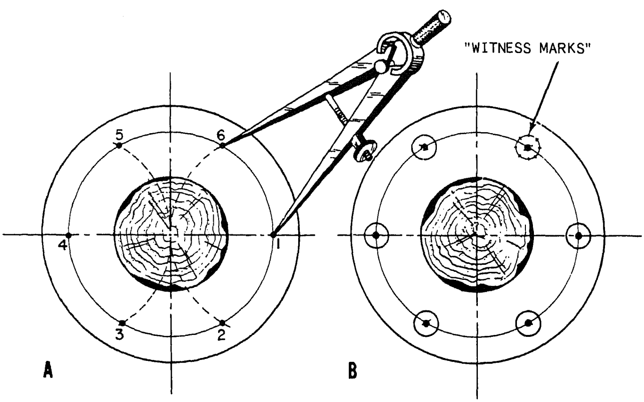


Figure 3-29.—Development of a 6-hole flange.

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

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

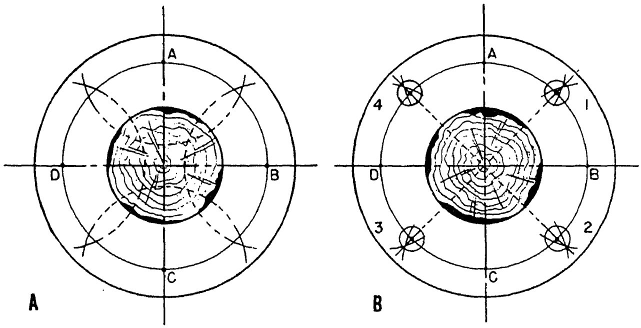


Figure 3-30.—Four-hole flange development.

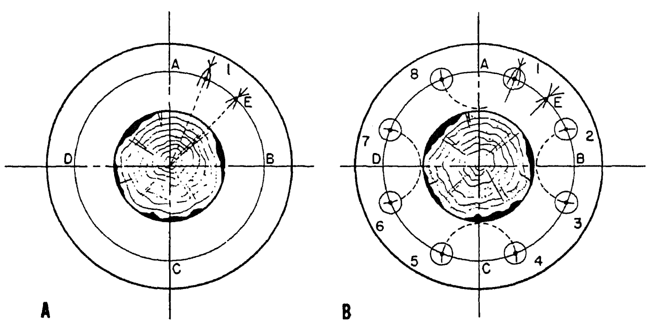


Figure 3-31.—Eight-hole flange development.

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 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 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.

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

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 can 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. Whenever you repair equipment, you should note such things as fastening devices, fits between mating parts, and the uses of gaskets and packing. 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 in correct locations and positions when the unit is reassembled.

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 in removing 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.

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 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 their loss. Use tags or other methods of marking the parts to identify 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.) Pay close attention to details of the equipment you are taking apart and fix in your mind how the parts fit together. When you

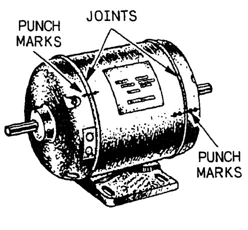


Figure 3-32.—Mating parts location marks.

heavy pressure is required to separate parts. An overlooked pin, key, or setscrew that locks parts in place can 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 machining 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 similar 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 that you make 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 until 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 you have worked out all the high spots, smooth out (or replace if necessary) the bluing on the shaft or mandrel and repeat the process until you have produced an acceptable seating pattern. 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 the bearing is placed into service.

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 piece of 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.

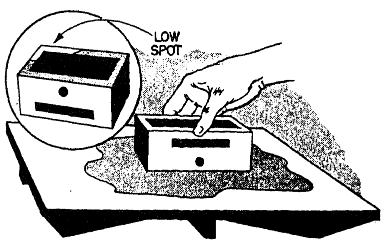


Figure 3-33.—Checking a surface.

Hand Reaming

When you need a round hole that is accurate in size and smooth in finish, reaming is the process that you will probably select. There are two types of reaming processes—machine reaming and hand reaming. Machine reaming requires a drill press, lathe, milling machining 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 is the method you will probably use most in precision benchwork.

A hand reamer has a straight shank and a square machined on its 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 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.

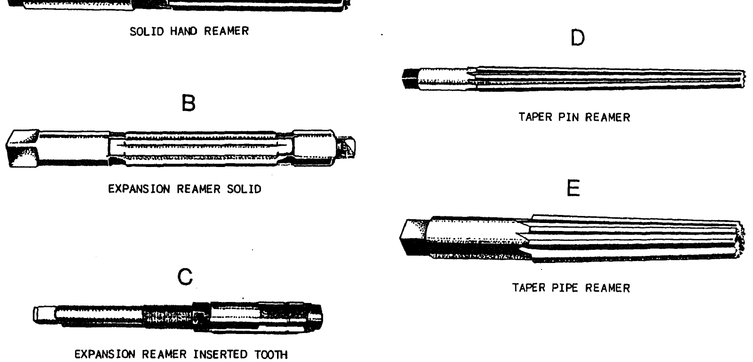


Figure 3-34.—Hand reamers.

The expansion reamer in C of figure 3-34 has a much greater range for varying its size. Each reamer is adjustable to allow it to overlap 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 accurately 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. Insert a center in the spindle of the machine and place it in the center hole in the shank of the reamer to guide the reamer.

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 the hole and the surface 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 ships 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 both 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 you ream 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 have 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 (fig. 3-35).

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.

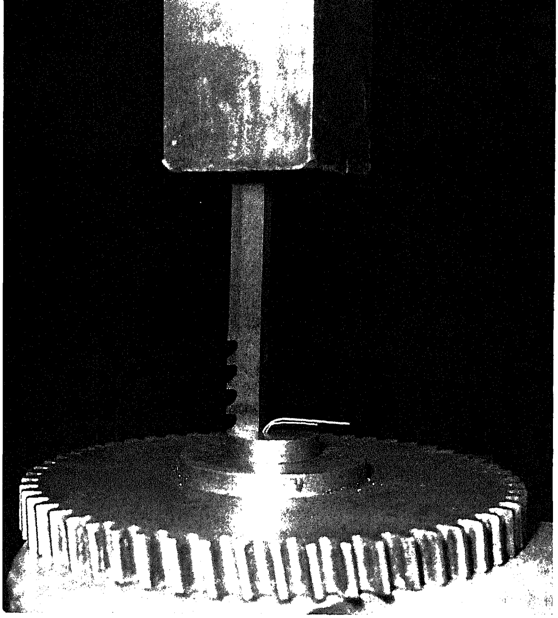
Most Navy machine shop applications involve the use of either a mechanical or a hydraulic press to force the broach through the part. A considerable amount of pressure is required to broach, so be sure that the setup is rigid and that all applicable safety precautions are strictly observed. A slow even 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, for internally threaded parts, or hand dies for externally 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 use 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 (fig. 3-36) 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 roughly 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 catalog special ordered from a tap manufacturer. The information in this section concerns 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.



28.33

Figure 3-35.—Broaching a keyway on a gear.

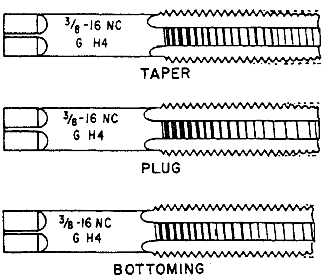


Figure 3-36.—Set of taps.

Taps usually come in a set of three for each different diameter and number of threads per inch. A taper, or starting tap (fig. 3-36), 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

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.

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. 3-36) 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 additional common thread series symbols are NF, American National Fine; NS, American National Special; NEF, American National Extra Fine; and 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. 3-36) 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 × 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 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 0.060 inch and each tap number after that increases in diameter by 0.013 inch. As an example:

Size 0 = 0.060 inch dia.

Size 3 = 0.099 inch dia. [0.060 + 3 × 0.013]

Size 14 = 0.242 inch dia. [0.060 + 14 × 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 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 you may use 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%.

($DS = TD - \frac{1}{N}$). As an example, the drill size for a 1/4 - 20 NC tap is required 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 inch and a number 7 drill has a diameter of 0.201 inch. 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 60% thread height is desired for a 1/4 - 20 NC tap, the drill size is figured as follows:

Step 1: $DS = 1/4 - .60 \times 2(0.032)$

Step 2: $DS = 0.250 - .60 \times 0.064$

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. You should consider this in planning the operation. Additional information on drilling holes is in chapter 5.

and shape. You MUST be sure that the part cannot vibrate loose and be thrown out of the vise or off of 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 you have done this, insert the tap drill into the drilling machine or drill press and drill the hole. If the hole is very large, 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 you make 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 wrench over the square shank, (fig. 3-37) and slowly turn the tap while applying a

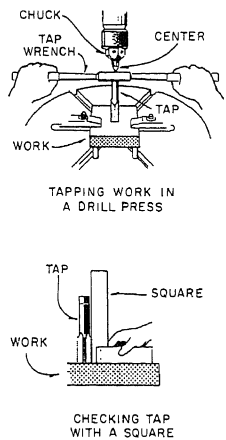


Figure 3-37.—Starting a tap.

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 the part is machined flat) and bring the square into contact with one set of teeth. Do the same check on the next set of teeth in either direction around the tap (fig. 3-37). If the tap is not perpendicular or square with the surface at both points, back it 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 tap. 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 is acceptable.

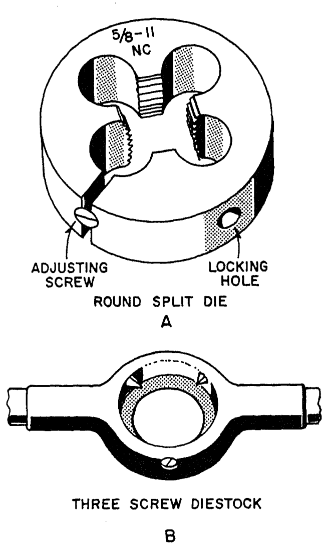


Figure 3-38.—Die and diestock.

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. 3-38). 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. 3-38), 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 will cut a thread that has a 5/8-inch diameter and 11 American National Coarse threads per inch. 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 squarely 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 die several turns, then look carefully at the die and the part to ensure that they are square to one another. Threads that are deeper on one side than the other indicate a misaligned die. Turn the die about three 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 you reach the desired thread length. If you are cutting the threads 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 you can use in 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 locking pliers and removed. Use a scribe first to remove as many as of the chips as possible 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 nearby. 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 locking pliers. 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 you have loosened the tap in the hole. When you have removed the tap, examine the hole and threads closely to ensure that no fragments of the tap or jagged threads remain to cause problems when you use another tap to finish or clean up the threads.

Another method is to use a punch and apply sharp blows to the broken tap. You will probably use this method 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 you do this type of of the tap. As you break a fragment of the tap 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 into 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. Request the assistance of a Hull Maintenance Technician (HT). After the stud is welded to the tap, you can apply a more even pressure in removing the tap if you grind a square on the top of the stud so that you can use a tap wrench. The heat generated by the welding process could have expanded the tap slightly so that when it cooled and contracted, it may have loosened slightly. On the other hand, the tap may bind even more and the structure and condition of the surrounding metal may have changed.

If the tap is broken off below the surface of the part, you can use a tool called a tap extractor (fig. 3-39) to remove it. You should try this method first as it does no damage to the threads. Tap extractors are 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 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

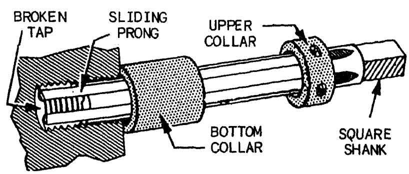


Figure 3-39.—Tap extractor.

possible to the top 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 wench 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 you can use with any of the removal methods if the particular situation lends itself to their use. As previously mentioned, you can apply penetrating oil around the threads. You can also apply a controlled heat to the area surrounding the tap to cause expansion. Be very careful to limit the heat so 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 you must conduct 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 you mix 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 the skin with water for at least 15 minutes and seek medical attention. You will use nitric acid often in identifying metals. 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 later in this manual.

Classes of Fit

The following information concerns 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. The resulting fit can be a clearance (loose) fit or interference (tight) fit, or a transitional (somewhere between loose and tight) fit. These three general types of fit are further divided into classes of fit, 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 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. The classes of fit 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 specified 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. Some parts that fall ing rings (loose removal).

TRANSITIONAL FITS.—Transitional fits are subdivided into three types known as locational clearances, locational transition and locational interference fits. Each of these three subdivisions contains different classes of fit which provide either a clearance or an interference allowance, depending on the intended use and class. All of the classes of fit 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 fit. 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 installed without the 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 fit, 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 −0.00023 inch. These classes of fits are used when parts must be located very accurately while maintaining alignment and rigidity. They are not

other part.

INTERFERENCE FITS.—There are five classes of fit 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 fit 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 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, 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 assembly. 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. You must consider all of these variables 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 electro-hydraulic pump.

Regardless of the type of press equipment you use, be sure to operate it correctly. The only way you can determine the amount of pressure a hydraulic press exerts is by watching the pressure gauge. 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 whenever you use a 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 of the surface of the workpiece.

- Watch the pressure gauge. You cannot determine the pressure exerted by "feel." If you begin to apply excessive pressure, release the pressure and double check 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 the parts must be pressed directly affects the required pressure, and increased interference allowance requires greater pressure. 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. Do this with great care, and only with proper supervision. The operation of the oxyacetylene torch, as used in heating parts only, is explained in this chapter along with safety precautions which you must observe when you use 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 or a separate cutting torch. Accessories include a spark lighter to light the torch; an apparatus wrench to fit the various connections, 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 on board ship is usually taken from compressed gas cylinders.

than 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 an 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 connected 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 has a 3/16- or 1/4-inch inside diameter, and contains one or two plies of fabric. For heavy duty welding and handcutting 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 have the same grade but differ in color and have different types 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 switching the hoses 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

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, you must keep the tip 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 clear it with wire tip cleaners of the proper diameter, or with soft copper wire. Do not clean tips 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 gauges 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 they cannot be upset. Remove the protective caps.
2. Crack (open) 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. Turn the regulator screws out until you feel little or no resistance then open the cylinder valves slowly. Then 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 gauge 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 you need 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 screw the mixing head onto the torch body. Tighten the mixing head/tip assembly by hand and adjust the tip to the proper angle. Secure this adjustment by tightening the assembly with the wrench provided with the torch.

9. Adjust the working pressures—Adjust the acetylene pressure by turning the acetylene gauge 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 ıor 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 with one part of acetylene. The bottled oxygen, together with the oxygen in the air, 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 to 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 extinguishes the flame and prevents flashback. (Flashback is discussed later.) Then close the oxygen needle valve.

2. Close both the 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 above procedure whenever your work will be interrupted for an indefinite period. If your work is to stop for only a few minutes, securing the cylinder valves and draining the hoses is not necessary. However, for any indefinite work in areas other than the shop, it is a good idea to remove the pressure regulators and the 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, you must observe certain safety precautions 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 you use cylinders, 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 you use the cylinder so 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 the 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.

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 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. Avoid acetylene working pressures in excess of 15 pounds per square inch. Oxygen cylinder pressure must likewise be reduced to a suitable low working pressure; high pressure may burst the hose.

- Stow all cylinders carefully according to prescribed procedures. Store cylinders 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. Stow all cylinders in an upright position. If they are not stowed in an upright position, do not use them until they have been allowed to stand upright for at least 2 hours.

will not fall on your legs or feet, on the hose and cylinder, or on any flammable materials. Be sure a fire watch is posted as required to prevent accidental fires.

Be sure you and anyone nearby wear flameproof protective clothing and shaded goggles to prevent serious burns to the skin or the eyes. A number 5 or 6 shaded lens should be sufficient for your heating operations.

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 and a flashback are two common problems encountered 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 characteristized 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 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, and (4) you have purged the system of air before using it. Purging the system of air is especially necessary when the hose and torch have been newly connected or when a new cylinder is put into the system.

PURGING THE OXYACETYLENE TORCH.—

1. Close the torch valves tightly, then slowly open the cylinder valves.
2. Open the acetylene 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

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 into three general groups: threads, keys, and pins.

The selection of the correct fastener (specified in blueprints, list 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 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 fit described in the Unified (UNC, UNF) and the American National (NC, NF) thread systems explained in chapter 9. The range of general applications for any given fastener. However, some equipment requires such specialized fasteners that the fasteners can only be used for that specific purpose. 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 squarely or hexagonally shaped head. Bolts are designed to be inserted into holes slightly larger than their diameter. A nut is attached to the threaded end to draw the mating parts together. As a general rule, the width of the the diameter of the threads. The length of the thread ranges from 2 times the thread diameter plus 1/4 inch to a point just below the head, depending on the intended use. The length of the bolt is measured from the under side of the head to the tip of the threaded portion. It is best to use a bolt that has an unthreaded 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 class 2A for the bolt and class 2B for the nut. This 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.

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

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 onto the other end of the stud. One type of stud is continuously threaded, 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 portion in the center of the stud. The unthreaded portion may have the same diameter as the major diameter of the threads, or it may be recessed to provide clearance. A continuously threaded stud generally has a class 2A or 3A fit to allow relative ease in assembly. A stud with the center portion 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 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 jamming either 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 externally threaded part to which it will be attached. Nuts come in either square or a hexagon shapes 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. You may 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 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 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. Available point shapes are a cone (90° point), a cup (recessed point), an oval, a flat, and a half-dog (a short, reduced diameter). The point selection depends on whether the setscrew is intended to prevent slippage of a pulley or gear on a shaft or to hold 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 revolutions per minute and horsepower. If the equipment specifications do not provide this information, you may obtain it from most handbooks for machinists. Setscrews are normally made of hardened steel, although setscrews are available for use when saltwater or corrosive liquids are involved.

Screw Thread Inserts

A screw thread insert (fig. 3-40) is a helically wound coil 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 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 in 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, and so on. Screw thread inserts are normally made from stainless steel; however phosphor bronze and nickel alloy inserts 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 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. 3-41). There are several 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 charactistics. 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 3-40) solidly engages the shoulder or recess on the end of the tool. Then install the insert by turning the tool until the correct depth is reached. Remove the tool by reversing the direction of rotation.

After you have the insert properly installed, break off the tang to prevent any interference with the fastener that will be screwed into the hole. A tang break-off tool is available for all insert sizes

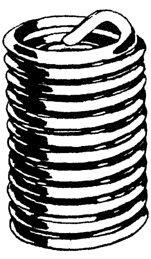


Figure 3-40.—Screw thread insert.

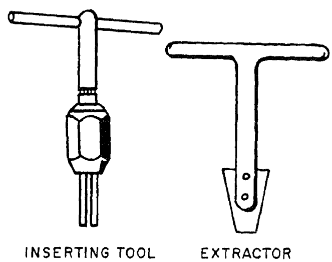


Figure 3-41.—Screw thread insert tools.

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. 3-41). There are several different sized tools that cover a given range of insert sizes; be sure you select the correctly sized tool. Insert the tool into the hole so 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 are 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.

3. Select the correct size of 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 previously 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 top of the hole. Remove all chips from the hole.

4. Tap the hole. Use standard tapping procedures in this step. If the tapping procedure calls for both roughing and finishing taps, be sure to use both taps prior to attempting to install the insert. Use lubricants to improve the quality of the threads. When you have completed the tapping, 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

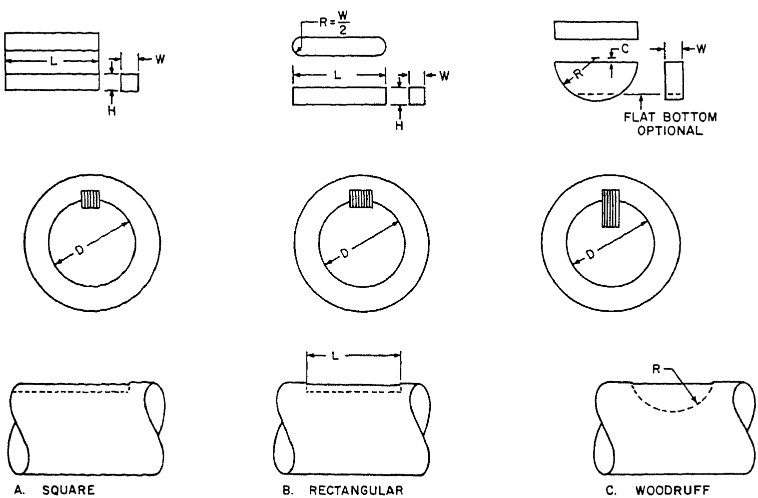


Figure 3-42.—Types of keys and keyseats.

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. Avoid using similar metals such as a stainless insert and a stainless bolt to prevent galling and seizing of the threads.

Keyseats and Keys

Keyseats are grooves cut along the axis of the cylindrical surface of a shaft and the bored hole in a hub. Metal keys of various shapes are fitted into these grooves to transfer torque between the shaft and the 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 when replacing a key to prevent selecting one that will not perform as required.

Square keys (fig. 3-42A) are recommended for applications where the shaft diameter is 6 1/2 inches and below, while rectangular keys (fig. 3-42B) are recommended for shaft diameters over 6 1/2 inches. Some applications may require that two keys be installed to drive equipment under high torque conditions. The width and height of a key depend 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 fit from 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

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 showing the recommended key sizes for different shaft diameters and the allowable tolerance for each of the classes of fit are available in most handbooks for machinists.

The ends of square or rectangular keys are often prepared with a radius equal to one-half of the width as shown in the top illustration of figure 3-42B. 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. 3-42C) 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.

For 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 applications such 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

round metal stock which are used primarily to lock nuts in place on bolts. All pins come in a variety of standard 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 the installation of gaskets, packing, or seals to prevent leakage. Gaskets are used mainly for sealing fixed 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° of 1000°F are within the range that these gaskets can effectively seal. Each application requires a specific gasket and substitutions should not be considered. When installing this gasket, thickness required for the particular application.

Synthetic rubber and cloth inserted rubber gaskets are used on freshwater and seawater systems with pressures of 50 to 400 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 a deterioration of the gasket resulting in contamination of the system and a hazardous situation if a leak should develop.

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 exists, 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 pits 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 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.

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 pits, 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 during 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, oscillating, or rotary motion exists 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 value 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 according to 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 interchangeably. A thermosetting 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 presented 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, the 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 stresses are applied. The side of the metal on the outside of the bend undergoes tensile stress as it is stretched, while the metal on the inside of the bend is squeezed under compression 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 by 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.

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 and the hardness of metals is often present. As the hardness of a metal is increased, the tensile strength is also increased and vice versa.

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 on 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 applying 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,

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, its 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 the load is applied and removed many times. In this way, a shaft may break after months of use even though 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.

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 either coating or alloying them with other metals that are corrosion resistant.

HEAT RESISTANCE

Heat resistance is the 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 austenitic 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 of 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.

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, free-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 with lower ratings.

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 condition), 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 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.

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	64

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	80

Cast Steel

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

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, based on 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 is brittle and has a comparatively low tensile strength. Therefore, it 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

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. The addition of sulfur and phosphorous limits their ability to be formed hot.

LOW CARBON STEEL (0.05% TO 0.30% carbon), 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% carbon) 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 affects 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 used 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 hardenability, corrosion resistance, toughness, and wear resistance. The most 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 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 of their cutting edges 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.

are included among the nonferrous metals. You will find that these metals, and their 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, copper ranks next to silver; it also offers a high resistance to saltwater corrosion.

Copper becomes hard when worked but can be softened easily by being heated to a cherry red and then cooled. Its strength, however, decreases rapidly at 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 contain 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 either 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 alloying metals. Some of the more common alloys, their chemical analyses 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% tin (Sn), 3%-5% zinc Zn), 0.3% lead (Pb), 0.15% iron (Fe), 0.05%

alloy, the term "copper-tin" is used only to designate the major alloying 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%-9% 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 effects 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 one alloy to be more effective than another for a given application. Remember this if 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, 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 (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

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, you can improve the metal's machinability considerably by annealing it immediately before machining. K-Monel is used for the shaft sleeves on many pumps because of its resistance to the heating 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 hard 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, in which the metal is dipped into a bath of molten zinc; (3) sherardizing, in which 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 between dissimilar metals caused by electric currents set up by saltwater. Zinc plates bolted on the hull, especially near the propellers, decompose quite rapidly, but in doing so, greatly reduce localized pitting of the hull steel.

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 has 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

You must have knowledge of the standard designations of metals and the systems of marking metals used by the Navy and industry so 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. You may find several different designations which refer to a metal with the same chemical composition, or 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 systems where a similar metal is involved. Some of the numbering systems that you may 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 designations for a metal 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 by the various metal producers is voluntary and it could be some time before any widespread uses 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 those of the 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 they can be identified readily.

FERROUS METAL DESIGNATIONS

You should be familiar with the SAE and AISI systems of steel classifications. These systems, 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" 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, you can

(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% 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 most easily machined steels 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).

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 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 sometimes used to designate certain alloys.

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, there is no special control on individual impurities. Digits 1 through 9, indicate 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, and so on 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 letter or letter/number is included as a temper designation. The temper designation follows the four-digit alloy number and is separated from it

aluminum-copper alloy, solution heat treated, then artificially aged; T6 is the temper designation. The aluminum alloy temper designations and their meanings are:

- W 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. Do not depend only on the markings to ensure that you are using the correct metal. 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.

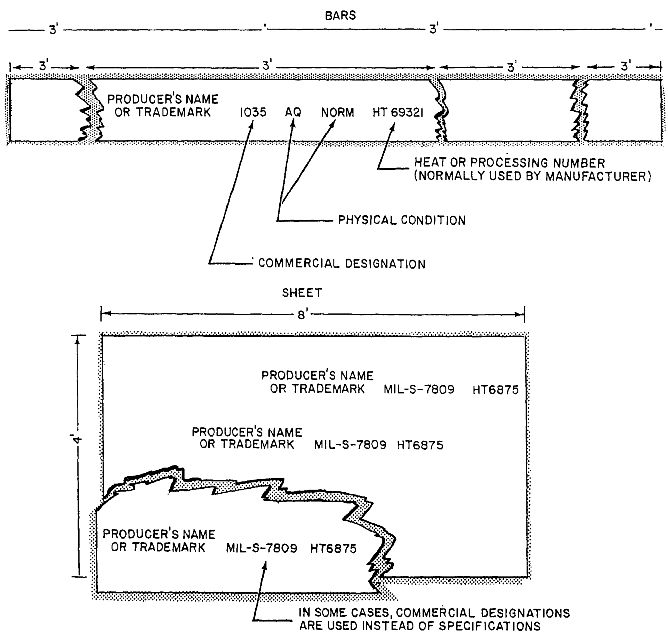
CONTINUOUS IDENTIFICATION MARKING

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 portions 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 Standard 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



tion 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 below:

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

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 another. There are, however, a number of 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, however, do not provide positive identification and should not be used in critical situations where a specific metal is required. 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 you perform these tests, you should have a known sample of the desired material and make a

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.

comparison. You will also need good lighting, a strong permanent magnet, and access to a lathe. A word of caution: when you perform these tests, DO NOT be satisfied with the results of only one test. Use as many tests as possible so 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 shape of the 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. Many shops maintain specimens of known composition for comparison with unknown samples.

Proper lighting conditions are essential for good spark testing practice. You should perform the test in an area where there is enough light, but should avoid harsh or glaring light. 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. A bench grinder is mounted inside the cabinet. 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, because air drafts can change the tail of the spark stream and may result in improper identification of the sample.

The speed of the grinding wheel and the pressure you exert on the samples greatly affect 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 at 3600 rpm. This provides a surface speed of 7,537 feet per minute.) The pressure of the piece against the wheel has a similar effect: the more pressure applied to the test piece, the larger and longer the spark stream will be. Hold the test piece lightly but firmly against the wheel with just enough pressure to prevent the piece from bouncing. Remember, you must apply the same amount of pressure to the test specimen as to the sample are testing.

The grain size of the grinding wheel sh be from 30 to 60 grains. Be sure to keep the w clean at all times. A wheel loaded with part of metal will give off a spark stream of the of metal in the wheel mixed with the spark str of the metal being tested. This will ten confuse you and prevent you from prop identifying the metal. Dress the wheel before begin spark testing and before each new te a different metal.

The spark test is made by holding a sa of the material against a grinding wheel. sparks given off, or the lack of sparks, assi identifying the metal. The length of the s stream, its color, and the type of sparks ar features for which you should look. There four fundamental spark forms produced wh sample of metal is held against a power grin (See fig. 4-2.) Part A shows shafts, b breaks, and arrows. The arrow or spearhe characteristic of molybdenum, a metallic ele of the chromium group which resembles iron is used for forming steel-like alloys with car

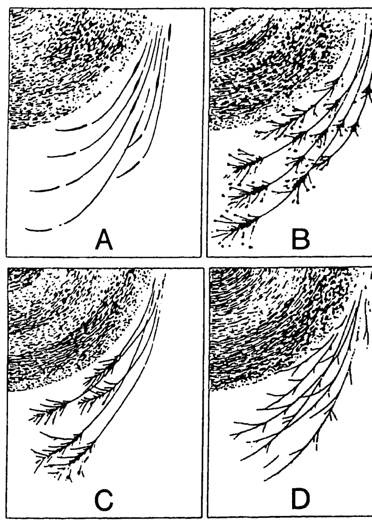


Figure 4-2.—Fundamental spark forms.

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 particles. If you examine the small steel particles under a microscope when they are cold, you will see that they are hollow spheres with one end completely blown away.

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 so that you 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. 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 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.

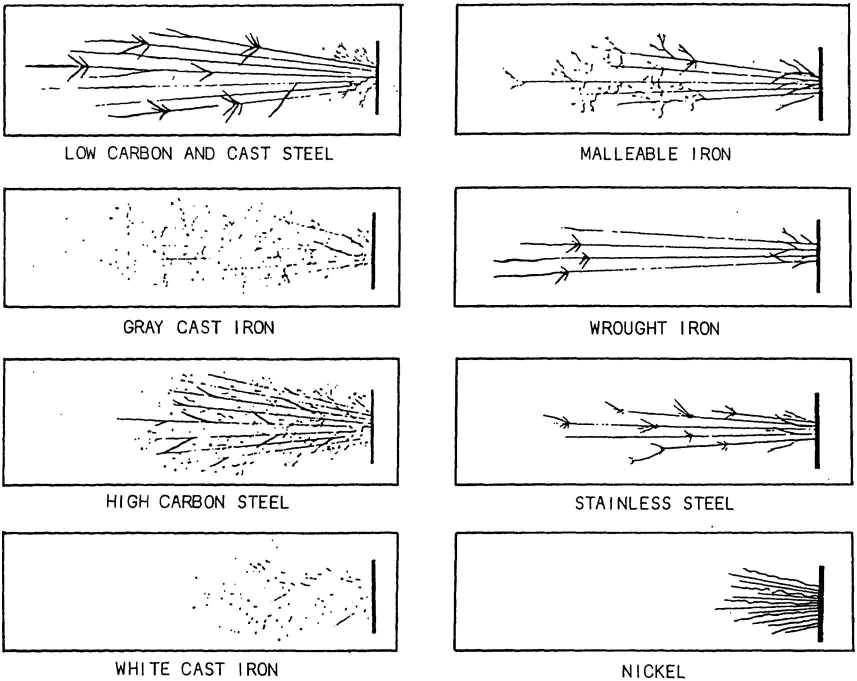
White cast iron produces a spark stream approximately 20 inches long (see fig. 4-3). The volume of sparks is small with many small, repeating sparklers. The color of the spark stream close to the wheel is red, while the outer end of the stream is straw-colored.

Gray cast iron produces a stream of sparks about 25 inches long. It is small in volume with fewer sparklers than in the stream 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 long. 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 long. The stream has a large volume with 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 long, 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.



11.37

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

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

Monel 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 situation. Here again, experience is the best teacher.

ACID TEST

The nitric acid test is the most commonly used test for metal identification in 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. Whenever you perform an acid test, be sure to observe the following safety precautions.

- 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 you spill any acid, dilute it with plenty of water to weaken it so it can be safely swabbed up and disposed of.

flush 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 for 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, 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 operations, including heating and cooling of a metal in its solid state, that 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 a metal to and holding it at a suitable temperature and then cooling it 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.

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** (Stellate): Heat to between 2100° and 2150°F. Hold at this temperature for a suitable time, depending on thickness. Follow by rapid cooling in a quenching medium.

BRASS: Annealing to relieve stress may be done at a temperature as low as 600°F. Fuller anneals may be done with increased temperatures. Larger grain size and loss of strength will result from too high temperatures. Do NOT anneal at temperatures exceeding 1300°F. Slowly cool the brass 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 air drafts.

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 its 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. The steel must then be 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

... 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 a 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 TEST

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 the person conducting the test. Before discussing the hardness test equipment, let us consider hardness itself, and the value of such information to a Machinery Repairman.

... 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 materials 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.

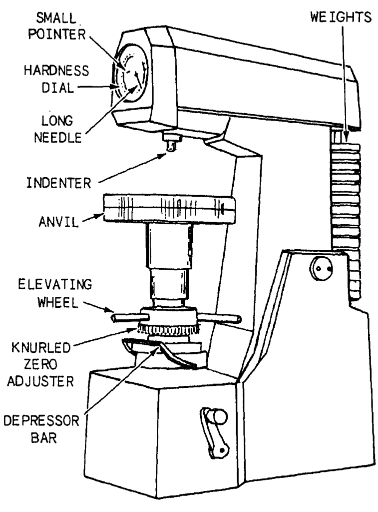
ROCKWELL HARDNESS TEST

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° spheroconnical 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 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.



102.90

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

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.

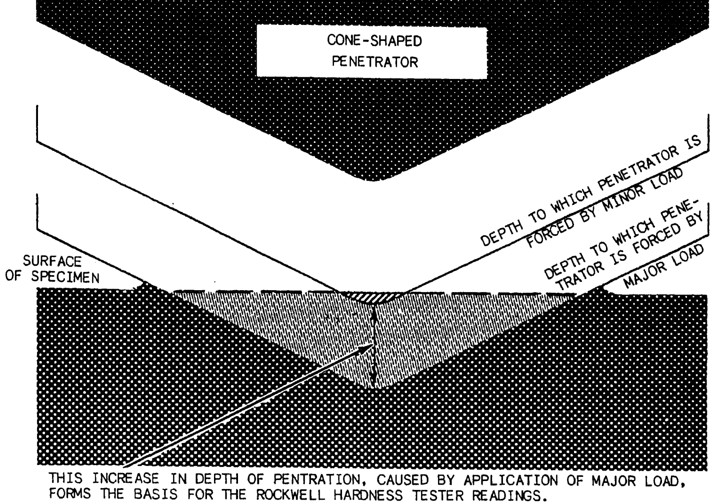
The Rockwell superficial tester differs from the normal Rockwell tester in the amount of load applied to perform the test and in the kind of scale used to interpret the results. When the major loads on the normal tester are 100 and 150 kg, the major loads on the superficial tester are 15, 30, and 45 kg. One division on the dial gauge of the normal tester represents a vertical displacement of the indenter of 0.002 millimeter (mm). One division of the dial gauge 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 that, if they were thicker, would usually 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.

If you have properly prepared a sample and have selected the appropriate penetrator and weights, you can use the following step-by-step procedure to operate a Rockwell tester:

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 gauge is nearly vertical and slightly to the right of the dot.

3. Watch the long pointer on the gauge; continue raising the work with the elevating wheel until the long pointer is nearly upright—within approximately five divisions, plus or minus, on



126.87

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

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 you have made the test with the 1/16-inch ball and a 100-kilogram weight, take the reading from the red, or B, scale. If you have made the test with the spheroconical penetrator and a weight of 150 kilograms, take the reading from the black, or C scale. (In the first example prefix the number 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) height 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 the 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 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 metals 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 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 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

Hardness No. 3,000 kg	Rockwell Hardness No. C Scale	Strength Approximate X 1,000 psi	Brinell Hardness No. 3,000 kg	Rockwell Hardness No. C Scale	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			

[redacted]

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		

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 proven 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 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 as hard, as steel; some are as pliable as rubber; some are more transparent than glass; and some are lighter than aluminum.

within 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 that 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 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 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-based material is thin and quite brittle, breaking if bent sharply, but canvas-based material is difficult to break. As layers are added to paper-based 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 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 you may 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

You can use several types of saws—bandsaw, jigsaw, circular saw—to cut blanks from plastic stock. Watch the saw speed 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, you may prefer a special coolant.

Lathe Operations

Lathe operations are substantially the same for plastics as for metals, except for the 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

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.

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.

with high-speed steel tools and 500 to 1500 fpm with carbide-tipped tools.

Finishing Operations

Plastics must be finished to remove tool marks and produce a clean, smooth surface. Usually, sanding and buffing are sufficient for this purpose.

You can remove surface scratches and pits by hand sandpapering with dry sandpaper of fine grit. You can also wet sand by hand, with water and abrasive paper of fine grade. If you need to remove a large amount of material, use sanding wheels or disks.

After you have removed the pits and scratches, buff the plastic. You can do this on a wheel made of loose muslin buffs. Use tripoli and rouge buffing compounds, depositing a layer of the compound on the outside of the buffing wheel. Renew the compound frequently.

When you buff large flat sheets, 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.

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 we discuss 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:

- DO NOT operate a power saw that you are not fully qualified and authorized to operate.
- Wear goggles or a face shield at all times when you are operating a power saw.
- NEVER make adjustments to the saw or relocate the stock to be sawed while the saw is in operation.
- Keep your hands as far away as possible from the saw blade 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

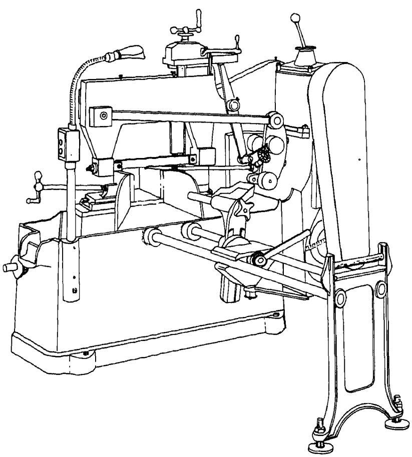
The power hacksaw is found in many Navy machine shops. It is used for cutting bar stock, pipe, tubing, or other metal stock. The power hacksaw consists of a base, a saw frame, and a work-holding device. Figure 5-1 is an illustration of a standard power hacksaw.

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 (off center) 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 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 it 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) keeps long stock from falling when being cut.

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



11.18

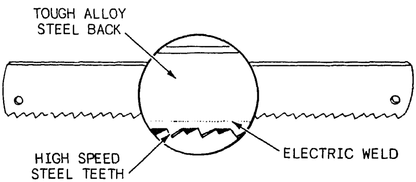
Figure 5-1.—Standard power hacksaw.

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 differ by 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 material composition 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 teeth from the blade. In general, select blades according to the following information:

1. Coarse (4 teeth per inch), for soft steel, cast iron, and bronze.



11.19

Figure 5-2.—Hacksaw blade.

tool steel and high-speed stock.

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

A power hacksaw will have one of three types of 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, in which weights are placed on the saw frame and shifted to give more or less pressure of the saw blade against the material being cut.

To prevent unnecessary wear on the back sides of the saw blade teeth, the saw frame and blade are automatically raised clear of the surface being cut on each return stroke. The rate of feed or the pressure exerted by the blade on the cutting stroke 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 a soft, small diameter piece of stock. 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.

The speed of 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 good 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 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.

CONTINUOUS FEED CUTOFF SAW

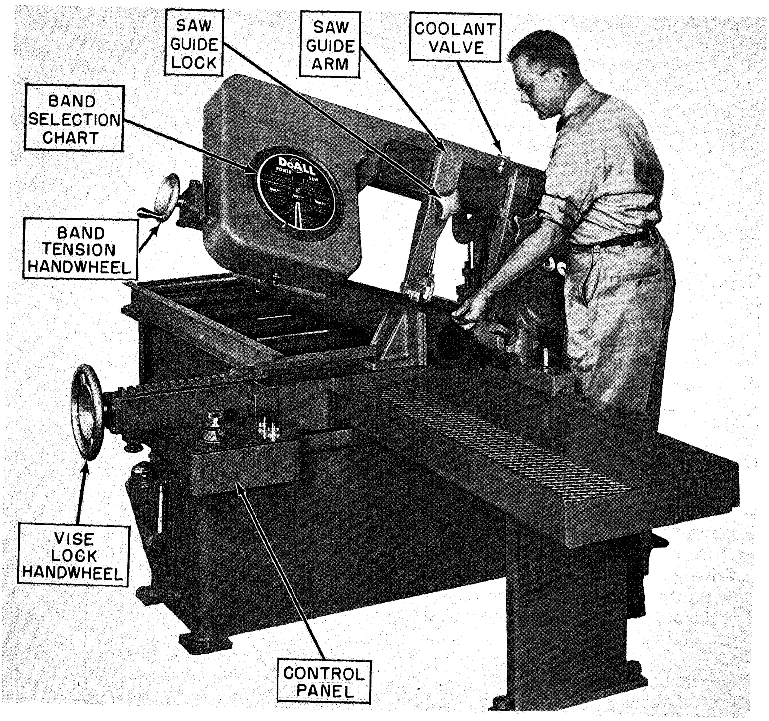
Figure 5-3 illustrates a type of cutoff saw that is now being used throughout the Navy. There are different models of this saw, but the basic design and operating principles remain the same.

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, you 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 make the proper selection. The bands come in two different forms; ready made 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.)

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.



28.297X

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 according to the manufacturer's manual. Do not set the distance between the two guide arms 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.)

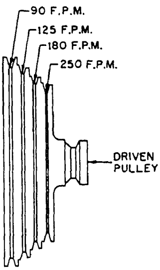
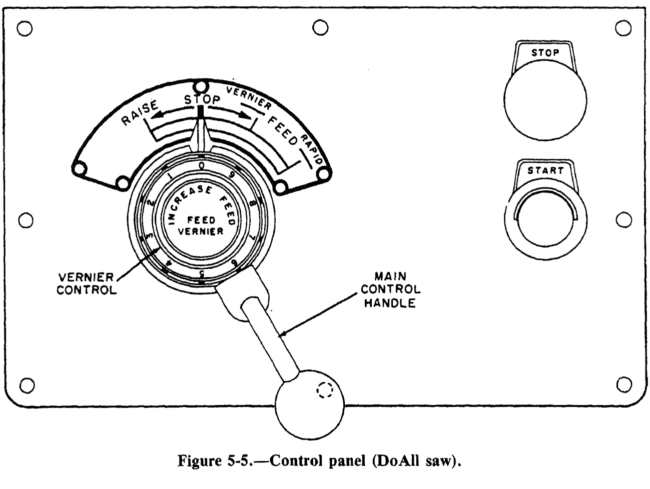


Figure 5-4.—Speed change pulley.

be sawed is held securely in the machine. The movement of the saw head is controlled from the control panel (fig. 5-5). You 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, you can get the maximum cutting efficiency for the type of material being cut. When the cut is complete, the machine will automatically stop. To raise the head above the workpiece for the next cut, 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 rise.

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



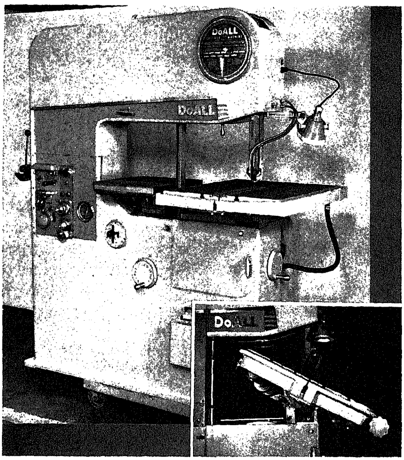
28.296X

Figure 5-5.—Control panel (DoAll saw).

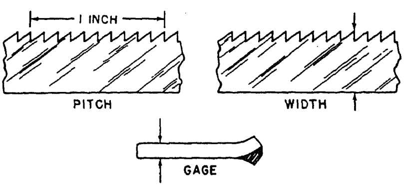
polishing. A bandsaw has a greater degree of flexibility for straight cutting than a power hacksaw in that it can cut objects of any reasonable size and of regular and irregular shapes. A bandsaw also cuts faster than a power hacksaw because the cutting action of the blade is continuous.

Figure 5-6 illustrates a metal cutting bandsaw with a tiltable table. 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.



11.21X
Figure 5-6.—Tiltable (contour) metal-cutting bandsaw.

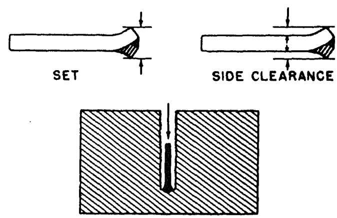


29.15X
Figure 5-7.--Pitch, width, and gauge.

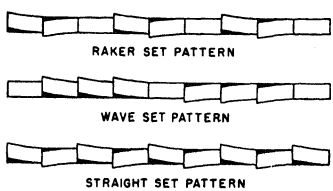
The tiltable table type is convenient for contour cutting because the angle at which work is fed to the blade can be changed readily. This machine usually has special attachments and accessories for precision inside or outside cutting of contours and disks and for mitering and has special bands for filing and polishing work.

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.



28.39X
Figure 5-8.—Set and side clearance.



28.43X
Figure 5-9.--Set pattern.

PITCH: The number of teeth per linear inch.

WIDTH: The distance across the flat face of the band. The width measurement is always expressed in inches, or fractions of an inch.

GAUGE: 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.

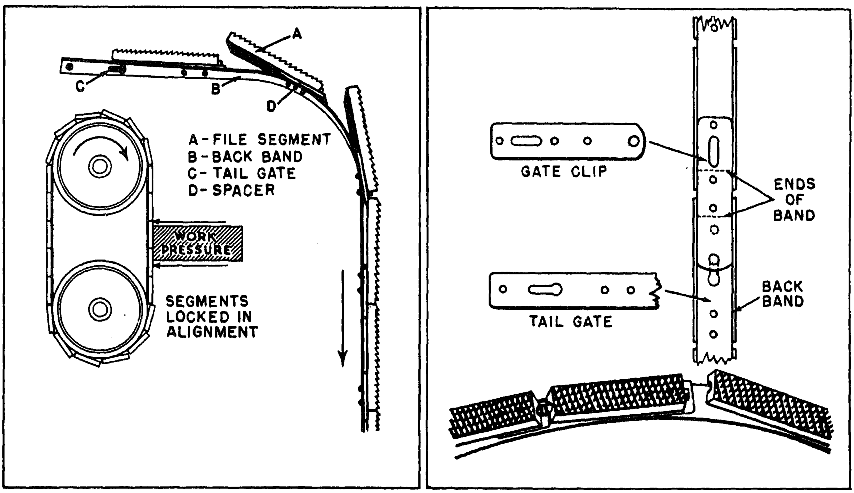
SIDE CLEARANCE: The difference between the dimension of the band back (gauge) and the set of the teeth. Side clearance provides running room for the band back in the kerf or cut. Without side clearance, a band will bind in the kerf.

used for cutting hollow materials, such as pipe 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.

TEMPER: The degree of hardness of the teeth, indicated by the letters **A** and **B**, temper **A** being the harder. Temper **A** bands are used for practically all bandsaw metal cutting work.

File Bands

A file band consists of a long steel strip upon which are mounted a number of file segments that can be flexed around the band wheels and still 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



28.41X

Figure 5-10.—File band flexing principle and construction.

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 on which the file segments are mounted. Do not confuse this term with **BAND BACK,** which refers to a part of a saw band.

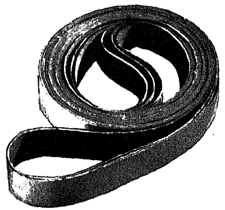
GATE CLIP: A steel strip at the leading end of the back band—a part of an adapter for joining the back band ends to form the file band loop.

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.

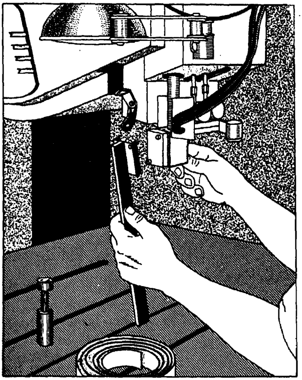
Polishing Bands

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.

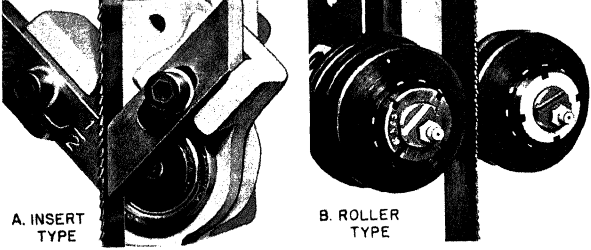


28.43X

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 the sawing operation, place the upper guide so that it will clear the top of the workpiece by 1/8 to 3/8 of an 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

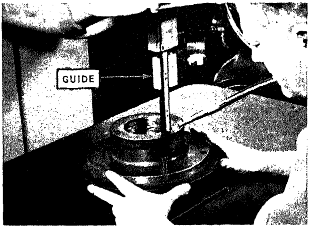


28.44X

Figure 5-13.—Saw band guides.

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 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.



28.45X

Figure 5-14.—File band guide.

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 gauges—0.025, 0.032, and 0.035 inch. The gauge 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 gauges 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 ready made loops of the correct length for some machines, while for others they come in coils of 100 feet or more from which a length must 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 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 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, the speed at which to operate the machine, and the power feed pressure to use to cut various materials.

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 gives you some of that information. Although this table does not cover all types and thicknesses of metals nor recommended feed pressure, it provides a basis on which you can build, using your own experience.

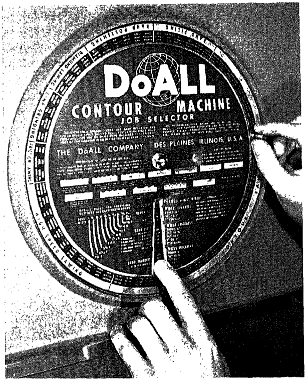
Tooth Pitch

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 tend to cause the teeth to snag and tear loose from the band. For cutting thick material, you should not have too many teeth in contact with the work, because as you increase the number of teeth in contact, you must increase the feed pressure in order to force the teeth into the material.

Excessive feed pressure puts severe strain on the band and the band guides. It also causes the band to wander sideways which results in off-line cutting. Other points to consider in selecting a saw band of proper pitch for a particular cutting job are 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 Gauge

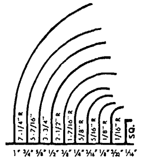
The general rule is to use the widest and thickest saw band that can do the job successfully. For example, you should use a band of maximum width and thickness (if bands of different thickness are available) when the job calls for only straight cuts. On the other hand, when a layout requires radius cuts (curved cuts), the band you select 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



28.46X

Figure 5-15.—Job selector.

WIDTH OF SAW BAND	MINIMUM RADII CUT
1/16"	SQ.
3/[illegible]	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.

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

sharpest curve that can be cut with a particular width saw band. Note that the job selector illustrated in figure 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 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 and maximum band life.

Adjustment of the machine to obtain the proper band velocity cannot be covered in detail here because speed change is done by different methods on 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. However, 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, you can use table 5-2 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

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

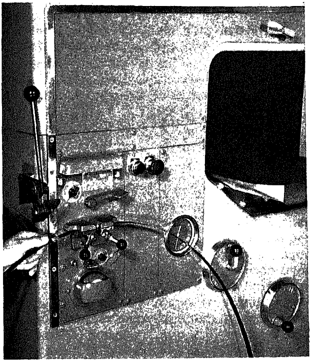
*
L–light, M–medium, H–heavy.

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, multiplying by 2, and adding the circumference of one wheel.



28.48X

Figure 5-17.—Butt welder-grinder unit.

adjust the upper wheel so that it is approximately halfway between the upper and lower limits of its 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 180 degrees and then place the band ends on top of each other. This will provide a set of teeth and a band back edge on both sides of the stacked ends. Ensure that the band back edge and the teeth are in a straight line on both sides. Carefully touch the tips of 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 the adjusting lever 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 the 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 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 welded section in the center of the jaw opening. Close the jaws.

8. The band is ready to be annealed. Push and then quickly release the anneal button 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 of the band until the band fits snugly into the correct slot on the saw band thickness gauge mounted on the welder. 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.

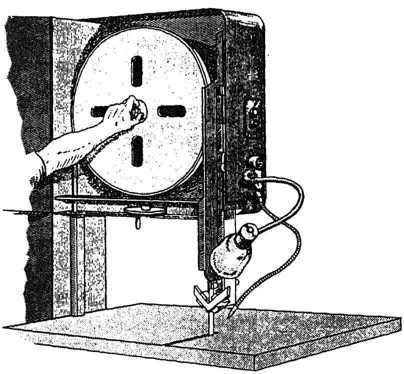
Installing Bands

Insert saw band or tool guides of the correct size for the band you are going to install. Adjust the upper band wheel for a height that will allow you to easily loop the band 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

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 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 you have set the band guide clearance, increase the band tension. The amount of tension to put on the band depends on the width and gauge of the band. A narrow, thin band will not stand as much tension as a wider or thicker band. Too much tension will cause the saw 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 slightly.

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.



28.49X

Figure 5-18.—Upper wheel tilt adjustment.

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 1/8 to 3/8 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

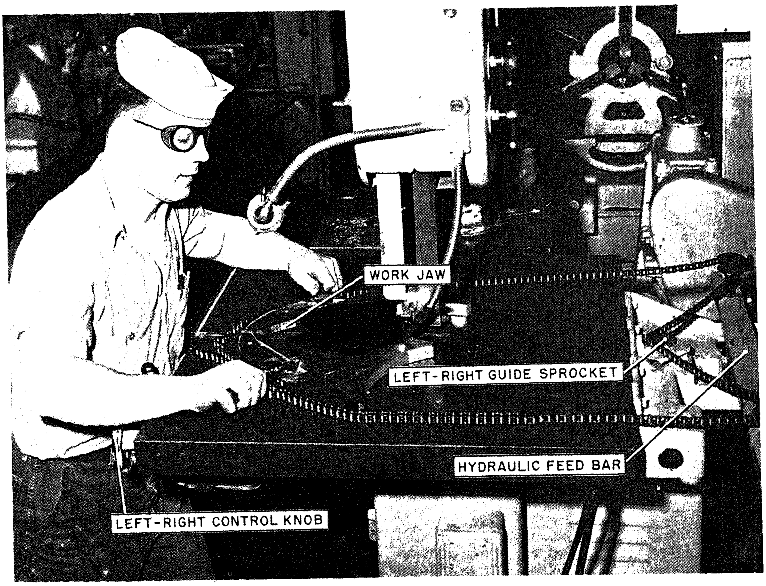
Angular or bevel cuts on flat pieces are made in the same way as straight cuts except that the table is tilted to the desired angle of the cut as shown in figure 5-20.

Contour Cutting

Contour cutting, that is, following straight, angle, and curved layout lines, can be done

28.51X

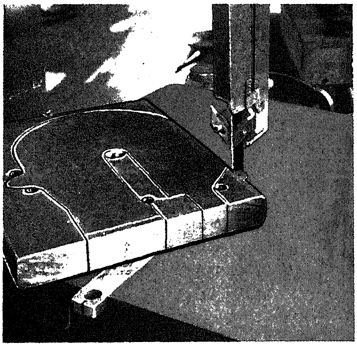
Figure 5-20.—Angular cutting.



28.50X

Figure 5-19.—Work jaw and feed chain adjustment.

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 work table. 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. Figure 5-21 shows the placement of corner holes on a contour cutting layout.



28.52X
Figure 5-21.—Sharp radii cutting eliminated by drilling corner holes.

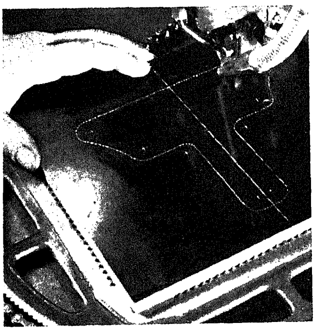
To make an inside cut, drill a starting hole slightly larger in diameter than the width of the band you are going to use. Remove the band from the machine. Shear the band; slip one end through the hole, and then splice the band. When the band has been spliced and reinstalled, the machine is ready for making the inside cut as illustrated in figure 5-22.

Disk Cutting

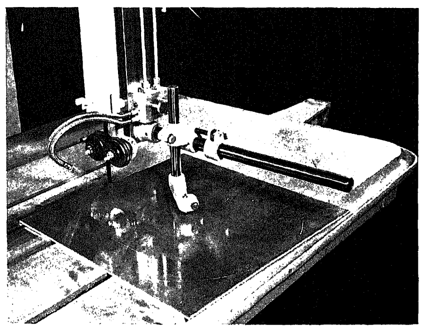
Disk cutting can be done either offhand by laying out the circle on the workpiece and following the layout circle or by using a disk cutting attachment which automatically guides the work so that a perfect circle is cut. Figure 5-23 shows a disk cutting attachment in use. The device consists of a radius 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.

Filing and Polishing

In filing and polish finishing, the work is manually fed and guided to the band. Proper



28.53X
Figure 5-22.—Inside cutting.



28.54X
Figure 5-23.—Disk-cutting attachment.

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.

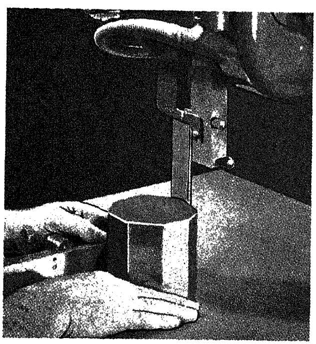
DRILLING MACHINES AND DRILLS

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 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 you operate 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.



28.55X

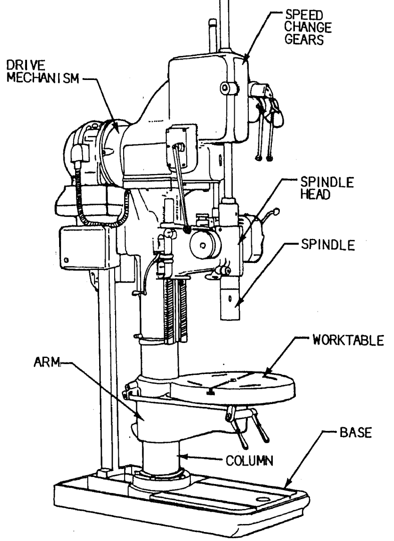
Figure 5-24.—Polish finishing.

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 operating characteristics but differ 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. The basic components of this machine are shown in the illustration.



11.9

Figure 5-25.—General purpose drill press.

The **BASE** has a machined surface with T-slots for heavy or bulky work.

The **COLUMN** supports the work table, the drive mechanism and the spindle head.

The **WORK TABLE** and **ARM** can be swiveled around the column and can be moved up or down to adjust for height. In addition, the work table may be rotated 360° about its own center.

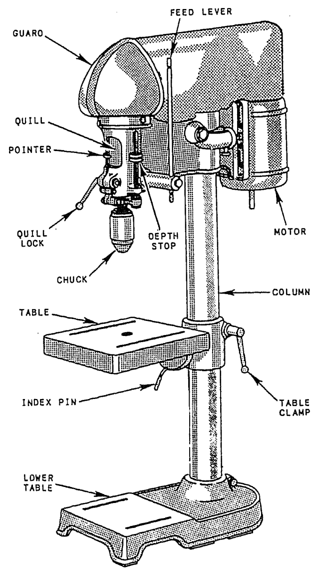
The **SPINDLE HEAD** guides and supports the spindle and can be adjusted vertically to provide maximum support near the spindle socket.

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.

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 work table moves only vertically. The work table 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 only vertically, 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



11.10

Figure 5-26.—Sensitive drill press.

by gearing would be undesirable. Sensitive drill presses are used in drilling holes less than one-half inch in diameter. The 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 on 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 o cutting tool; and (5) coolant systems to pro lubrication and coolant to the cutting tool

On other machines the control levers ma placed in different positions; however, they the same purposes as those shown. In usin locking clamps to lock or "dog down" the or head of a drill after it is positioned ove work, make sure that the locking action doe cause the drill or work to move slightly o position.

TWIST DRILL

The twist drill is the tool generally use drilling holes in metal. This drill is formed e by forging and twisting grooves in a flat str steel or by milling a cylindrical piece of st

In figure 5-28 you see the principal par a twist drill: the **BODY**, the **SHANK**, and **POINT**. The portion of the **LAND** behind **MARGIN** is relieved to provide **BO CLEARANCE**. The body clearance assis reducing friction during drilling. The **LIP** i cutting edge, and on the **CONE** of the drill i

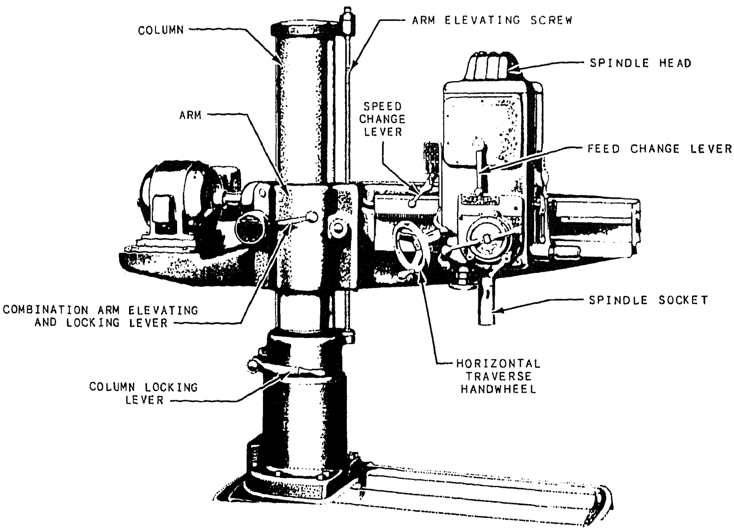
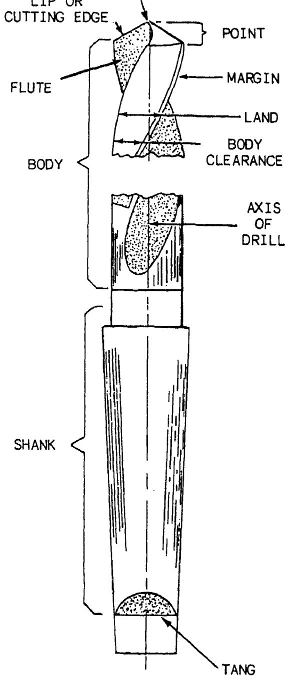


Figure 5-27.—Radial drill press.



44.20

Figure 5-28.—The parts of a twist drill.

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. Do not confuse the point of the drill 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.

The **TANG** is found only on tapered-shank tools. It fits into a slot in the socket or spindle 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 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

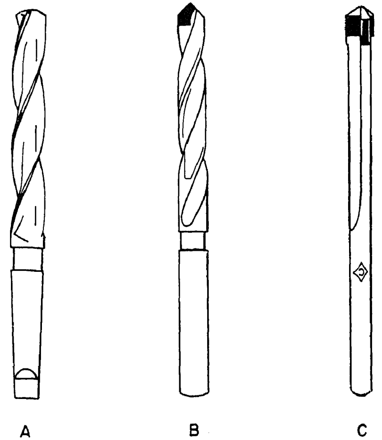


Figure 5-29.—Twist drills: A. Three-fluted core drill; B. Carbide tipped drill with two helical flutes; C. Carbide tipped die drill with two flutes parallel to the drill axis.

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 to 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 process 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 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
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

$$\text{rpm} = \frac{\text{fpm} \times 12}{\pi \times D}$$

$$= \frac{50 \times 12}{3.1416 \times 1}$$

$$= \frac{600}{3.1416}$$

$$= 190$$

where

fpm = required speed in feet per minute

$\pi = 3.1416$

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 your experience 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

Use the lower feed rate given for each range of drill sizes for the harder materials such as tool steel, corrosion-resistant steel and alloy steel. Use the higher feed rate for brass, bronze, aluminum, and other soft metals.

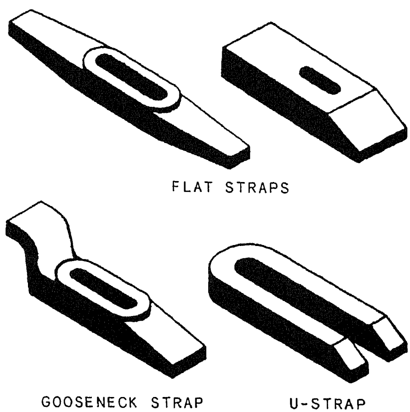
corrosion-resistant steel and certain nonferrous metals such as Monel. For most drilling operations, you can use soluble oil. You may drill aluminum, brass, cast iron, bronze and similarly soft metals dry unless you use a high drilling speed and feed. Use mineral-lard oil for the exceptionally hard metals.

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



11.15

Figure 5-30.—Common types of clamping straps.

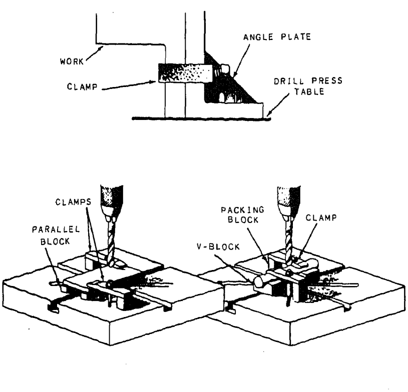
are used to elevate the work above the table so you can better see the progress of the drill. V-blocks are used for supporting round stock, and angle plates are used to support work where a hole is to be drilled at an angle to another surface. Some examples of setups are shown in figure 5-31.

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 off center 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.

1. Before setting up the machine, wipe all foreign matter from the spindle and the 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.



11.16

Figure 5-31.—Work mounted on the table.

3. Bring the spindle with the inserted center down to the center-punch mark and hold it in place lightly while fastening the 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 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 for 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 (lips) chip during drilling, too much lip clearance has been ground into the drill, or you are using too heavy a feed rate.

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, back out the drill frequently to clear the chips from the flutes.

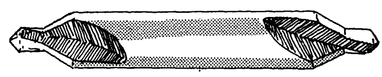


Figure 5-32.—Combined drill and countersink (center drill).

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, you can expect a 1/2-inch drill to produce a hole approximately 0.505 in diameter ([0.004 × 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

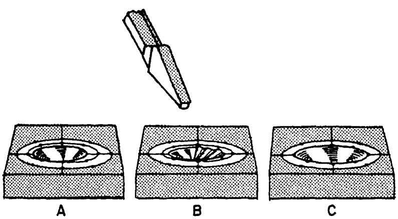
A drill may start off center because of improper center drilling, careless starting of the drill, improper grinding of the drill point, or hard spots in the metal. To correct this condition, take a half-round chisel and cut a groove on the side of the hole toward which the center is to be drawn. (See 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 you use this method to correct an off center condition, 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 you establish the new center point. (Heavy feeds cause a sudden bite in the groove which may result in the work being 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 conterbore 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.



11.17

Figure 5-33.—Using a half-round chisel to guide a drill to the correct center.

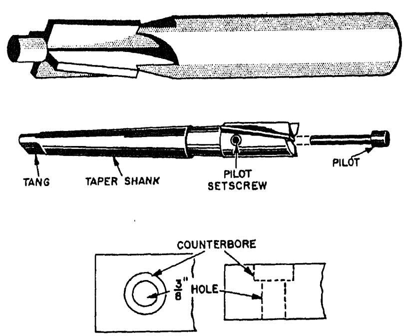


Figure 5-34.—Two types of counterbore

Countersinks are used for seating flathead screws flush with the surface. The basic difference between 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.

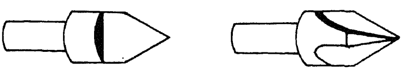
Spotfacing is an operation that cleans up the surface around a hole so that a fastening device can be seated flat on the surface. This operation is commonly required on rough surfaces that have not been machined and on the circumference of concave or convex workpieces. Figure 5-36 shows an example of spotfacing and the application of spotfacing in using fastening devices. This operation is commonly done 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 enlarge a previously used hole for new applications.

Machine reamers have tapered shanks that fit the drilling machine spindle. Be sure not to confuse them with hand reamers, which have straight shanks. Hand reamers will be ruined if they are 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.



28.59

Figure 5-35.—Countersinks.

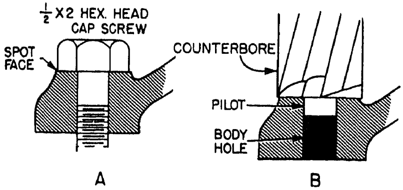
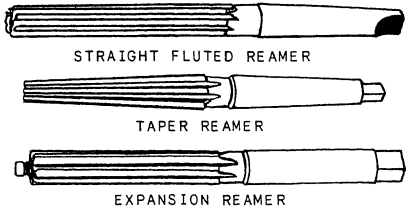


Figure 5-36.—Examples of spotfacing.



5.10

Figure 5-37.—Reamers.

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.

To ream a hole, follow the steps outlined below:

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.
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 downfeed 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, is described briefly in the following paragraphs.

EQUIPMENT

The equipment required to drill angular holes is specialized and is designed to do 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 spindle of the lathe or drill press and prevents it from revolving,

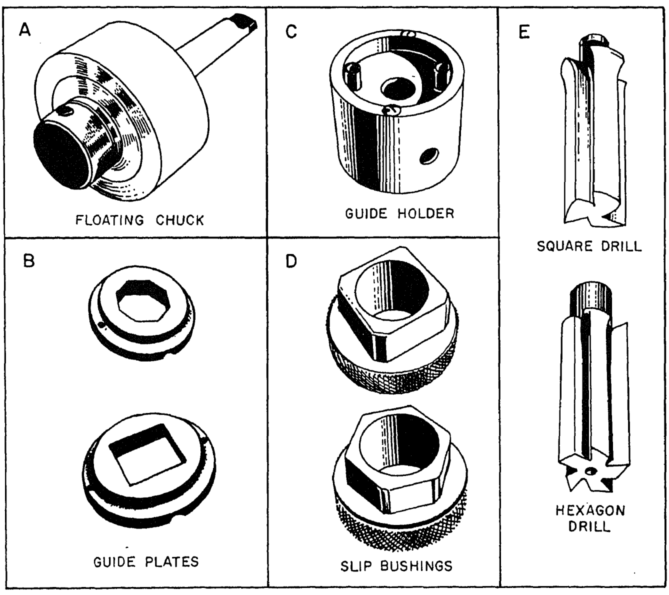


Figure 5-38.—Equipment for drilling angular holes. A. Chuck; B. Guide plate; C. Guide holder; D. Slip bushing; E. Angular drill.

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 you use a lathe and directly to the work when you use 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 drill a hole using the tailstock since it and the chuck will automatically center the pilot hole. In a drill press, you must devise a method to assist you in aligning the pilot hole. A slip bushing will do 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. After you have installed the bushing, 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 they are designed to drill. The drills have straight shanks with flats machined on them to permit securing

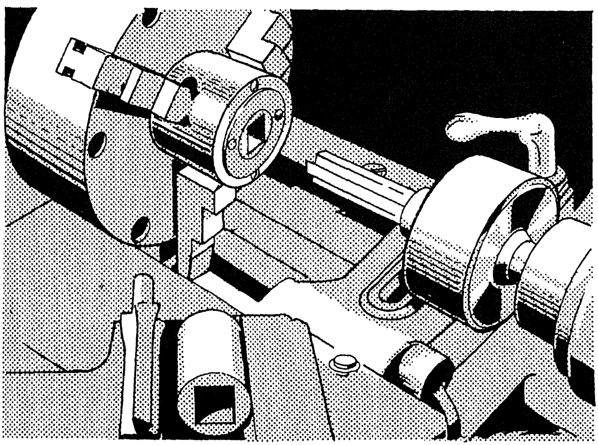


Figure 5-39.—Lathe setup for drilling an angular hole.

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 that for 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 of 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.

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 have drilled the pilot hole, do not move the setup. Any movement will result in misalignment between 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 remove such a piece 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 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.

You can obtain the specific operating procedure 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 you must do. 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 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 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. Use either a machinist's square laid on the table or a dial indicator mounted on the disintegrating head 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

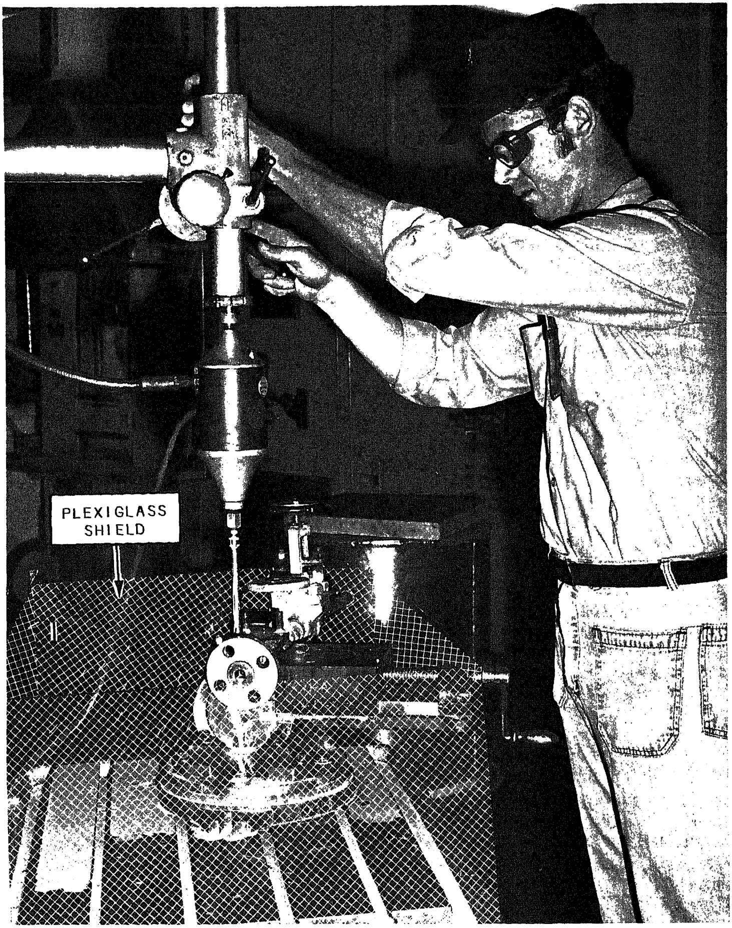


Figure 5-40.—Metal disintegrator removing a broken stud.

spindle, attach the disintegrator's auxiliary ground cable 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. Use a scribe and a small magnet to remove any of the stud material not disintegrated.

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 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.

OFFHAND GRINDING OF TOOLS

One requirement for advancement in the MR rating is to demonstrate the 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 how to grind small tools by 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 and how to sharpen or dress them. You must also 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 composition of the material from which each tool is made and the operations for which the tool is used.

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 are proper grinding techniques. Observance of safety precautions, posted on or near all grinders used by the Navy, is mandatory for your safety and the safety of personnel nearby.

What are some the injuries that result from grinding operations? Eye injuries caused by grit generated during 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 exploding wheel, or a tool "kicked" away from the wheel are other sources of injury. Additionally, prior cuts and abrasions can become infected if they are 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 the 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:

- Secure all loose clothing and remove rings or other jewelry.
- Inspect the grinding wheel, wheel guards, toolrest, and other safety devices to ensure that 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.
- Clean and adjust transparent shields properly, if they are installed. 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 you begin grinding; too much pressure on a cold wheel may cause the wheel to fail.

- 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.

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. Position this shield to deflect the grinding wheel particles away from you.

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.

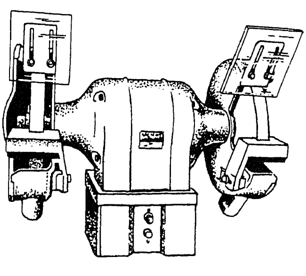


Figure 6-1.—Bench grinder.

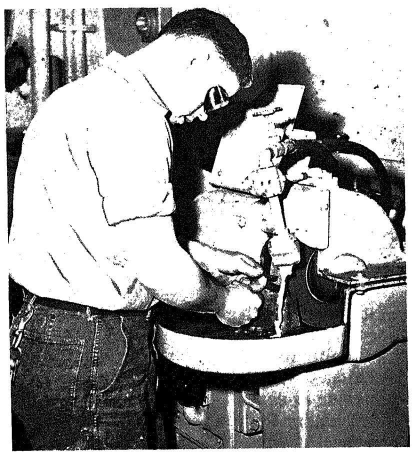
GRINDING WHEELS

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 from the bond, allowing new sharp grains to contact the work.

SIZES AND SHAPES

Grinding wheels come in various sizes and shapes. The size of a grinding wheel is determined



28.61

Figure 6-2.—Grinding on a pedestal grinder.

spindle hole, and the width of its face. All the shapes of grinding wheels are too numerous to list in this manual, but figure 6-3 shows most of the frequently used wheel shapes. The type

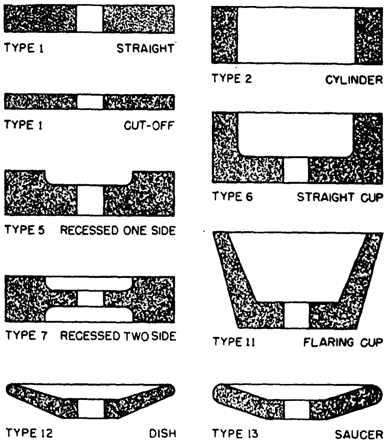


Figure 6-3.—Grinding wheel shapes.

manufacturers. The shapes are shown in cross-sectional views. The specific job will dictate the shape of the wheel to be used.

WHEEL MARKINGS AND COMPOSITION

Grinding wheel markings are composed of six stations. Figure 6-4 illustrates the standard marking. 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

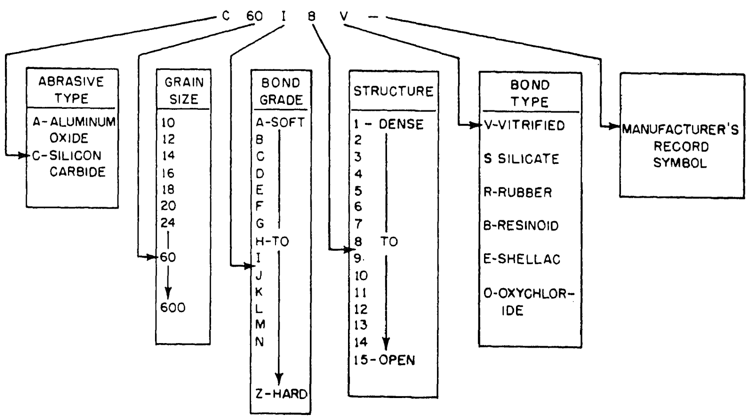


Figure 6-4.—Standard marking system for grinding wheels (except diamond).

work such as cleaning up steel 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 of the grinding wheel marking is the grain size. Grain sizes range from 10 to 500. 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 to be 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. The grading of a grinding wheel from soft to hard grade does not mean that the bond or the abrasive is soft or hard; it means that the wheel has either a small amount of bond (soft grade) or a large amount of bond (hard grade). Figure 6-5 shows magnified portions of both soft grade and hard grade wheels. 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 a hard grade and the wheel with a small amount of bond is a soft grade.

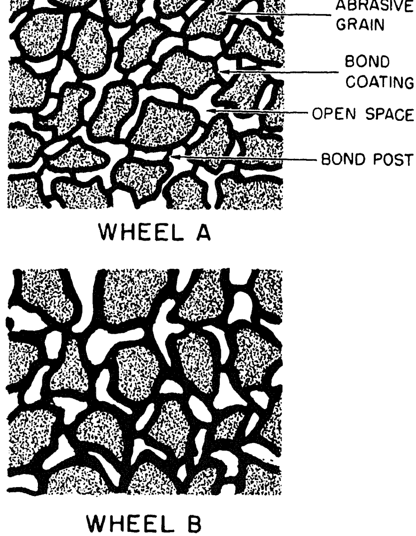


Figure 6-5.—How bond affects the grade of the wheel. Wheel A, softer; wheel B, harder.

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. Wheels with grains that are very closely spaced are said to be dense; when grains are wider apart, the wheels 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 of 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 rubber, resinoid, shellac, and oxychloride.

VITRIFIED BOND.—Designated by the letter V, this is the most common bond used 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 bond 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 bond wheels are designated by the letter S. The bond is made of silicate of soda. Silicate bond wheels are used mainly for large, slow rpm machines where a cooler cutting action is desired. Silicate bond wheels are softer than vitrified wheels; they release the grains more readily than vitrified wheels. Silicate bond wheels are heated to approximately 500 °F when they are made. This type of wheel, like the vitrified bond wheel, must not be run at a speed greater than 6500 surface feet per minute.

RUBBER BOND.—Rubber bond 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 bond wheels are very strong and are elastic. They are used for thin cutoff wheels. Rubber bond wheels produce a high finish and can be run at speeds between 9,500 and 16,000 surface feet per minute.

RESINOID BOND.—Resinoid bond 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 bond wheels, can be run at a speed of 9,500 to 16,000 surface feet per minute.

SHELLAC BOND.—Shellac bond 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

bond wheels give a high finish and have a cool cutting action when used as cutoff wheels. Shellac bond wheels can be run at speeds between 9,500 and 12,500 surface feet per minute.

OXYCHLORIDE BOND.—Oxychloride bond 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 used in grinding wheels but is used extensively to hold abrasives on sanding disks. Oxychloride bond wheels can be run at speeds between 5,000 and 6,500 surface feet per minute.

Manufacturer's Record Symbol

The sixth station of 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

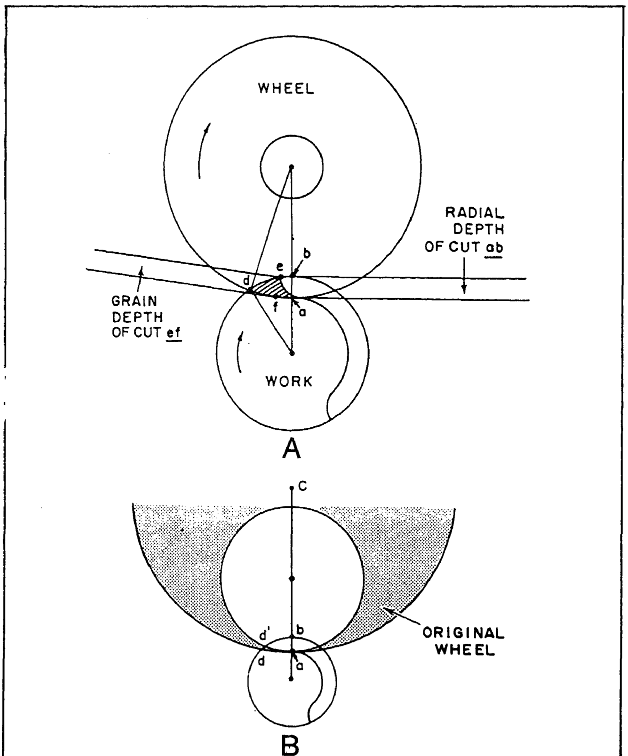
Diamond grinding wheels are classed by themselves. Wheels of this type are very expensive and should be used with care and only for grinding carbide cutting tools. Diamond wheels can be made from natural or manufactured diamonds. They are marked similarly to aluminum-oxide and silicon-carbide wheels, 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, or 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 the diamond section. This is the thickness of the abrasive layer and ranges from 1/32 to 1/4 inch. Cutting speeds range from 4,500 to 6,000 surface feet per minute.

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 impractical and unnecessary to keep on hand a wheel for 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 grain begins cutting into the work, increasing its depth of cut until it reaches a maximum depth at some point along the arc of contact. This greatest depth is called the grain depth of cut.

To understand what part grain depth of cut plays in grinding, look at figure 6-6. Part A 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 a to d in a given amount of time. During the same time, a point on the workpiece rotates



wheel B. During this time the grain will remove an amount of material represented by the shaded area ade. Now refer to part B and assume that the wheel has worn down to a much smaller size, while the wheel and work speeds remain unchanged. The arc of contact ad′ of the smaller wheel is shorter than the arc of contact ad of the original (larger) wheel. Since the width of the grains remains the same, decreasing the length of the arc of contact will decrease the surface (area = length × width) that a grain on the smaller wheel covers in the same time as a grain on the larger wheel. If the depth that each grain cuts into the workpiece remains the same, the grain on the smaller wheel will remove a smaller volume (volume = length × width × depth) of material in the same time as the grain on the larger wheel. However, for both grains to provide the same cutting action, they both have to remove the same volume of material in the same length of time. To make the volume of material the grain on the smaller wheel removes equal that of the grain on the larger wheel, you have to either make the grain on the smaller wheel cut deeper into the workpiece or cover a larger workpiece surface area at its original depth of cut.

To make the grain cut deeper, you must increase the feed pressure on the grain. This increase of feed pressure will cause the grain to be torn from the wheel sooner, making the wheel act like a softer wheel. Thus, the grain depth of cut theory says that as a grinding wheel gets smaller, it will cut like a softer wheel because of the increase in feed pressure required to maintain its cutting action.

The opposite is true if the wheel diameter increases. For example, if you replace a wheel that is too small with a larger wheel, you must decrease feed pressure to maintain the same cutting action.

The other previously mentioned way to make a grain on a smaller wheel remove the same amount of material as a grain on a larger wheel is to keep the depth of cut the same (no increase in feed pressure) while you increase the surface area the grain contacts. Increasing the surface area requires lengthening the contact area, since the width remains the same. To lengthen the contact area, you can either speed up the workpiece rotation or slow down the wheel rotation. Either of these actions will cause a longer surface strip of the workpiece to come in contact with the grain on the wheel, thereby increasing the volume of material removed.

you increase the wheel diameter. To keep from removing a larger volume of material, you must decrease the surface of the workpiece with which the grain comes into contact. You can do this by either slowing down the workpiece rotation or speeding up the wheel rotation.

Keep in mind that all of these actions are based on the grain depth of cut theory. That is, making adjustments to the grinding procedure to make one wheel cut like another. The following summary shows the actions you can take to make a wheel act a certain way.

MAKE THE WHEEL ACT SOFTER (INCREASE THE GRAIN DEPTH OF CUT)

- Increase the work speed
- Decrease the wheel speed
- Reduce the diameter of the wheel and increase feed pressure

MAKE THE WHEEL ACT HARDER (DECREASE THE GRAIN DEPTH OF CUT)

- Decrease the work speed
- Increase the wheel speed
- Increase the diameter of the wheel and decrease feed pressure

GRINDING WHEEL SELECTION AND USE

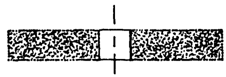
The selection of grinding wheels for precision grinding is based on such factors as the physical properties of the material to be ground, the amount of stock to be removed (depth of cut), the wheel speed and work speed, and the finish required. The selection of a grinding wheel that has the proper abrasive, grain, grade, and bond is determined by one or more of these factors.

An aluminum oxide abrasive is the most suitable for grinding carbon and alloy steel, high-speed steel, cast alloys and malleable iron. A silicon carbide abrasive is the most suitable for grinding nonferrous metals, nonmetallic materials, and cemented carbides.

Generally, as you grind softer and more ductile materials, you should select coarser grain wheels. Also, if you need to remove a large amount of material, use a coarse grain wheel (except on very hard materials). If a good finish is required, use a fine grain wheel. If the machine you are using is worn, use may need to use a harder grade to help offset the effects of wear on the machine. Using a coolant also permits you to use 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 outside diameter, the thickness, and the spindle hole size are the three dimensions that you must check. If necessary, use an adapter (bushing) to decrease the size of the spindle hole, so that it fits your grinder.

The wheels recommended for grinding and sharpening single point (lathe, planer, shaper, and so on) 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



STRAIGHT WHEEL

Figure 6-7.—Grinding wheel for bench and pedestal grinders.

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

and possibly injure someone nearby.

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 its side. Rotate the wheel 1/4 of a turn (90°) and repeat the test. A good wheel gives out a clear ringing sound when tapped. If the tapping produces a dull thud, the wheel is cracked and should not be used.

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

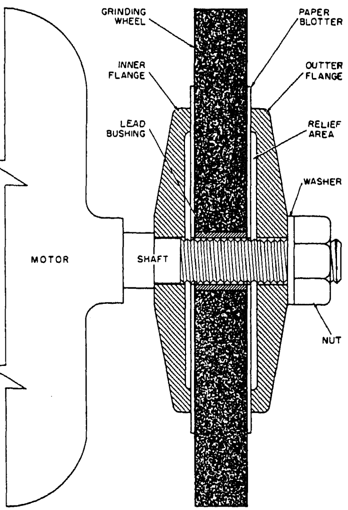


Figure 6-8.—Method of mounting a grinding wheel.

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 the shaft when the grinder is operating.

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 ONTO THE SHAFT**. Forcing the wheel onto the shaft may cause the wheel either to be slightly out of axial alignment or to crack when it is used.

The next item to install is another blotter, followed by 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 their cutting face. This cleaning breaks away dull abrasive grains and smoothes 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

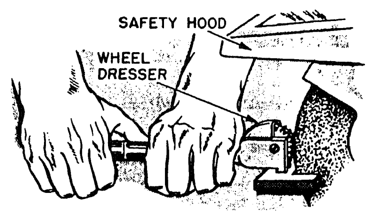


Figure 6-9.—Using a grinding wheel dresser.

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.

CARBIDE TOOL GRINDER

The carbide tool grinder (fig. 6-10) looks much like a pedestal grinder with the toolrest on the side 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.

Unlike the pedestal grinder where the grinding is done on the periphery of the wheel, the carbide tool bit grinder has the grinding done 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 have tips that are (1) disposable, having three or more pre-ground cutting edges or (2) brazed, having cutting edges that must be ground. 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.

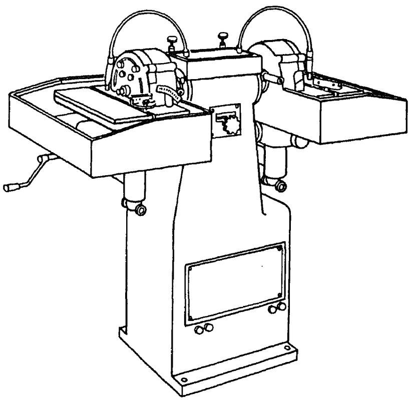


Figure 6-10.—Carbide tool grinder.

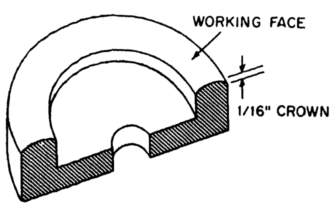


Figure 6-11.—Crown on the working face of a wheel for a carbide tool bit grinder.

You can obtain the best results from carbide tipped tools by using four different grinding wheels to sharpen them. Use the aluminum oxide wheel recommended for grinding high-speed steel tools 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. When you are ready to grind the carbide tip, use wheels that have silicon carbide as the abrasive material. Use a C6018V wheel for rough grinding and a C100H8V wheel for semifinish grinding. To finish grind the tip, use a diamond impregnated grinding wheel with the designation SD 220-P50V.

OPERATION OF THE CARBIDE TOOL GRINDER

Use the following procedure to sharpen a carbide tipped tool bit.

- 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.

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.

If you use silicon carbide wheels, grind the carbide tip dry. If you use diamond wheels, be sure to use coolant on both the tool and the 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

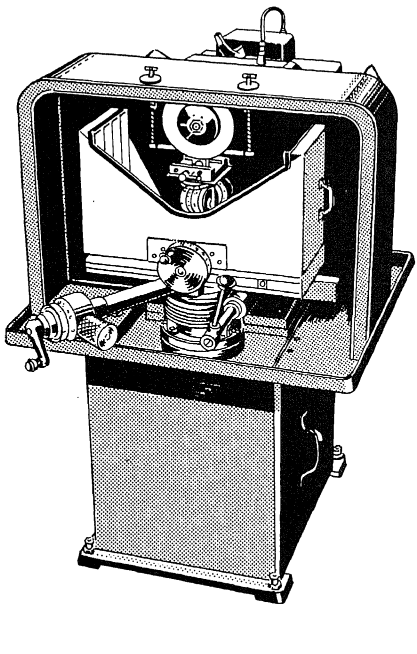


Figure 6-12.—Chip breaker grinder.

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 wheel but differs from other type 1 wheels in that it is normally less than 1/4 inch thick. An SD150R100B grinding wheel is normally recommended.

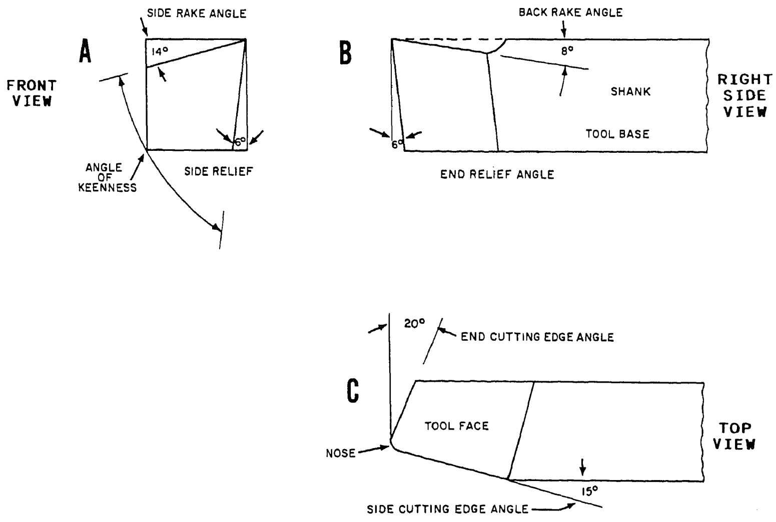
Chip breaker grinders have a coolant system that either floods or slowly drips coolant onto the tool being ground. The main objective in using coolant is to prevent the grinding wheel from loading up or glazing over from the grinding operation.

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 cutting a rake angle and a relief angle.



making a slope either away from or toward the side cutting edge. Figure 6-13A shows a positive side rake angle. When the side rake is cut toward the side cutting edge, the side rake has a negative angle. 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. 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 softer 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.

BACK RAKE.—The back rake is the angle at which the top surface of the tool is ground away mainly to guide the direction of the flowing chips. 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 (fig. 6-13B) 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 of the tool is ground to prevent the tool bit from rubbing into the work. The side relief angle, like the side rake angle, influences 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 grinding the side rake and side relief and is the sum of the side rake and side relief angles. 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 vary 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 shafts, 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 urning 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 good finish. 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. A radius (rounded end) of from 1/64 to 1/32 inch is normally used for turning operations.

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 tools can be ground with the diamond wheel on the chip breaker grinder. High-speed 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 and 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. Several different materials 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.

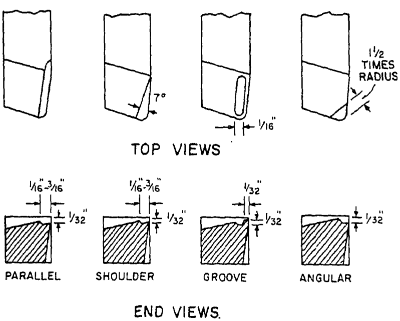


Figure 6-14.—Chip breakers.

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. Tools 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 that of 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 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 are similar in their ability to resist abrasive wear and to 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 made.

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 high-speed steel, retaining their hardness up to an operating temperature of approximately 1400 °F. This characteristic allows cutting speeds approximately 60% greater than for high-speed steel tools. 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 as Stellite, Rexalloy, and Tantung.

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 that cannot be readily machined with a standard of tools required in machinery, such as turning, facing, threading, and grooving are available with different grades of carbide tips already brazed onto steel shanks. Small carbide blanks are also available that you can braze onto a shank.

Brazing on a carbide tip is a relatively simple operation that can be performed by anyone qualified to operate an oxyacetylene torch. To braze on a carbide tip, first, thoroughly clean the steel shank by grinding or sandblasting and degreasing it with an approved solvent. Next, completely coat the steel shank and the carbide tip with a flux to further remove any contamination and to prevent oxidation during brazing. A thin shim-like brazing alloy is available that you can cut to the size needed and place 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. Begin heating the tool at 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 bond from holding the tip in place. After the tool is cooled, grind it 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. 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 of 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. You may also use these same types of chip breakers on high-speed steel cutters.

Mechanically Held Tip (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 of 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 set-up 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. Methods 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.

CERAMIC

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.

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

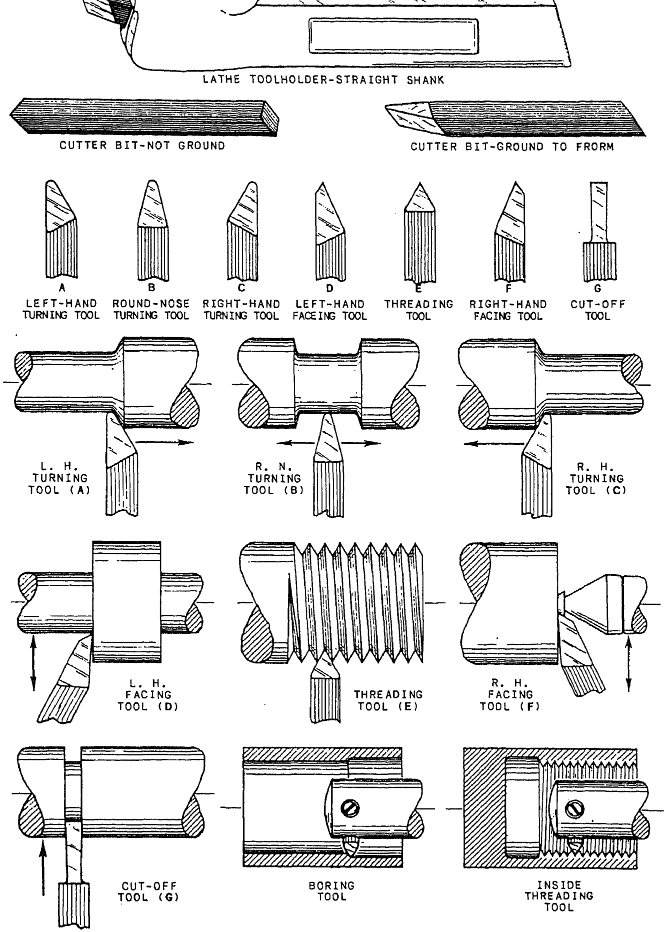


Figure 6-15.—Lathe tools and their application.

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 san shape as the left-hand turning tool so that tl cutting edge is on the front side of the cutter b 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 affect 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 sit in the various holders. You must consider these angles with respect to the angles ground in the tools and the angle that you set the toolholder 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 uses a left-hand turning tool and a left-hand toolholder uses a right-hand turning tool. Study figure 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 machinist usually forms the contour as he or she 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 you grind tool bits. Finish the cutting edge by honing it 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 the cutting edge is ground on the periphery of a wheel less than 6 inches in diameter, it will be undercut and will not have the correct angle.) Keep the tool cool while grinding.
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

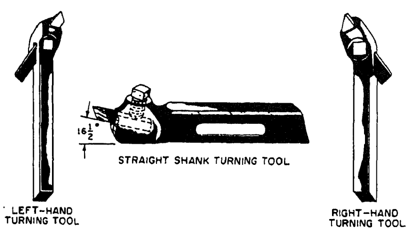


Figure 6-16.—Standard lathe toolholders.

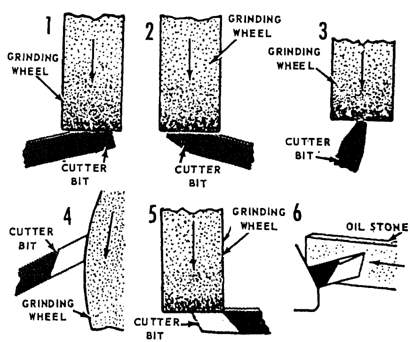


Figure 6-17.—Grinding and honing a lathe cutter bit.

preferable, as a large radius may cause chatter. Hold the tool lightly against the wheel and turn it 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. The more metal you leave on the tool, the better the tool will absorb 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 oil-stone when honing. Honing will not only improve the cutting quality of the tool, but 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 nose of the tool. The only portion of a tool ground in this manner that will be cutting is the nose. Grinding a larger back rake angle makes a more acute, chisel-type nose. Decreasing side rake and side clearance provides more support for the cutting edge. By increasing the radius of the nose, you ensure that more of the cutting edge will be 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 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 for roughing and finishing cuts for tools made 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

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

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 larger angles are 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 tools. 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 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, side rake, 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

Table 6-3.—Angles for Grinding Turrent 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

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 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 the carbide-tipped types.

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 has 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 their 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.

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.

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 relief and end relief of the tool must be ground to give a projecting cutting edge. 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 will tend to break during the cut. The front and side clearance angles seldom exceed 3° to 5°.

In addition to having relief angles, the tool bit must slope away from the cutting edge. This slope is known 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 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.

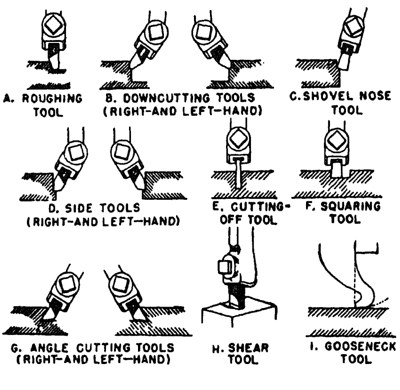


Figure 6-18.—Standard shaper and planer tools.

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.

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 eight 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.

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, NAVEDTRA 10085 (series), contains detailed descriptions of the off-hand grinding of twist drills and handtools. Therefore, these subjects are not discussed here. You should study NAVEDTRA 10085 (series) 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 by mishandling and improper storage. Whenever possible, do not 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 cut-off 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.

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: (1) either the diameter of work it will swing over the bed or the diameter of work it will swing over the cross-slide and (2) either the length of the bed or the maximum distance between centers. For example, a 14-inch × 6-foot lathe has a bed that is 6 feet long and will swing work (over the bed) up to 14 inches in diameter.

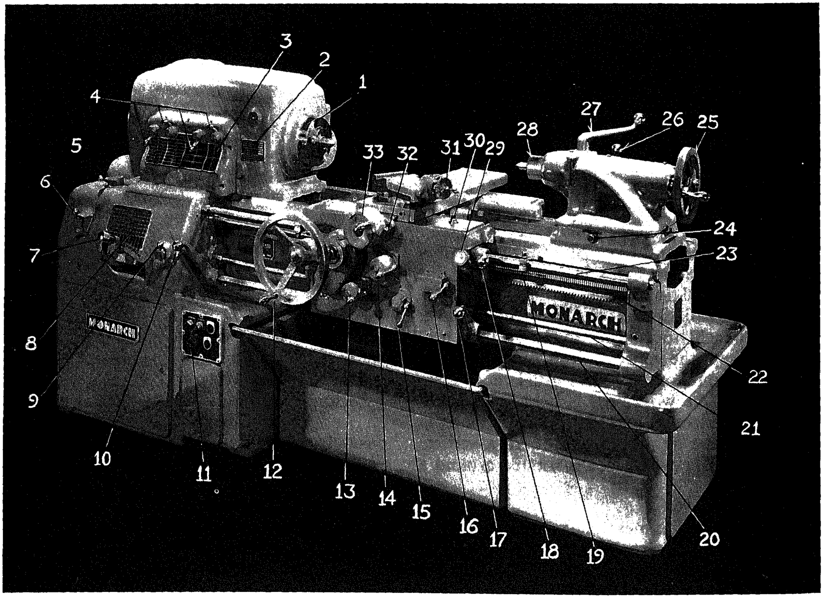
Engine lathes range in size from small bench lathes with a swing of 9 inches to very large lathes for turning work of large diameters, such as low-pressure turbine rotors. A 16-inch swing lathe is a good, average size for general purposes and is usually the size installed in ships that have only one 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 to the end at the operator's left.

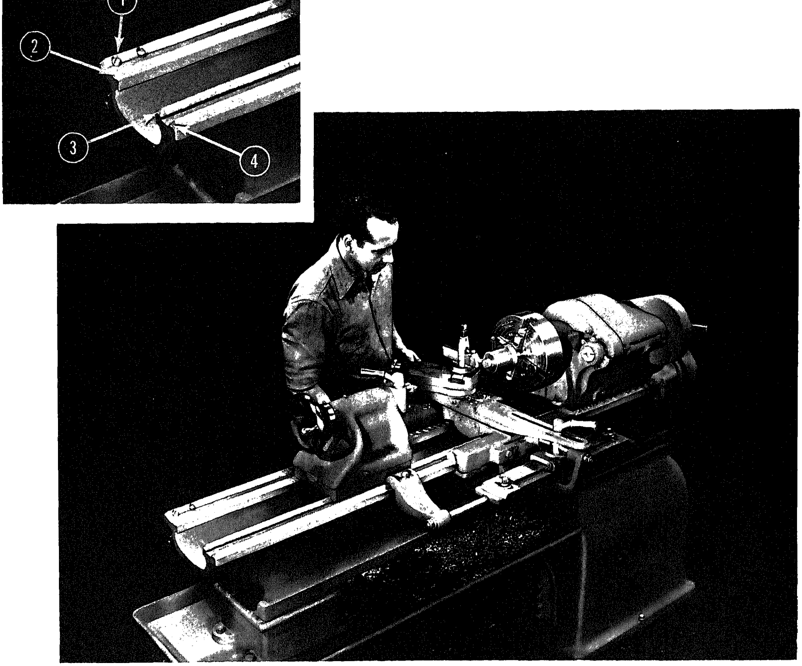
Figure 7-2 shows the ways of a typical lathe. The inset shows the inverted V-shaped ways (1, 3, and 4) and the flat way (2). The ways are accurately machined parallel to the axis of the spindle and to each other. The V-ways are guides that allow the carriage and tailstock to move over them only in their 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 way



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.



28.70X

Figure 7-2.—Rear view of lathe.

number 3, keep the carriage aligned with the headstock and the 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.

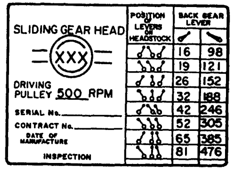
For a lathe to perform satisfactorily, the ways must be kept in good condition. A common fault of careless machinists is to use the bed as an anvil for driving arbors or as a shelf for hammers, wrenches, and chucks. Never allow anything to strike a hard blow on the ways or damage their finished surfaces in any way. Keep them clean and free of chips. Wipe them off 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, shows the lever positions for the different spindle speeds. Figure 7-4 shows this plate for the geared headstock in figure 7-3. Always stop the lathe when you shift gears to avoid damaging the 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 and intermediate shafts 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 high and low speed positions for each combination of the other gears (figure 7-4).

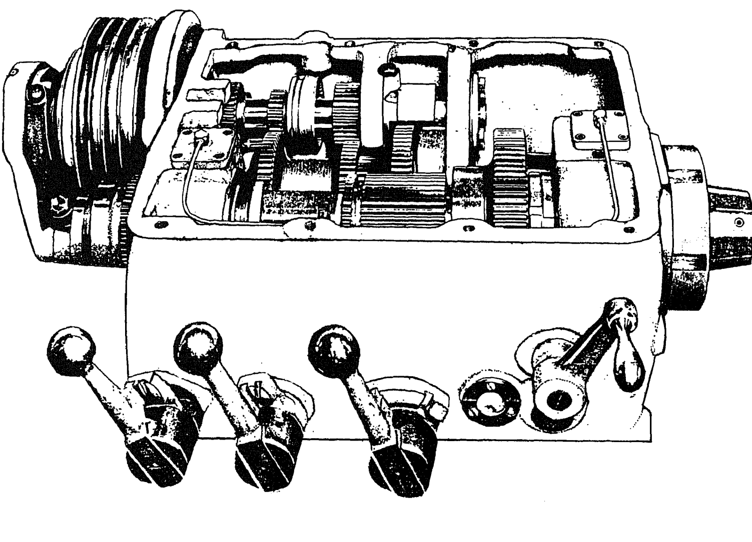


28.73

Figure 7-4.—Speed index plate.

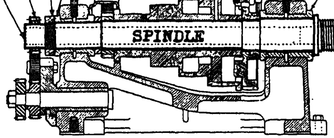
The headstock casing is filled with oil to lubricate the gears and the shifting mechanism it contains. Parts not immersed in the oil are lubricated by either the splash produced by the revolving gears or by an oil pump. Be sure to keep the oil to the oil level indicated on the oil gauge, and drain and 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



28.72

Figure 7-3.—Sliding gear type headstock.



28.74X

Figure 7-5.—Cross section of a belt-driven headstock.

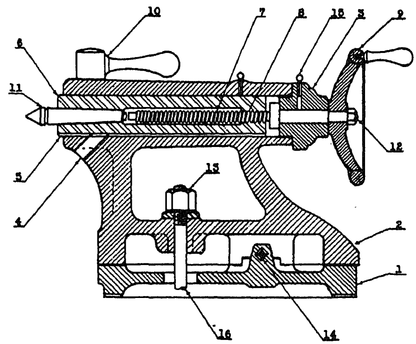
of the headstock through which it projects. The section of the spindle between the bearings carries the pulleys or gears that turn the spindle. The nose of the spindle holds the driving plate, the faceplate, or a chuck. The spindle is hollow throughout its length so that bars or rods can be passed through 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

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.

The spindle is subjected to considerable torque because it both drives the work against the resistance of the cutting tool and 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** or **LIVE** 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.



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.

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.

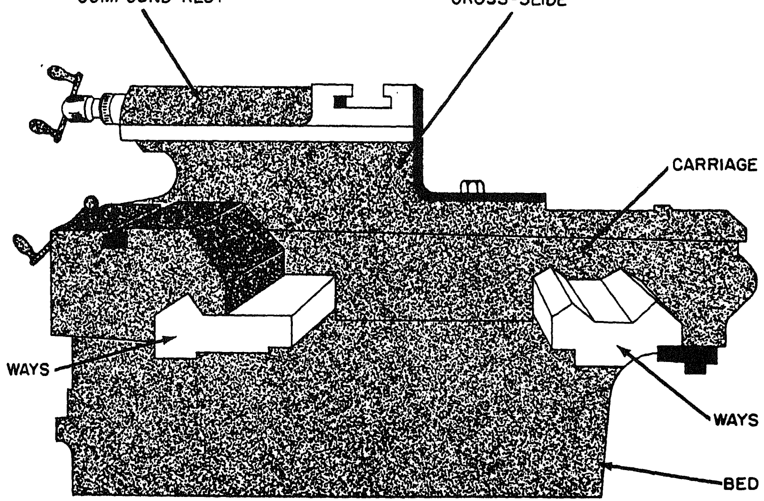
The tailstock body is made in two parts. The bottom, or base (1), is fitted to the ways; the top (2) can move laterally on its base. The lateral movement can be closely adjusted by setscrews. Zero marks inscribed on the base and top indicate the center position and provide a way to measure setover for taper turning. 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 i in the spindle so that the tool will not revolve. I the drill or reamer is allowed to revolve, it wil score the tapered hole and destroy its accuracy The spindle of the tailstock is engraved witl graduations which help in determining the deptl of a cut when you drill or ream.

CARRIAGE

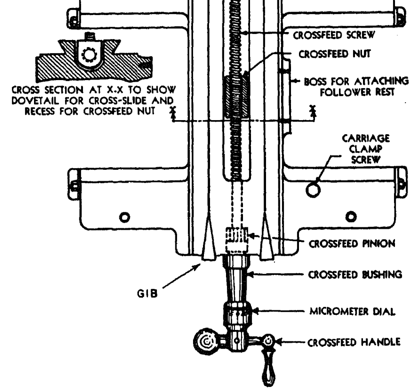
The carriage carries the crossfeed slide and th compound rest which in turn carries the cuttin tool in the toolpost. The carriage slides on th ways along the bed (fig. 7-7).

Figure 7-8 shows a top view of the carriage The wings of the H-shaped saddle contain th bearing surfaces which are fitted to the V-way of the bed. The crosspiece is machined to forn a dovetail for the crossfeed slide. The crossfee slide is closely fitted to the dovetail and has tapered gib which fits between the carriag dovetail and the matching dovetail of th crossfeed slide. The gib permits small adjustment to remove any looseness between the two parts The slide is securely bolted to the crossfeed nu



28.7

Figure 7-7.—Side view of a carriage mounted on the bed.



28.77X

Figure 7-8.—Carriage (top view).

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 to 1 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

In general, a lathe apron contains the following mechanical parts:

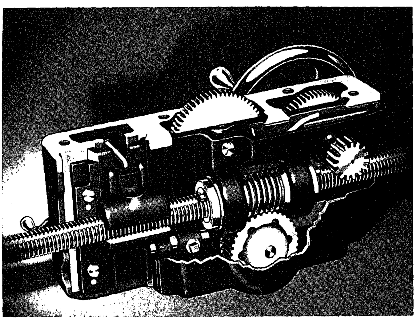
1. A longitudinal feed **HANDWHEEL** for moving the carriage by hand along the bed. This handwheel turns a pinion that meshes with a rack gear secured to the lathe bed.
2. **GEAR TRAINS** driven by the feed rod. These gear trains transmit power from the feed rod to move the carriage along the ways and 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 the 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 the 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 cross feed. In others, both feeds are driven from the same driving gear on the feed rod through a common clutch, with one feed at a time connected to the drive by a selector lever. 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 cross feed 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 spline in the lead screw to serve the 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.



28.79X

Figure 7-9.—Rear view of a lathe apron.

LEAD SCREW

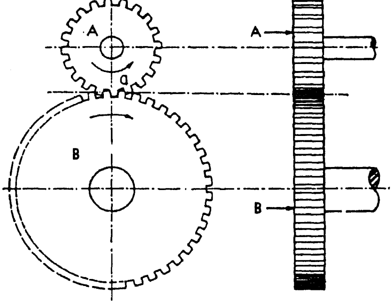
The lead screw is used for thread cutting. Along its length are accurately cut Acme threads which engage the threads of the half-nuts in the apron when half-nuts are clamped over it. When the lead screw turns in the closed half-nuts, the carriage moves along the ways a distance equal to the lead of the 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 gear 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



28.81X

Figure 7-10.—A simple gear arrangement.

inversely proportional to their size. The 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

$$\frac{\text{rpm of lead screw}}{\text{rpm of spindle}} = \frac{\text{number of teeth on spindle gear A}}{\text{number of teeth on screw gear B}}$$

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 desired ratio.

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.

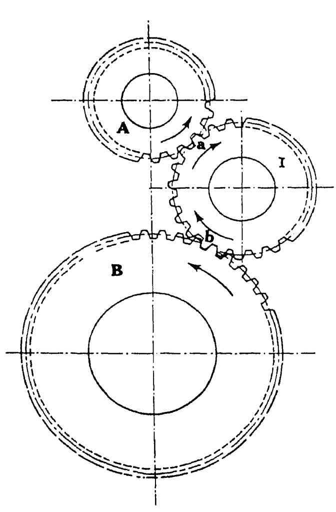
Suppose you want to cut 8 threads per inch on a lathe that has a lead screw with 6 threads per inch. The carriage must carry the thread-cutting tool 1 inch along the work while the work makes eight complete revolutions. Since the lead screw has 6 threads per inch, it must revolve six times in the half-nuts 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, 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 vise 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 B.



28.82X

Figure 7-11.—Idler gear inserted between a driving gear and a driven gear.

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 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, you must 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.

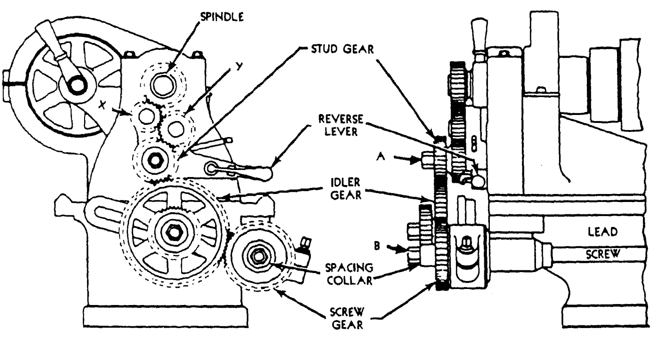
A simple rule to follow in determining what stud and screw gears to use is: Multiply the desired number of threads per inch and the number of threads per inch in the lead screw by the same number; if the products correspond to the number of teeth in any two of the change gears at hand, use those gears; 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 that you do have.

If you cannot get the proper ratio of gears with the change gears you have at hand or if the gears would be too small or too large to connect properly or conveniently (as would be the case if



28.83X

Figure 7-12.—Simple gearing on a lathe.

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 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.

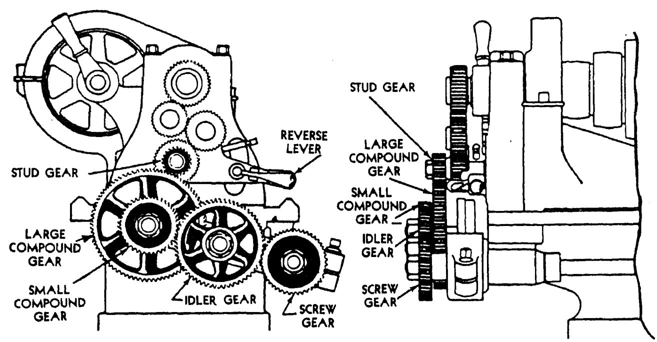
The advantage of compounding is best demonstrated by the following example:

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 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 to the gear train by moving 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.



28.84X

Figure 7-13.—Compound gearing on a lathe.

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.

Twenty-four different gear ratios are provided by the quick-change gear box shown in figure 7-15. The lower lever has eight positions, 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.

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 G 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). Adjust 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 the 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 onto the lead screw.

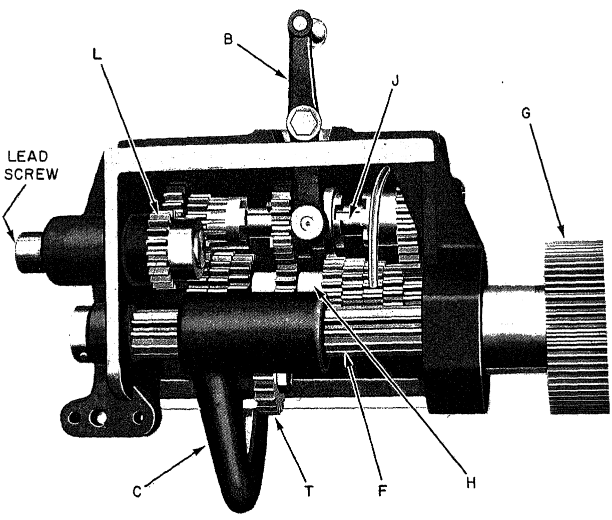
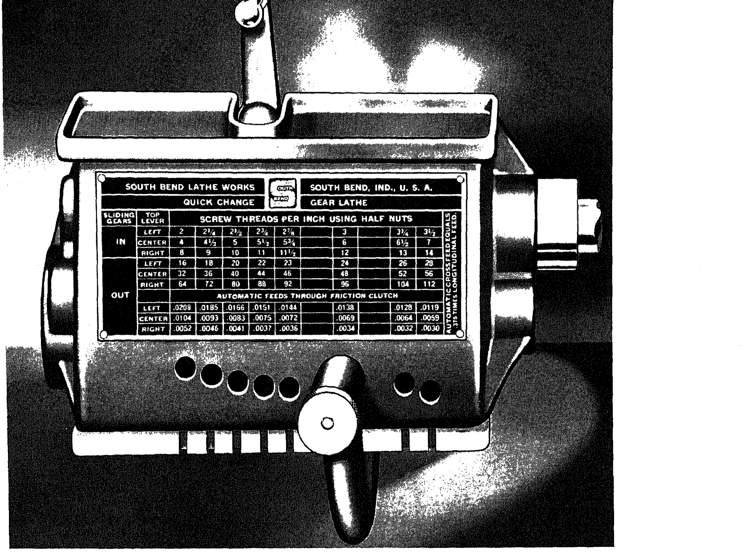
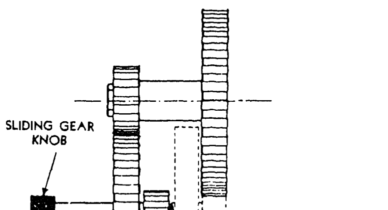


Figure 7-14.—Quick-change gear box (rear view).



28.87X

Figure 7-15.—Quick-change gear box.

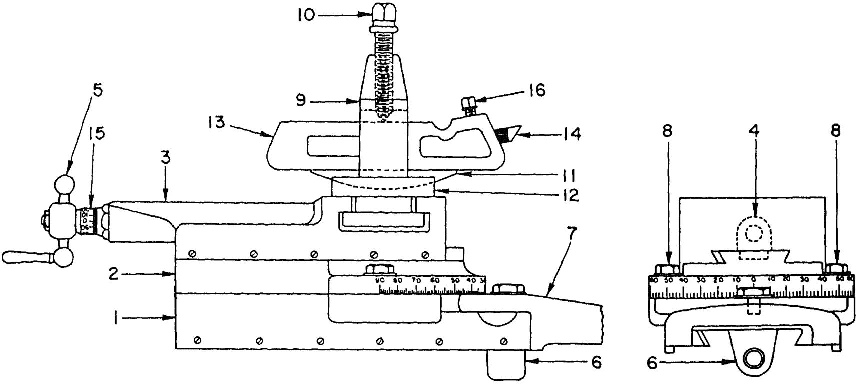


28.86X

Figure 7-16.—Showing how the gear ratio is changed by sliding gear.

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 finest longitudinal feed is 0.0030 inch per revolution of spindle, the next finest is 0.0032 inch, and so on. 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.

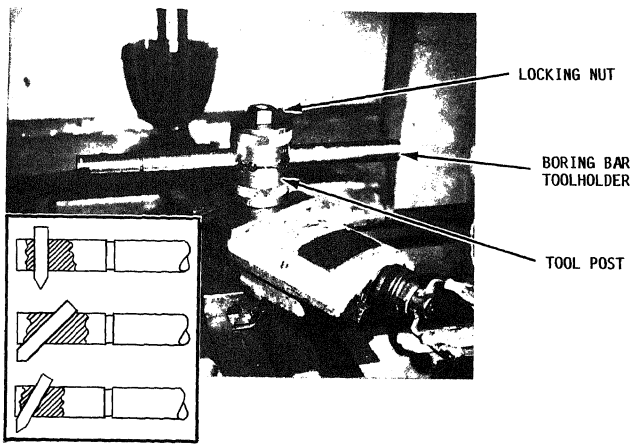
On a lathe with a separate feed rod, a feed-thread shifting lever located at the gear box (part 9 in fig. 7-1) connects the drive to the feed rod or the lead screw as desired. When the feed rod is engaged, the lead screw is disengaged and vice versa.



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.
16. Toolholder setscrew.

28.88X

Figure 7-17.—Compound rest.



28.299

Figure 7-18.—Castle type toolpost and toolholder.

the lead screw to the spindle gear train that provides the correct conversion ratio. You can find information on this in handbooks for machinists, in the equipment technical manual, and through 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 slide. 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 cross feed described previously (fig. 7-8). 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 a right angle 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

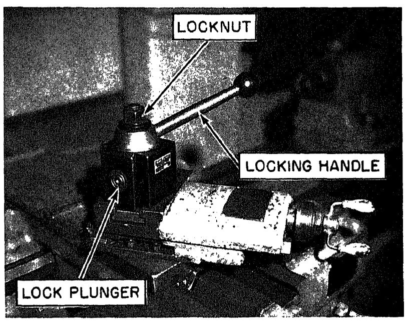
Accessories are the tools and equipment used in routine lathe machining operations. Attachments are special fixtures which may be secured to the lathe to extend the versatility of the lathe to include taper-cutting, milling, and grinding. Some of the common accessories and attachments used on lathes are described in the following paragraphs.

quick change—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 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 toolholders. It mounts in the T-slot and the toolholder (boring bar) passes through it 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 shops. 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 of 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 of toolpost, only the toolholders are changed, allowing the toolpost to remain firmly in place.

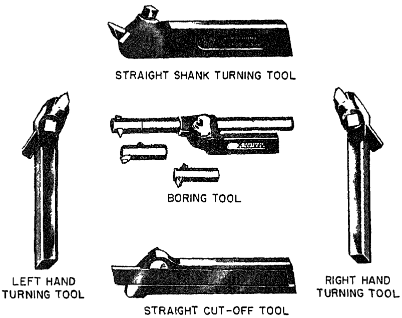


28.300

Figure 7-19.—Quick change toolpost.

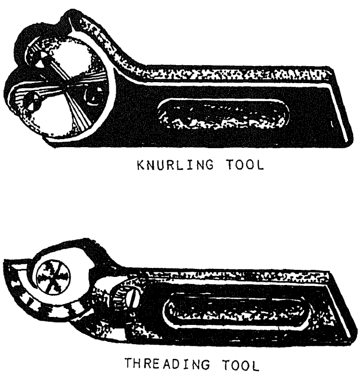
TOOLHOLDERS

Lathe toolholders are designed to be used with the various types of toolposts. Only the 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.



28.67

Figure 7-20.—Standard lathe toolholders.



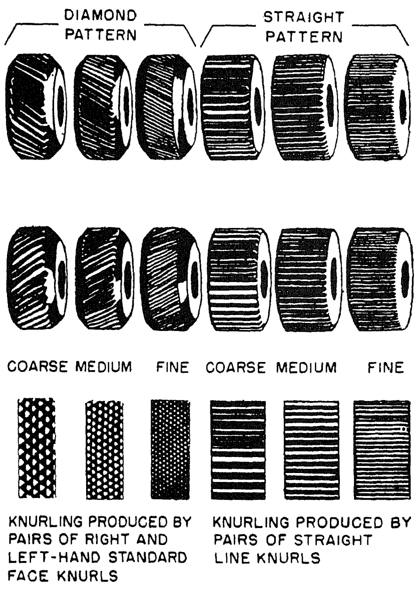
28.67

Figure 7-21.—Knurling and threading tools.

Standard toolholders were discussed briefly in chapter 6 of this manual. However, there are more types (fig. 7-20) than those 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



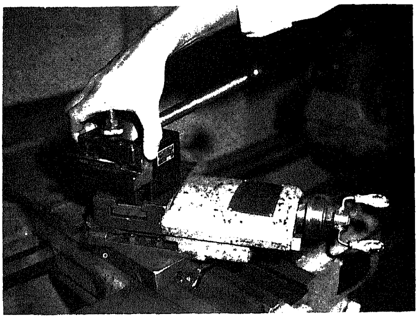
28.301

Figure 7-22.—Types of knurling rollers.

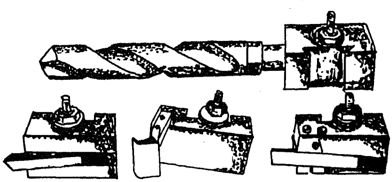
knees, to give a better grip for handling. The knurled roller comes in a wide variety of patterns. (See fig. 7-22.)

The **BORING BAR** 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. 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 with the castle type toolpost.

The **QUICK CHANGE** toolholder (fig. 7-23) is mounted on the toolpost by sliding it from



28.302
Figure 7-23.—Quick change toolpost and toolholder.



28.303
Figure 7-24.—Quick change toolholder.

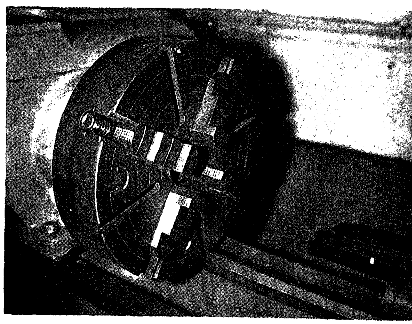
toolholder has a height adjusting ring to allow you to set the proper height prior to locking it in place. The quick change toolholder comes in a wide range of styles. A few of these styles are shown in figure 7-24.

LATHE CHUCKS

The lathe chuck is a device for holding lathe work. It is mounted on the nose of the spindle. 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.

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



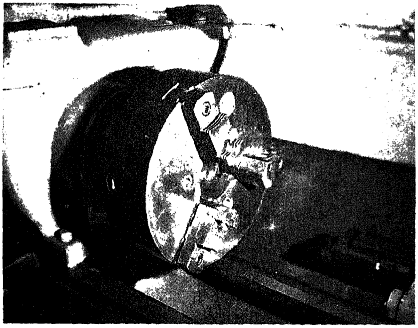
28.304
Figure 7-25.—Four-jaw independent chuck.

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 chuck.

COMBINATION CHUCKS are universal chucks that have 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 faceplate 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 onto a boss on the back of the chuck to secure the chuck to the spindle nose. On small lathes, chucks are screwed directly onto the threaded spindle nose. (See fig. 7-5.)



28.305

Figure 7-26.—Three-jaw universal chuck.

The **DRAW-IN COLLET** chuck is used to hold small work for machining. It is the most accurate type of chuck and is intended for precision work.

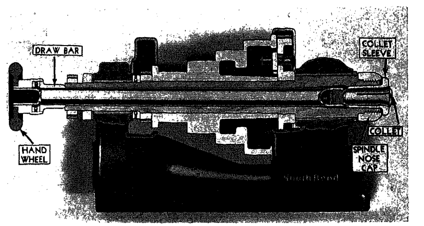
Figure 7-27 shows the 5 parts of the collet chuck assembled in place in the lathe spindle. The collet, which holds the work, is a split cylinder with an outside taper that fits into the tapered closing sleeve and screws into the threaded end of the hollow drawbar that passes through the hollow spindle. When the handwheel, which is attached by threads to the outside of the drawbar, is turned clockwise, the drawbar pulls the collet 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 the 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 available.

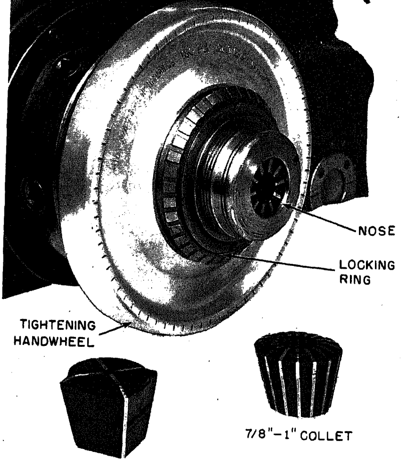
The **RUBBER COLLET CHUCK** (fig. 7-28) is designed to hold any bar stock from 1/16 inch up to 1 3/8 inch. It is different from the draw-in type collet previously mentioned in that the bar stock does not have to be exact in size.

The rubber flex collet consists of rubber and hardened steel plates. The nose of the chuck has



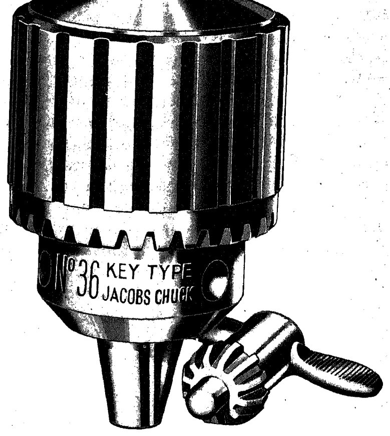
28.91X

Figure 7-27.—Draw-in collet chuck assembled.



28.306

Figure 7-28.—Rubber flex collet chuck.



28.92X

Figure 7-29.—Drill chuck.

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 enables you to align the stock very accurately. The locking ring, when pressed in, gives a safe lock that prevents the collet from coming loose when the machine is in operation.

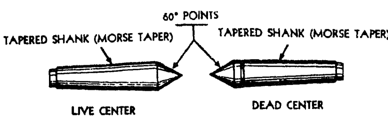
DRILL CHUCKS are used to hold center drills, straight shank drills, reamers, taps, and small rods. The drill chuck is mounted on a tapered shank or arbor which fits the Morse taper hole in either the headstock or tailstock spindle. Figure 7-29 shows the three-jaw type. A revolving sleeve operated by a key opens or closes the three jaws simultaneously to clamp and center the drill in the chuck.

FACEPLATES are used for holding work that cannot be swung on centers or in a chuck because of its shape or dimensions. The T-slots and other openings on the surface of the faceplate provide convenient anchor points for bolts and clamps used to secure the work to the faceplate. 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 to transmit 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



28.93

Figure 7-30.—Lathe centers.

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 with the work and 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.

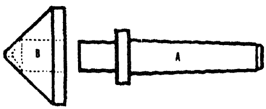
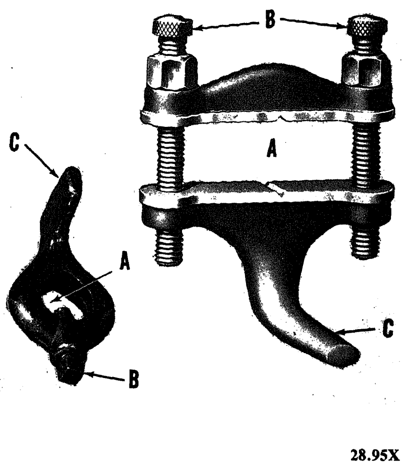


Figure 7-31.—Pipe center.



28.95X

Figure 7-32.—Lathe dogs.

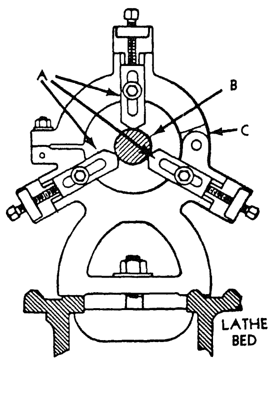
To remove the headstock center, insert a brass rod through the spindle hole and tap the center to jar it loose; you can then pull it out with your hand. To remove the tailstock center, run the spindle back as far as it will go by, turning the handwheel to the left. When the end of the tailstock screw bumps the back of the center, it will force the center 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 or center is used.

LATHE DOGS

Lathe dogs are used with a driving plate or faceplate to drive work being machined on centers whenever the frictional contact alone between the live center and the work is not sufficient to drive the work.



28.96X

Figure 7-33.—Center rest.

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 irregularly 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 from springing due to cutting pressure 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 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 the center rest in position without removing the work from the centers or changing the position of the jaws.

FOLLOWER REST

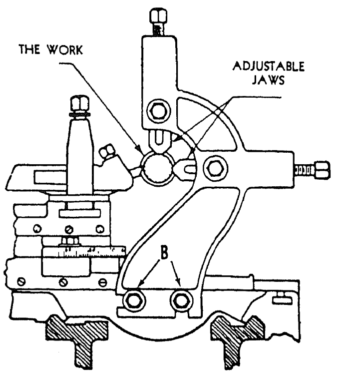
The follower rest is used to back up work of small diameter to keep it from springing under the pressure of cutting. This rest gets its name because it follows the cutting tool along the work. As shown in figure 7-34, it is attached directly to the saddle by bolts B. The adjustable jaws bear directly on the finished diameter of the work opposite and above the cutting tool.

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 carriage moves longitudinally, thereby causing the cutting tool to move at an angle to the axis of the work to produce a taper.

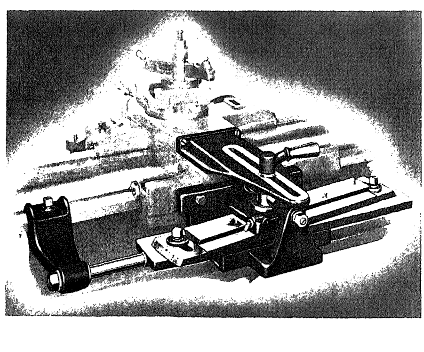
The angle of the desired taper is set on the guide bar of the attachment, and the guide bar support is clamped to the lathe bed.

Since the cross-slide is connected to a shoe that slides on the guide bar, the tool follows along a



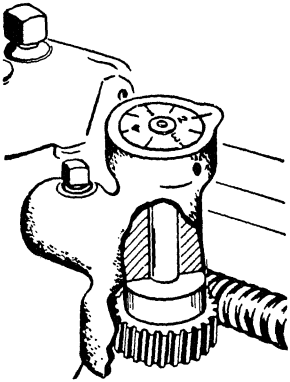
28.97X

Figure 7-34.—Follower rest.



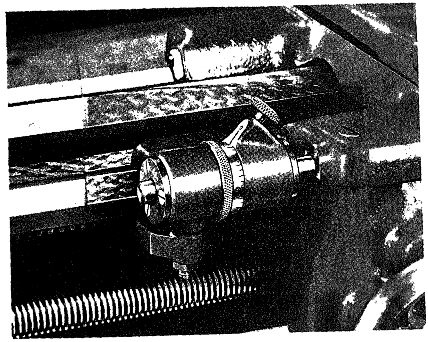
28.98X

Figure 7-35.—A taper attachment.



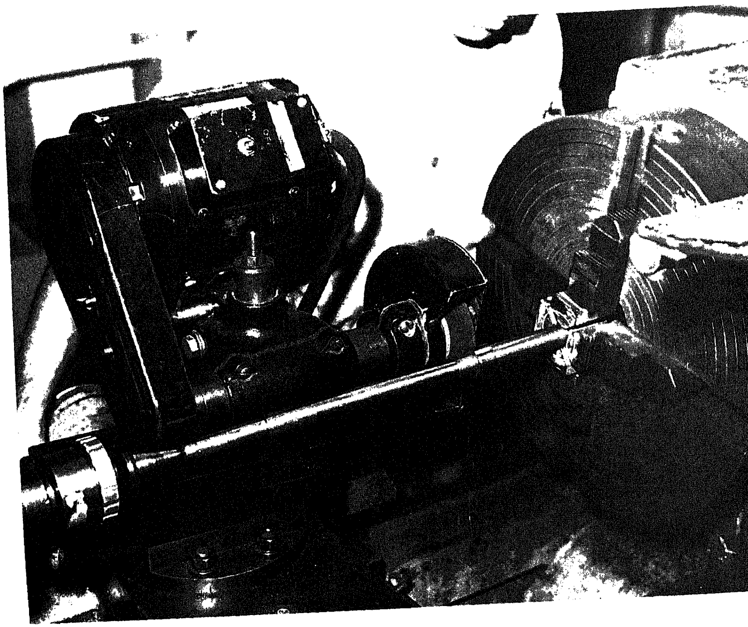
28.99X

Figure 7-36.—Thread dial indicator.



28.100X

Figure 7-37.—Micrometer carriage stop.



28

Figure 7-38.—Grinder mounted on compound rest.

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 explained further 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, it should be disengaged 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. To operate the carriage stop, 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 using the micrometer adjustment. **NOTE**: Do not confuse this stop with the automatic carriage stop that automatically stops the carriage by disengaging the feed or stopping the lathe.

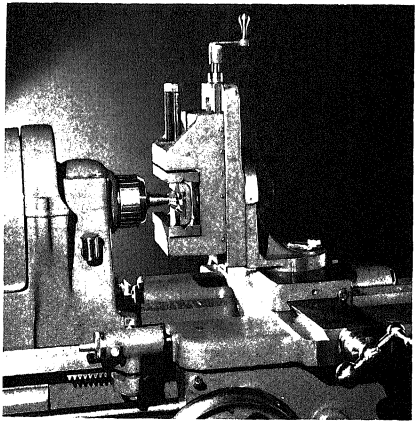
GRINDING ATTACHMENT

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 onto 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.

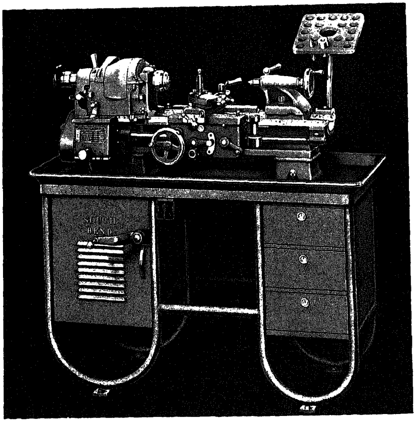
The milling cutter is held in an arbor driven by the lathe spindle. The work is held in a vise on the milling attachment. The milling attachment is mounted on the cross-slide and therefore its movement can be controlled by the longitudinal feed and cross feed of the lathe. The depth of the cut is regulated by the longitudinal feed while the length of the cut is regulated by the cross feed. 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.



28.102X

Figure 7-39.—Milling attachment.

A milling attachment is unnecessary in shops equipped with milling machines.

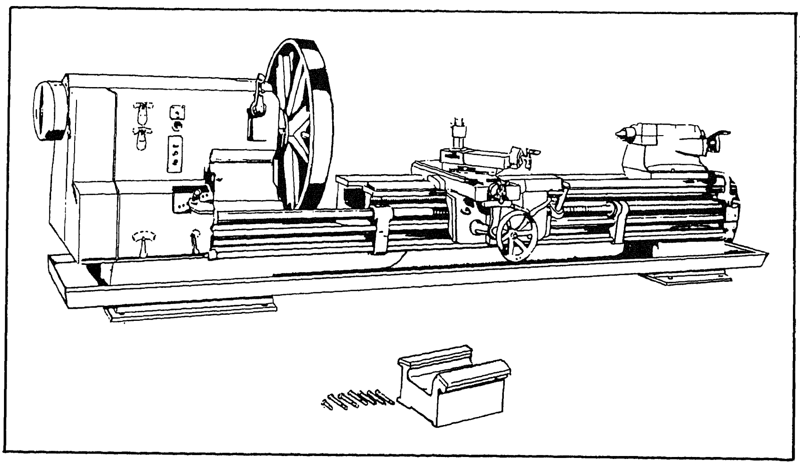


28.103X

Figure 7-40.—A bench lathe.

TRACING ATTACHMENTS

A tracing attachment for a lathe is useful whenever you have to make several parts that are identical in design. 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 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 tracer 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



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. 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

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 longitudinally to create the gap.

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 you must follow 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. Second, you must be familiar with the specific type of 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 your shirt sleeves and removing your watch, rings, and other jewelry that might become caught while you operate a machine.
2. Wear safety glasses or an approved face shield at all times when you operate a lathe or when you are in the area of lathes that are in operation.
3. Be sure the work area is clear of obstructions that might cause you to trip or fall.
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. Always get someone to help you handle 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 you remove 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 you talk 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 you must operate a lathe while underway, be especially safety conscious. (Machines should be operated only in relatively calm seas.)
12. Know where the cutting tool is while you take measurements or make adjustments to the machine.
13. Always observe the specific safety precautions posted for the machine you are operating.

MACHINE CHECKOUT

Before you attempt to operate any lathe, make sure you know how to run it. Read all operating instructions supplied with the machine. Know where the various controls are and how to operate them. When you are satisfied that you know how the controls work, check to see that the spindle clutch and the power feeds are disengaged; then 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 feel 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 the 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 you can close the half-nuts 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 you may make changes in the gear box with the lathe running slowly, but you must stop the lathe to make speed changes by shifting gears in the main gear train.

Maintenance is an important operational procedure for lathes and must be performed according to 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 wherever oil holes are provided. Oil the ways often, not only to lubricate them 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 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 the machine is 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 or she 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 you oil the lathe or the 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 cross feed 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 required changes. Ensure that the toolholder and the 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.

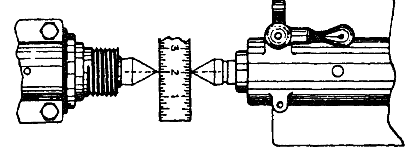
PREPARING THE CENTERS

The first step in preparing the centers is to see that they are accurately mounted in the headstock 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 and 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 gauge. The large notch of the center gauge 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°. To true a hardened tail center, either anneal it and then machine it or grind it if a grinding attachment is available.

Aligning and Testing

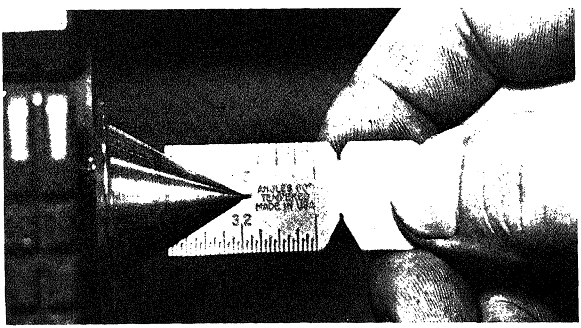
To turn a shaft straight and true between centers, be sure the centers are in the same plane parallel to the ways of the lathe. You can align the centers by releasing the tailstock from the ways and then moving the tailstock laterally with two adjusting screws. At the rear of the tailstock are two zero 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.



28.106X

Figure 8-2.—Aligning lathe centers.

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, adjust the tailstock in the direction required by using the set-over adjusting screws. Repeat the above test and adjustment until a cut at each end produces equal diameters.



28.105

Figure 8-1.—Checking center point with a center gauge.

You can also check for positive alignment of the centers by placing a test bar between the centers and checking both ends of the bar with a dial indicator clamped in the toolpost (fig. 8-3). If the reading on the dial is zero at both ends of the bar, the centers are aligned. 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.

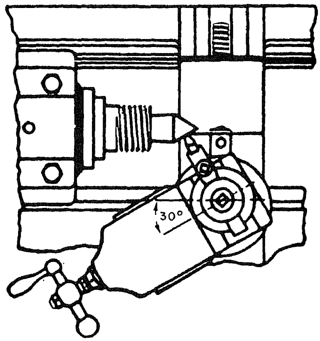
Another method you can use to check for positive alignment of lathe centers is to take a light cut over the work held between centers. Then measure the work at each end with a micrometer. If the readings differ, adjust the tailstock to remove the difference. Repeat the procedure until the centers are 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 gauge. All lathe centers, regardless of their size, are finished to an included angle of 60°.

Recall that if you must true the tailstock spindle lathe center, 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.

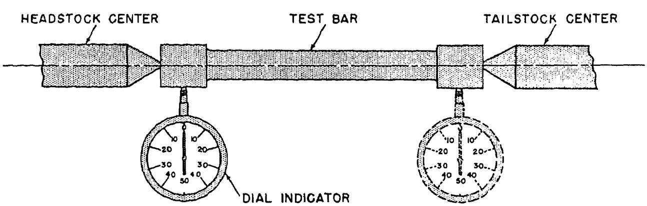
Also if a toolpost grinder is available, you may true the hardened center by grinding it without annealing it. As in machining, the first step after placing the center in the headstock spindle is to



28.108X

Figure 8-4.—Machining a lathe center.

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 the entire length of the center point. Then check the angle with a center gauge. Reset the compound rest if necessary and continue grinding until the center fits the center gauge exactly. To check the accuracy of the fit, place a light beneath the center and look for light between the center point surface and the edge of the center point gauge.



28.107

Figure 8-3.—Alignment of lathe centers with a dial indicator.

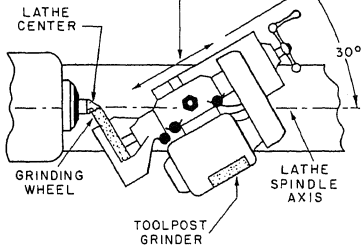
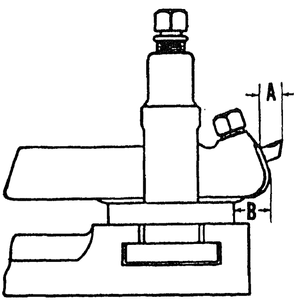


Figure 8-5.—Grinding a lathe center.

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 rigidly mounted on the tool post holder. Be sure the tool sits squarely in the toolpost and that the setscrew is tight. Reduce overhang as much as possible to prevent the tool from 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



28.110X

Figure 8-6.—Tool overhang.

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 cutting tool to straight turn steel, cast iron, and other relatively hard metals, set the point on center. The point of a high-speed steel cutting tool being used to cut aluminum, copper, brass, and other soft metals should be set exactly on center. The point of cast alloy (stellite and so on), carbide, and ceramic cutting tools should be placed exactly on center regardless of the material being cut. The tool point should be placed on center for threading, turning tapers, parting (cutting-off) or boring.

You can adjust the height of the tool in the toolholder illustrated in figure 8-6 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. You can place a dead center in the tailstock and 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. You can also 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

You cannot perform accurate work if the 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 held rigidly 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.

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 provide a smooth and even surface and will help prevent the center drill from "wandering" or breaking. 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 you must center a piece very accurately, bore the tapered center hole after you center drill to correct any run-out of the drill. You can do this by grinding a tool bit to fit a center gauge 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 center for height and adjust the tool to the proper angle with the center gauge 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. 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 show 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 to prevent breaking 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 you can jar it loose, or you may have to drive it out by using a chisel. But it may stick so hard that you

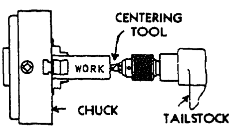


Figure 8-7.—Drilling center hole.

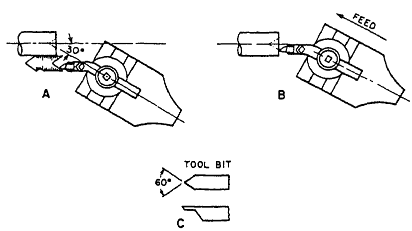
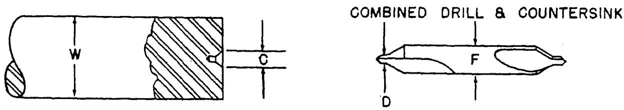


Figure 8-8.—Boring center hole.



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.

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.

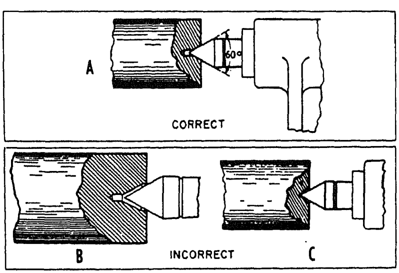
Figure 8-10 shows correct and incorrect countersinking for work to be machined on centers. In example **A**, the correctly countersunk hole is deep enough so that the point of the lathe centers does not come in contact with the bottom of the hole.

In example **B** of figure 8-10, the countersunk hole is too deep, causing only the outer edge of the hole 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 having too large an 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.

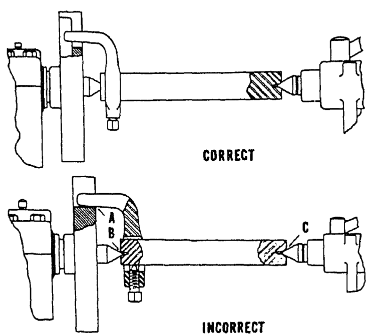
Mounting the Work

Figure 8-11 shows correct and incorrect methods of mounting work between centers. In



28.114X

Figure 8-10.—Examples of center holes.



28.115X

Figure 8-11.—Examples of work mounted between centers.

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.

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**, causing the work to revolve eccentrically.

When you mount work between centers for machining, there 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 of one 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 of one thousandth of an inch under the standard size of the mandrel, while on larger sizes 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, you will not be able to drive the mandrel 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 one made with a dial indicator. Mount the indicator on the toolpost so 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 you press or drive 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.

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 holds the work properly.

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.

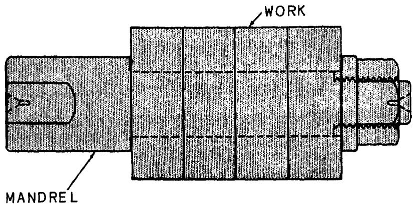
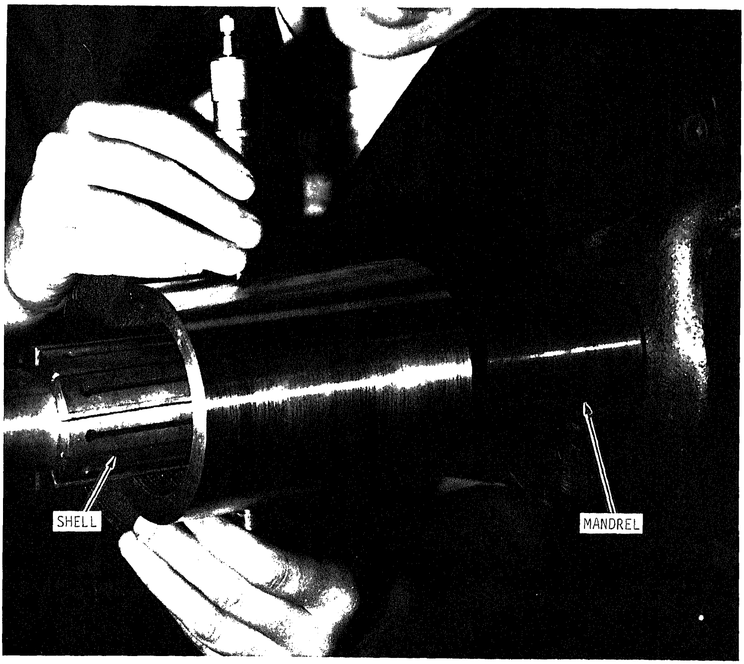


Figure 8-13.—Gang mandrel.

An **ECCENTRIC** mandrel has two sets of countersunk holes, one pair of which is off-center



28.116

Figure 8-12.—A split-shell expansion mandrel.

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 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.
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.

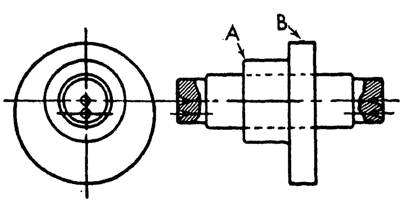


Figure 8-14.—Work on an eccentric mandrel.

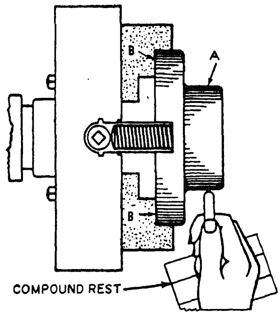


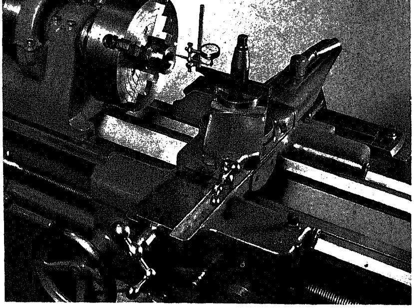
Figure 8-15.—Work mounted in a 4-jaw independent chuck.

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 tank.
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 clamp 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



28.120X
Figure 8-16.—Centering work with a dial indicator.

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 to a similar number on the chuck. When you remove a chuck jaw for any reason, always put it back into the proper slot.

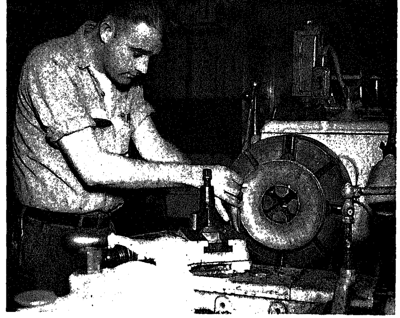
When the work to be chucked is frail or light, tighten the jaw carefully so the work will not bend, break, or spring.

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 may develop inaccuracies of from .002 to .010 inch in centering the work, requiring the run-out of the work to 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**.



28.121
Figure 8-17.—Work held from inside by a 4-jaw independent chuck.

When you chuck thin sections, be careful not to clamp the work too tightly, since the diameter of the piece will be machined while the piece is distorted. Then, when you release the pressure of the jaws after finishing 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 hollow 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. Turn the handle to the left to release 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 tapered 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 seat properly against the spindle shoulder and will prevent the chuck from running true. Screw the collar carefully onto the chuck and tighten it enough to make it difficult to remove the chuck. Never use mechanical power to install a chuck, but rotate the collar with your left hand while you support the chuck in the hollow of your right arm.

To remove a 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 in the direction you wish the chuck to rotate. When you mount or remove 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 tapped 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.

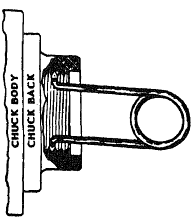
HOLDING WORK ON A FACEPLATE

A faceplate 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 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.)

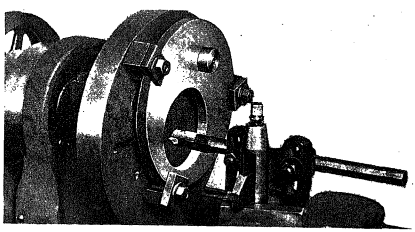
For work to be mounted accurately on a faceplate, the surface of the work in contact with the faceplate must be accurate. Check the accuracy with a dial indicator. If you find runout, reface the surface of the work that is in contact with the faceplate. It is good practice to place a piece of paper between 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



28.122X

Figure 8-18.—Tool for cleaning thread of a chuck or faceplate.



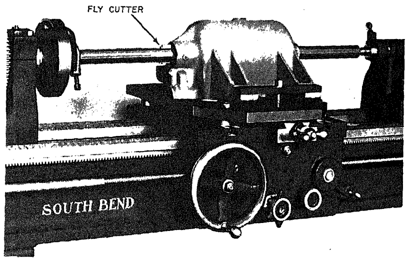
28.123X

Figure 8-19.—Eccentric machining of work mounted on a faceplate.

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.



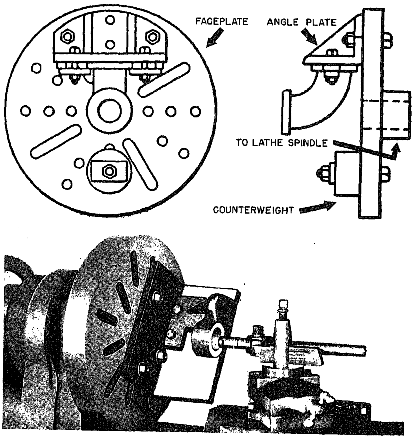
28.128X
Figure 8-21.—Work mounted on a carriage for boring.

USING THE CENTER REST AND FOLLOWER REST

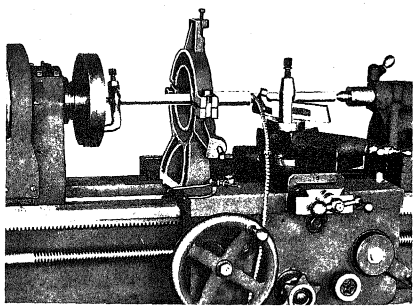
Long slender work often requires support between its ends while it is turned; otherwise the work would spring away from the tool and chatter. The center rest is used to support such work so it can be turned accurately at a faster feed and cutting speed than would be possible without the center rest. (See fig. 8-22).

Place the center rest 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



28.124X
Figure 8-20.—Work clamped to an angle plate.



28.125X
Figure 8-22.—Use of a center rest to support work between centers.

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.

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" to 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 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 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 and push 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 performing machine work on 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.

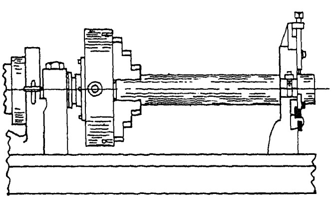
PLANNING THE JOB

It is important for you to 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 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

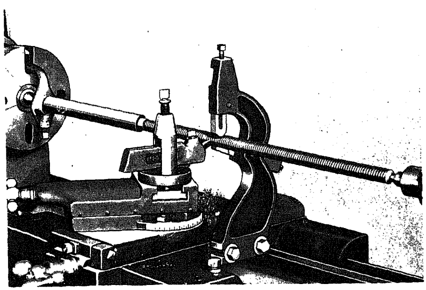
Cutting speed is the rate at which the surface of the work passes the point of the cutting tool. It is expressed in feet per minute (fpm).

To find the cutting speed, multiply the diameter of the work (DIA) in inches times 3.1416



28.126X

Figure 8-23.—Work mounted in a chuck and center rest.



28.127X

Figure 8-24.—Follower rest supporting screw while thread is being cut.

$$CS = \frac{DIA \times 3.1416 \times rpm}{12}$$

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. Use the following formula:

$$rpm = \frac{CS \times 12}{DIA \times 3.1416}$$

Table 8-1 gives the recommended approximate 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

TYPE OF MATERIAL	Cutting Speed (fpm)
Low carbon steel	40-140
Medium carbon steel	70-120
High carbon steel	65-100
Stainless steel, C1 302, 304	60
Stainless steel, C1 310, 316	70
Stainless steel, C1 410	100
Stainless steel, C1 416	140
Stainless steel, C1 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 practical 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 fast 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 practical speed. On many pieces, when tool failure is the limiting factor in the size of the 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 of when the depth of cut is 1/4 inch, the feed is 20 thousandths of an inch per revolution, and the speed is 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, run the work as fast as the tool will withstand to obtain the maximum speed in this operation. Use a sharp tool to finish turning.

Cutting Lubricant

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 machining 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 with a center rest or follower rest.

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 and perpendicular to the work.

If none of the above actions works, examine the lathe and its adjustments. Gibs may be 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 specific 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

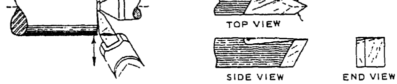
Regardless of how the work is held in the lathe, the tool should feed toward the headstock. This causes most of the pressure of the cut to be exerted on the workholding device and the spindle thrust bearings. When you must feed the cutting tool toward the tailstock, take lighter cuts at reduced feeds. In facing, the general practice is to feed the tool from the center of the workpiece toward the periphery.

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 the dog drives the end just faced. Use a steel ruler to layout the required length, measuring from the faced end to the end



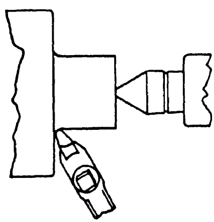
28.129X

Figure 8-25.—Right-hand side tool.

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.

In facing large surfaces, lock 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.)



28.130X

Figure 8-26.—Facing a shoulder.

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.

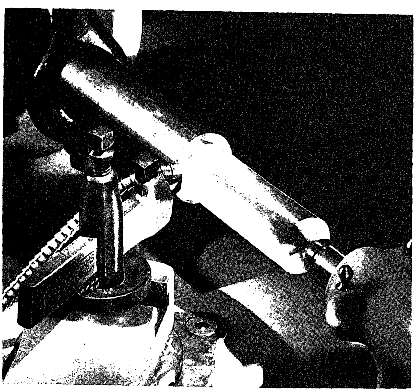
To remove large amounts of stock in most lathe machining, you will take a series of roughing cuts to remove most of the excess stock and then a finishing cut to accurately "size" the workpiece.

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 deeply enough to get under the scale in the first cut. If you do not, the scale on the metal will dull the point of the tool.



28.131X

Figure 8-27.—Rough turning.

Rough machine the work to almost the finished size; then be very careful in taking measurements on the rough surface.

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 the center and the center hole becoming worn. Always check the center carefully and adjust as needed during rough turning operations.

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.

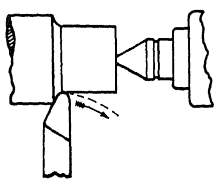
Finish Turning

When you have rough turned the work 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 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 the piece has expanded slightly because of the heat generated by turning and you turn it to size while it is hot, 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 taking accurate measurements. So much depends on the accuracy



28.132X

Figure 8-28.—Position of tool for heavy cut.

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 known dimensions. You will acquire the skill if you are persistent.

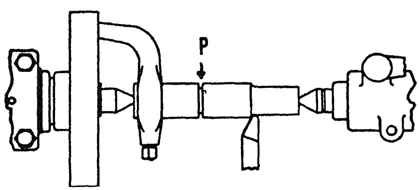
Turning to a Shoulder

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.



28.133X

Figure 8-29.—Machining to a shoulder.

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. The end cutting edge of the tool must feed directly toward the center of the workpiece. To ensure this, place a center in the tailstock and align the parting tool vertically with the tip of the 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 separates the work.

Cutting speeds for parting are usually slower than turning speeds. You should use a feed that

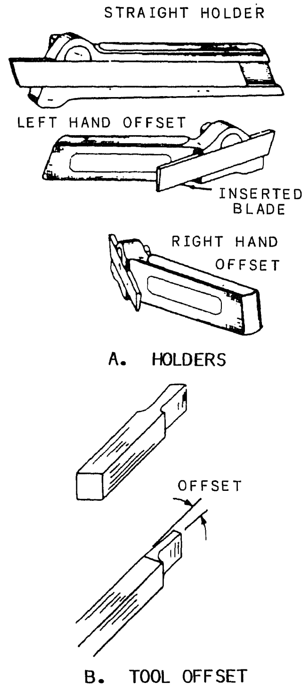


Figure 8-30.—Parting tools.

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, 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.

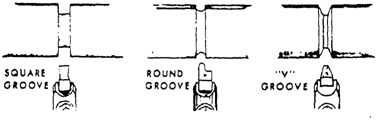


Figure 8-31.—Three common types of grooves.

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, set the work speed at 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 to 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 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.

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 the threading operation much easier to do.

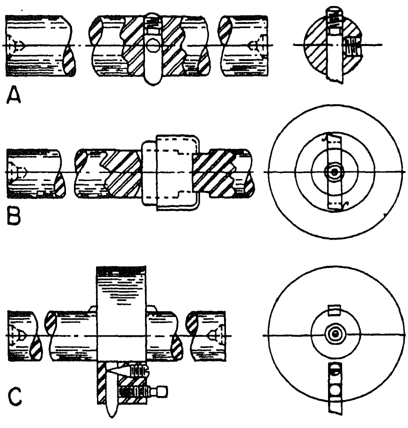
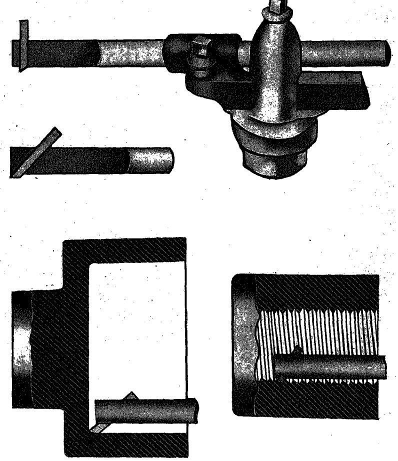


Figure 8-32.—Various boring bars.



28.135

Figure 8-33.—Application of boring bar holder.

KNURLING

Knurling is the process of rolling or squeezing impressions into the work with hardened steel rollers that have teeth milled into 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 also used for decoration. Knurling increases the diameter of the workpiece slightly when the metal is raised by the forming action of the knurl rollers.

The knurling tool (fig. 7-23) is set up so the faces of the rollers are parallel to the surface of the work and with the upper and lower rollers 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 rollers 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 rollers extends beyond the end of the work. This eliminates part of the pressure required to start the knurl impression. Force the knurling rollers into contact with the work. Engage the spindle clutch. Check the knurl to see if the rollers 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 rollers into contact with the correctly formed knurled impressions. The rollers 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 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.

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 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

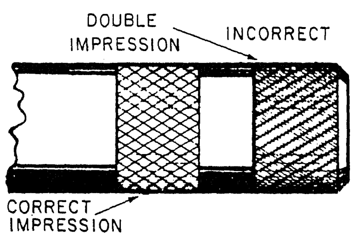


Figure 8-34.—Knurled impressions.

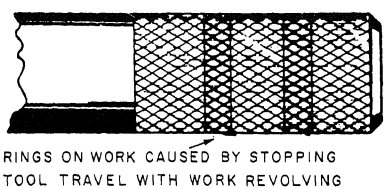


Figure 8-35.—Rings on a knurled surface.

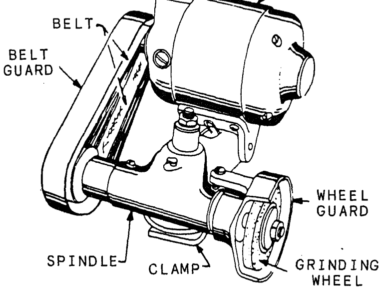


Figure 8-36.—Toolpost grinder.

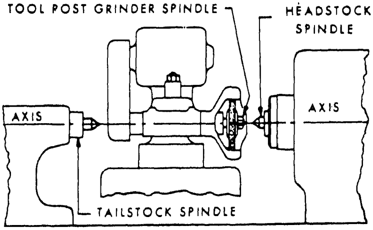


Figure 8-37.—Mounting the grinder at center height.

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.

Before you use the grinder, dress and true the wheel with a diamond wheel dresser. 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.)

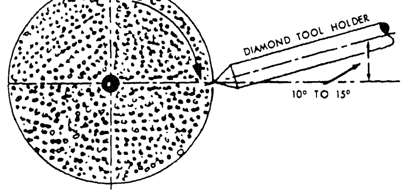


Figure 8-38.—Position of the diamond dresser.

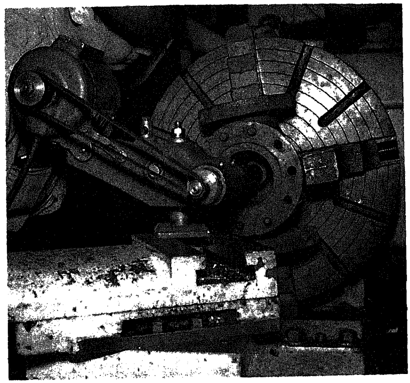
Bring the grinding wheel into contact with the diamond dresser by carefully feeding the cross-slide in by hand. Move the wheel slowly by hand back and forth over the point of the diamond, taking a maximum cut of .0002 inch. Move the carriage if the face of the wheel is parallel to the ways of the lathe. 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.

Rotate the work at a fairly low speed during the grinding operation. The recommended surface 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, as shown in figure 8-5, first ensure that the spindle holes, drill sleeves, and centers are clean and free of burrs. Install the lathe center to be refinished in the headstock. 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 inch. 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 the depth of cut; use the compound rest for traversing the grinding wheel across the work surface. Remember, whenever you grind 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.

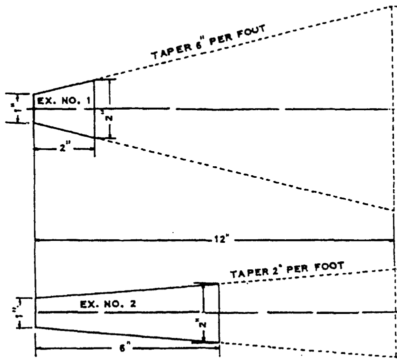


Figure 9-1.—Tapers.

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 × 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:

$$TPF = TPI \times 12$$

where:

TPF = TAPER PER FOOT

TPI = TAPER PER INCH

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

$$T = \frac{L \times TPF}{12} \text{ and } T = TPI \times L \text{ (in inches)}$$

$$TPI = \frac{TPF}{12}$$

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 center line. The included angle is the angle between the two angled sides of the taper. The angle with the center line is the angle between the center line and either of the angled sides. Since the taper is turned about a center line, the angle between one side and the center line is always equal to the angle between the other side and the center line. Therefore, the included angle is always twice the angle with the center line. 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 center line.

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 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 tapered holes in lathe and drill press spindles and the attachments that fit them, such as lathe centers, drill shanks, and so on; (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 **JARNO TAPER** (0.600 inch per foot) used by some manufacturers because of the ease with which its dimensions can be determined:

$$\text{Diameter of large end} = \frac{\text{taper number}}{8}$$

$$\text{Diameter of small end} = \frac{\text{taper number}}{10}$$

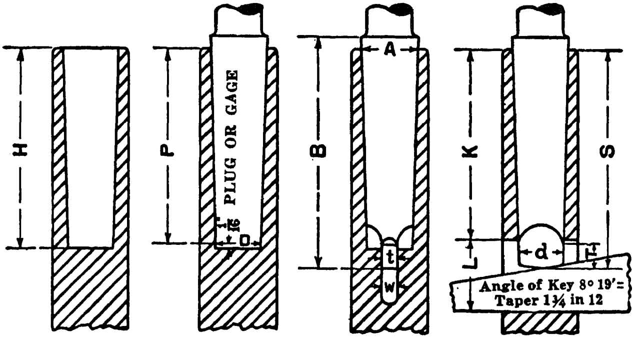
$$\text{Length of taper} = \frac{\text{taper number}}{2}$$

Two additional tapers that 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.

Table 9-1.—Tapers Per Foot/Angles

Taper per foot	Angle included		Angle with centerline		Taper per inch
	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



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.

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 on 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.

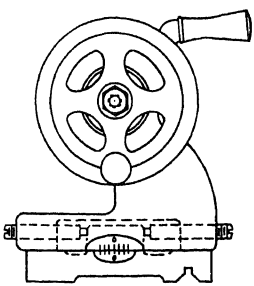
3. **USE THE TAPER ATTACHMENT,** which also causes the cutting tool to move at an angle to the axis of the lathe.

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

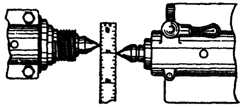
As stated in chapter 7, you can move the tailstock top sideways on its base by using the adjusting screws. In straight turning you use these adjusting screws to align the dead center with the tail center by moving the tailstock to bring it on the center line of the spindle axis. For taper turning, you deliberately move the tailstock off center, and the amount you move it determines the taper produced. You can approximate the amount of setover by using 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 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 WILL BE THE TAPER PRODUCED. Suppose you want to turn a taper on the full length of a piece 12 inchés 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 of 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 work than at the other end; so the diameter of the finished job will be 2 × 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.



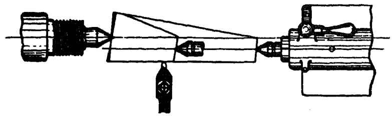
28.139X

Figure 9-3.—Tailstock setover lines for taper turning.



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.

By now you can see that the setover is proportional to the length between centers. Setover is computed by using the following formula:

$$S = \frac{T}{2} \times \frac{L}{12}$$

where:

S = setover in inches

T = taper per foot in inches

L = length of taper in inches

$\frac{L}{12}$ = length in feet of taper 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. 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 will make with the center line (that is, half of the included angle of the taper). Then feed the tool to the work at this angle by using the compound rest feed screw. The length of taper you can machine is short because the travel of the compound rest is limited.

One example of using the compound rest for taper work is the truing of a lathe center. Other examples are refacing an angle type valve disk and machining the face of a bevel gear. Such jobs are often referred to as working to an angle rather than as taper work.

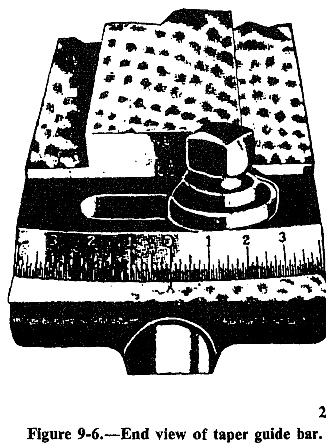
The graduations marked on the compound rest provide a quick means for setting it to the angle desired. When the compound rest is set at zero, the cutting tool is perpendicular to the lathe axis. When the compound rest is set at 90° on either side of zero, the cutting tool is parallel to the lathe axis.

To set up the compound rest for taper turning, first determine the angle to be cut, measured from the center line. This angle is half of the included angle of the taper you plan to cut. Then set the compound rest to the complement of the angle to be cut (90° minus angle to be cut). For example, to machine a 50° included angle (25° angle with the center line), set the compound rest at 90° − 25°, or 65°.

When you must set the compound rest very accurately, to a fraction of a degree for example, 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 that corresponds to the desired taper. The tool cross slide will be moved laterally by a shoe, which slides on the guide bar as the carriage moves longitudinally. The cutting tool will move along a line parallel to the guide bar. The taper produced will have the same angular measurement 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 provide for rapid setting. Figure 9-6 is a view of the end that is graduated in inches per foot of taper.

When you prepare 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 center 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)



28.142X

Figure 9-6.—End view of taper guide bar.

bar now controls the lateral movement of the cross slide. Set the guide bar for the taper desired; the attachment is ready for operation. To make the final adjustment of the tool for size, use 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.

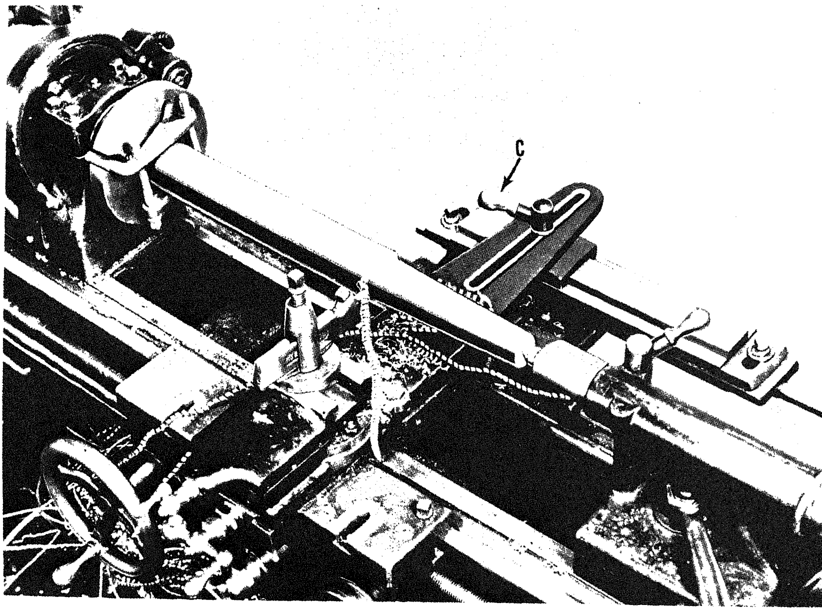
TAPER BORING

Taper boring is usually done with either the compound rest or the taper attachment. The rules 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 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 gauge and the "cut and try" method.

1. After you have taken one or two cuts, clean the bore.



28.143X

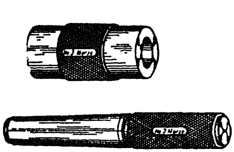
Figure 9-7.—Turning a taper using taper attachment.

3. Insert the gauge into the hole and turn it SLIGHTLY so the chalk (or prussian blue) rubs from the gauge 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 gauge to test the taper.

4. Areas that do not touch the gauge 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 gauge. Be sure the taper diameter is correct before you turn the taper to its finish diameter.

Figure 9-8 shows a Morse standard taper plug and a taper socket gauge. They not only give the proper taper, but also show the proper distance that the taper should enter the spindle.



28.144X

Figure 9-8.—Morse taper socket gauge and plug gauge.

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 that you must know before you can calculate and machine screw threads. Figure 9-9 illustrates some of the following terms:

EXTERNAL THREADS: A thread on the outside surface of a cylinder.

INTERNAL THREAD: A thread on the inside surface of a hollow cylinder.

RIGHT-HAND THREAD: A thread that, when viewed axially, winds in a clockwise and receding direction.

LEFT-HAND THREAD: A thread that, when viewed axially, winds in a counterclockwise and receding direction.

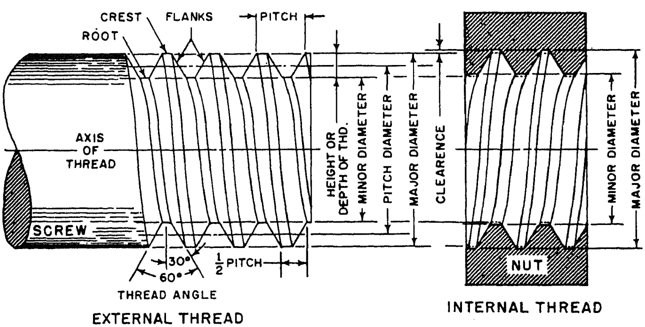


Figure 9-9.—Screw thread nomenclature.

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.

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: A group of threads designed for a certain type of fit. Classes of threads are distinguished from each other by the amount of tolerance and allowance specified.

THREAD FORM: The view of a thread along the thread axis 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 widths of the thread and the thread groove are equal. The pitch diameter is the diameter that 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 that is used for identification. For example, the nominal size of a 1/2-20 thread is 1/2 inch, but its actual size 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 an imaginary cylinder that passes through the crests of an external thread or the roots of an internal thread.

MINOR DIAMETER: The diameter of an imaginary cylinder that passes through the roots of an external thread or the crests of an internal thread.

HEIGHT OF THREAD: The distance from the crest to the root of a thread measured along a 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, the designer needs only to specify size, number of threads per inch, designation of the standard series and class of fit. With these specifications, all other information necessary for production can be 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
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

designated by nominal size, number of threads per inch, series symbol, and class symbol, in that order. For example, the designation 1/4-20 UNC-3A specifies a thread with the following characteristics:

Nominal thread diameter = 1/4 inch

Number of threads per inch = 20

Series (Unified coarse) = UNC

Class = 3

External thread = A

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 tap drill size is equal to the thread size minus the pitch, or the thread size minus ONE divided by the number of threads per inch.

$$\text{Tap Drill Size} = \text{Thread Size} - \frac{1}{n}$$

workpiece (at one half of the included angle of the thread), use the slant-depth 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 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 × pitch

2. AMERICAN NATIONAL THREAD

   Pitch = 1 ÷ number of threads per inch or $\frac{1}{n}$

   Straight depth of external thread = 0.64952 × pitch or 0.541266p

   Straight depth of internal thread = 0.541266 × pitch or 0.64952p

   Width of flat at point of tool for external and internal threads = 0.125 × pitch or 0.125p

   Slant depth of external thread = 0.750 × pitch or 0.750p

   Slant depth of internal thread = 0.625 × 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 × pitch or 0.61343p

   Straight depth of internal thread = 0.54127 inch × pitch or 0.54127p

   Width of flat at root of external thread = 0.125 inch × pitch or 0.125p

   Width of flat at crest of external thread = 0.125 inch × pitch or 0.125p

   Double height of external thread = 1.22687 inch × pitch or 1.22687p

   Double height of internal thread = 1.08253 inch × pitch or 1.08253p

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 screwed. The direction of the 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 is 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 formulas 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 × pitch

Root radius = 0.0714 × pitch

Depth of thread = 0.6627 × pitch

The classes of fit are: 1 = free, 2 = medium, 3 = close. The specific dimensions involved concern the tolerance of 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

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 sealing tape or 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 at this angle. The hole for the internal threads should be slightly larger than the minor diameter of the small end of the externally threaded part.

An example of a pipe thread is shown below.

NPT 1/4-18

where NPT = tapered pipe thread

1/4 = inside diameter of the pipe in inches

18 = threads per inch

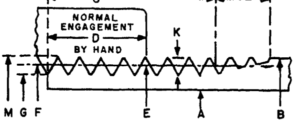
Figure 9-15 shows the typical dimensions of the most common tapered pipe threads.

STRAIGHT PIPE THREADS

Straight pipe threads are similar in form to tapered pipe threads except that they are not tapered. The same nominal outside diameter and thread dimensions apply. Straight pipe threads are used for joining components mechanically 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



BELOW.

ANGLE BETWEEN SIDES OF THREAD IS 60°. TAPER OF THREAD, ON DIAMETER, IS $\frac{3}{4}$ 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

Figure 9-15.—Taper pipe thread dimensions.

for each particular class. The tolerance (amount that a thread may vary from the basic dimension) decreases as the class number increases. For 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 follows:

- Classes 1A and 1B: Class 1A (external threads) and class 1B (internal) threads are used where quick and easy assembly is necessary and where a liberal allowance is required to permit ready assembly, even with slightly bruised or dirty threads.

- Classes 2A and 2B: Class 2A (external) and class 2B (internal) threads are the most commonly used threads for general applications including production of bolts, screws, nuts and similar threaded fasteners.

- Classes 3A and 3B: Class 3A (external) and class 3B (internal) threads are used where closeness of fit and accuracy of lead and angle of thread are important. These threads require consistency that is available only through high quality production methods combined with a very efficient system of gauging 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 that are used to determine the dimensions of threads.

The use of a mating part to estimate and check the needed thread is common practice when average accuracy is required. The thread is simply machined until the thread and 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 gauge. In these cases, you must measure 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, and so on. 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 GAUGES

Go and no-go-gauges, such as those shown in figure 9-17, are often used to check threaded parts. The thread should fit the "go" portion of the gauge, but should not screw into or onto the "no-go" portion. Ring and plug gauges 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 and the major and minor diameters, but also the lead of the thread.

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

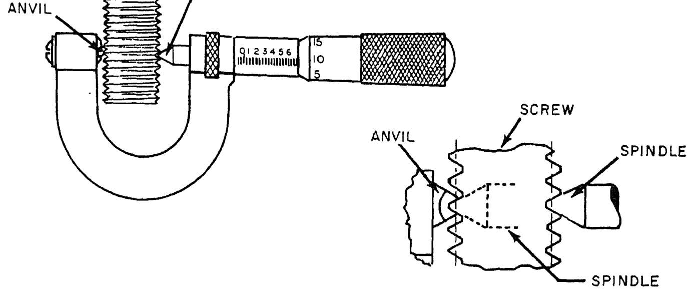


Figure 9-16.—Measuring threads with a thread micrometer.

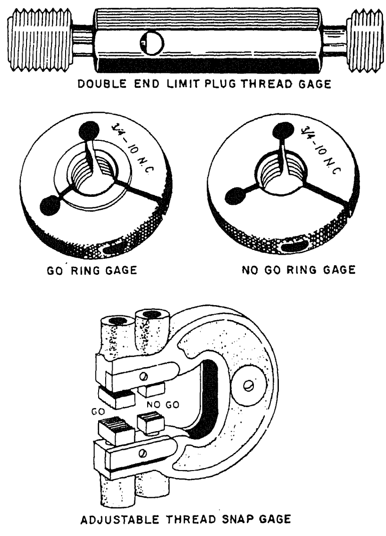


Figure 9-17.—Thread gauges.

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.

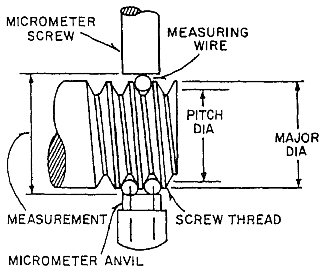


Figure 9-18.—Measuring threads using three wires.

The wire size you should use to measure the pitch diameter depends on the number of threads per inch. You will obtain the most accurate results 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 within a given range. Use a wire size as close as possible to the best wire size. To determine the wire sizes, use these formulas:

Best wire size = 0.57735 inch × pitch

Smallest wire size = 0.56 inch × pitch

Largest wire size = 0.90 inch × pitch

For example, the diameter of the best wire for measuring a thread that has 10 threads per inch is 0.0577 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 × pitch) + (3 × wire diameter)

$$M = PD - (0.86603 \times P) + (3 \times W)$$

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 thread? First, 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 inch. Use the maximum size (0.6832 inch) for this example. Next, 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 inch. As previously stated, the best wire size for measuring 10 TPI is 0.0577 inch, 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 $M = PD - (0.86603 \times P) + (3 \times W)$. Enter the collected data in the correct positions of the formula:

$$M = 0.6832 \text{ in.} - (0.86603 \text{ in.} \times 0.100 \text{ in.}) + (3 \times 0.0577 \text{ in.})$$

$$M = 0.6832 \text{ in.} - 0.086603 \text{ in.} + 0.1731 \text{ in.}$$

$$M = 0.769697 \text{ in.}$$

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 to is between 0.7482 and 0.7353 in.

CUTTING SCREW THREADS ON A LATHE

Screw threads are cut on the on the lathe by connecting the headstock spindle of the lathe with the lead screw through a series of gears to get a positive carriage feed. The lead screw is driven at the required speed in relation to the headstock spindle speed. You can arrange the gearing between the headstock spindle and lead screw so that you can cut any desired pitch. For example, if the lead screw has 8 threads per inch and you arrange the gears so 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-form, Acme, square, and so on.

MOUNTING WORK IN THE LATHE

When you mount work between lathe centers for cutting screw threads, be sure the lathe dog is securely attached before you start 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 you thread 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 you thread 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.

POSITIONING 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.

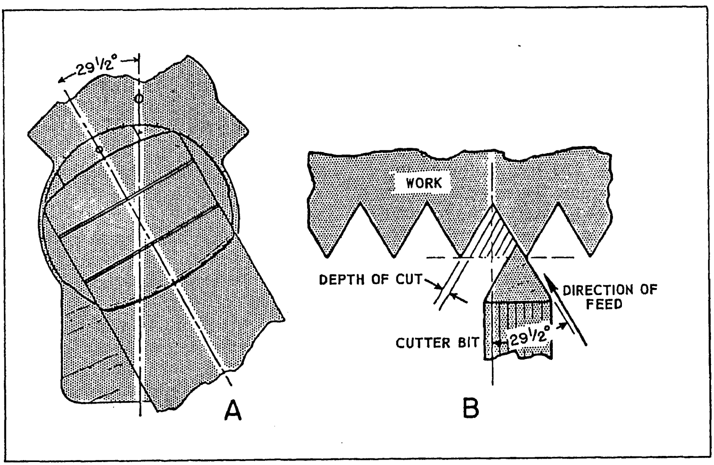
In cutting V-form threads and when maximum production is desired, it is customary to place the compound rest of the lathe at an angle of 29 1/2°, as shown in Part A of figure 9-19. When you set the compound rest in this position and use the compound rest screw to adjust the depth of cut, you remove most of the metal by using 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 keeps the thread from tearing. 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, and so forth, you must withdraw the thread-cutting tool quickly at the end of each cut. If you do not withdraw the tool 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



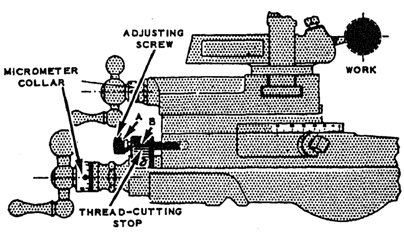
28.150X

(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. 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 you desire. 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. Make all adjustments for obtaining the desired depth of cut with the compound rest screw. Withdraw the tool at the end of each cut by turning the crossfeed screw to the right one turn, stopping at zero. You can then adjust the compound rest feed screw for any desired depth.



28.151X

Figure 9-20.—Adjustable thread-cutting stop mounted on

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 the half nut lever at the end of the cut by means of the threading lever. Use the threading dial (discussed in chapter 7 and illustrated in fig. 7-37) to determine when to engage the half-nuts so the cutting tool will follow 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 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.

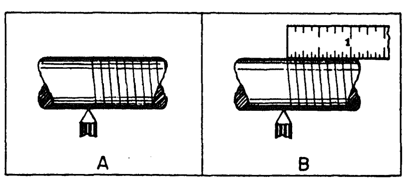
For all odd-numbered threads per inch, close the half-nuts at any numbered line on the dial.

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



28.152X

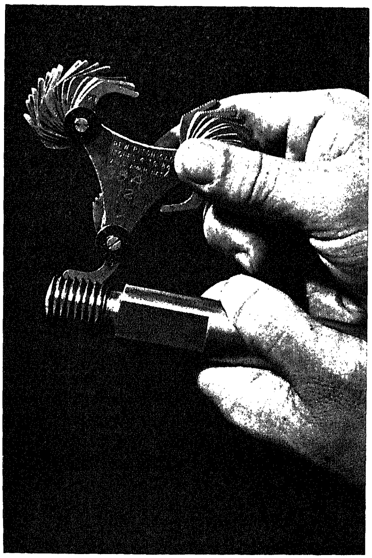
and the first inch mark. This will give the number of threads per inch.

It is quite difficult to accurately count fine pitches of screw threads. A screw pitch gauge, used as illustrated in figure 9-22, is very convenient for checking the finer screw threads. The gauge consists of a number of sheet metal plates in which are cut the exact forms 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.

LUBRICANTS FOR CUTTING THREADS

To produce a smooth thread in steel, use lard oil as a lubricant. If you do not use oil, 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.



28.153

Figure 9-22.—Screw pitch gauge.

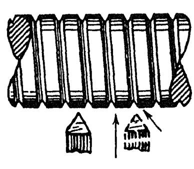
Apply the oil generously before each cut. A small paint brush is ideal for applying the oil when you cut external screw threads. Since lard oil is quite expensive, many machinists place a small tray or cup just below the cutting tool on the lathe cross slide to catch the surplus oil that drips off the 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 it will follow the original thread groove. To reset the tool, you may (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 in 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 gauge 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 near the groove into which you plan to put the tool, such as that shown by the solid tool in figure 9-23, stop the lathe without disengaging the thread mechanism.

To reset the cutting tool into the groove, you will probably use 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



28.154X

Figure 9-23.—Tool must be reset to original groove.

groove (phantom view of the 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 the 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.

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.

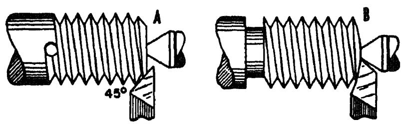
If it is inconvenient to use the compound rest for readjusting the threading tool, loosen the lathe dog (if used); turn the work so that the threading tool will match the groove, and tighten the lathe dog. If possible, however, avoid doing this.

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 reengage 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 it 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.



28.155X

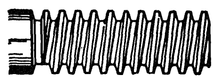
Figure 9-24.—Finishing the ·d of a threaded piece.

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 of axles, one end of a turnbuckle, or wherever an opposite thread is desired.

The directions for cutting a left-hand thread on a lathe are the same as those for cutting a right-hand thread, except that 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 into the workpiece. (See fig. 9-25). Such a groove



28.156X

Figure 9-25.—A left-hand screw thread.

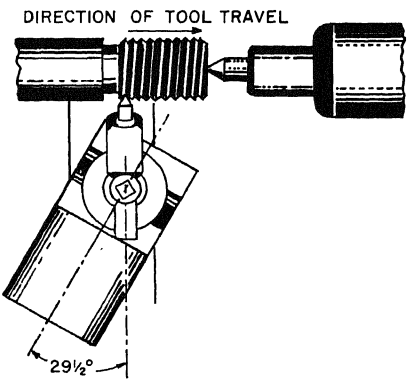


Figure 9-26.—Setup for left-hand external threads.

enables you to run the tool in for each pass, as you do for a right-hand thread.

Make the final check for both diameter and pitch of the thread, whether right-hand or left-hand, with the nut that is to be used, or with a ring thread gauge if one is available. The nut should fit snugly without play or shake but should not bind on the thread at any point.

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 the cylinder. Multiple threads are used when rapid movement of the nut or other attached parts is desired and when weakening of the thread must be avoided. A single thread having the same lead as a multiple thread would be very deep compared 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 as the number of threads increases. The general method for cutting multiple threads is about the same as for single screw threads, except that the lathe gearing must be based on the lead of the thread (number of single threads per inch), 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 rest parallel to the ways, using a faceplate, or using the change gear box mechanism.

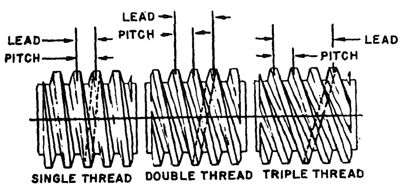


Figure 9-27.—Comparison of single and multiple-lead threads.

The use of the thread-chasing dial (fig. 9-28) 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 determine the possibility of using the thread-chasing dial, first find out if the lathe can be geared to cut a thread identical to one of the multiple threads. For example, if you want to cut 10 threads per inch, double threaded, divide the number of threads per inch (10) by the multiple (2) to get the number of single threads per inch (5). Then gear the lathe for 5 threads per inch.

To use the thread-chasing dial on a specific machine, refer to instructions usually found attached to the lathe apron. To cut 5 threads per inch, on most lathes, engage the half-nut at any numbered line on the dial, such as points 1 and 2 shown in figure 9-28. The second groove of a double thread lies in the middle of the flat surface between the grooves of the first thread. Engage the half-nut to begin cutting the second thread when an unnumbered line passes the index mark, as shown in figure 9-28. To ensure that you cut each thread to the same depth, engage the half-nut first at one of the numbered positions and cut in the first groove. Then engage the half nut at an unnumbered position so that alternate

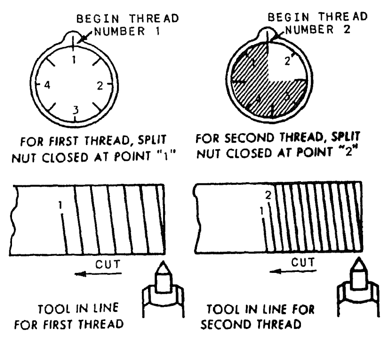


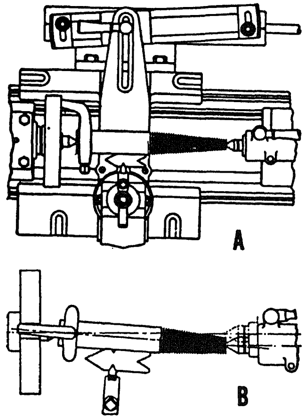
Figure 9-28.—Cutting multiple threads using the thread-chasing dial.

cuts bring both thread grooves down to size together. To cut a multiple thread with an even number of threads, first use the thread-chasing dial to cut the first thread. Then use one of the other multiple thread cutting procedures to cut the second thread.

Cutting of multiple threads by positioning the compound rest parallel to the ways should be limited to square and Acme threads. To use this method, set the compound rest parallel to the ways of the lathe and cut the first thread to the finished size. Then feed the compound rest and tool forward, parallel to the thread axis a distance equal to the pitch of the thread and cut the next thread.

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 between centers so the tail of the dog is 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, and so on. The number of multiples you can cut by this method depends on the number of equally spaced slots there are in the drive plate. There are special drive or index plates available, 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 gear or the spindle gear from the gear train in the end of the lathe after you cut a thread groove. Then turn the work and the 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. Carry the mark onto the driven gear, in this case the reversing gear. Also mark the tooth diametrically opposite the marked spindle gear tooth (the 20th tooth of the 40-tooth gear). Count the tooth next to the marked tooth as tooth number one. Then disengage the gears by placing the tumbler (reversing) gears in the neutral position, turn the spindle one-half revolution or 20 teeth on the spindle gear, and reengage the gear train. You may index the stud gear as well as the spindle gear. If the ratio between the spindle and stud gears is



28.158X

Figure 9-30.—Cutting thread on tapered work.

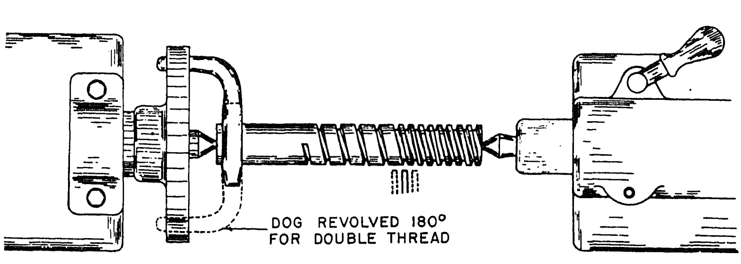


Figure 9-29.—Use of face plate.

not 1 to 1, you will have to give the stud gear a proportional turn, depending upon the gearing ratio. 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 lathe machines have a sliding sector gear that you can readily insert into or remove from 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

Use the taper attachment when you cut a thread on tapered work. If your lathe does not have a taper attachment, 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 gauge 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 gauge on the straight part of the work, NOT on the tapered section. This is very important.

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.

- Do NOT use a turret lathe that you are not authorized and fully qualified to operate.

- 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.

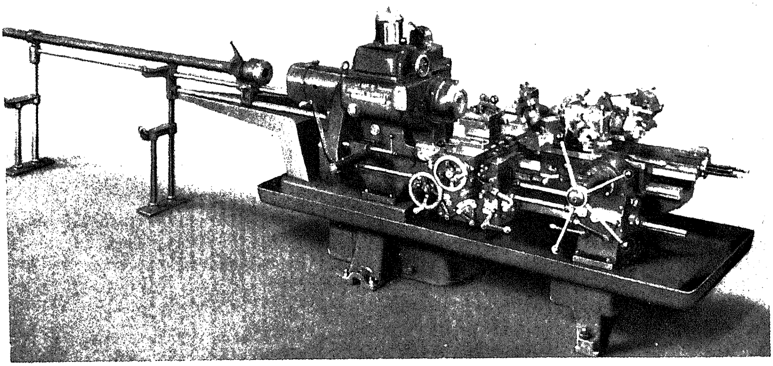
- NEVER completely trust the automatic stops on a turret lathe. Be alert at all times to the progress and movement of the cutting tool(s).

- NEVER exceed the recommended depth of cut, cutting speeds, and feeds.

- Before starting a vertical turret lathe, always be alert for tools, clamping devices, or other materials adrift on the lathe table.

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 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.

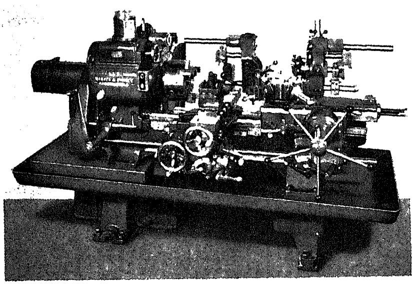


28.159

Figure 10-1.—Bar machine.

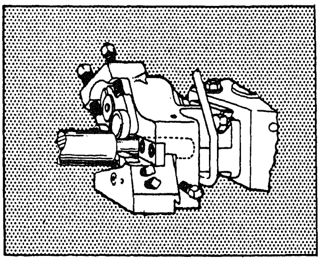
CLASSIFICATION OF HORIZONTAL TURRET LATHES

Figures 10-1 and 10-2 show two types 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

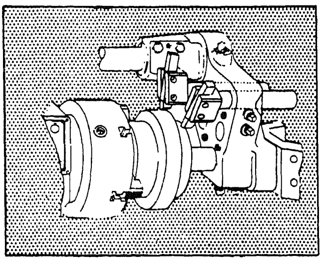


28.160

Figure 10-2.—Chucking machine.



A–BAR TURNING SETUP



B–CHUCKING SETUP

Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.161X

Figure 10-3.—Hexagonal turret turning tool setups.

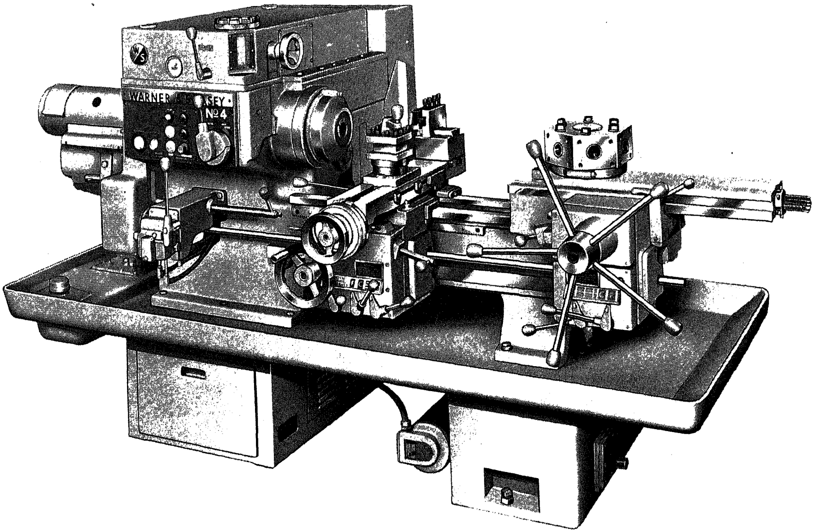


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.342X

Figure 10-4.—Ram type bar machine.

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 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, on 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 longitudinal machining with hexagonal turret-held tools.

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

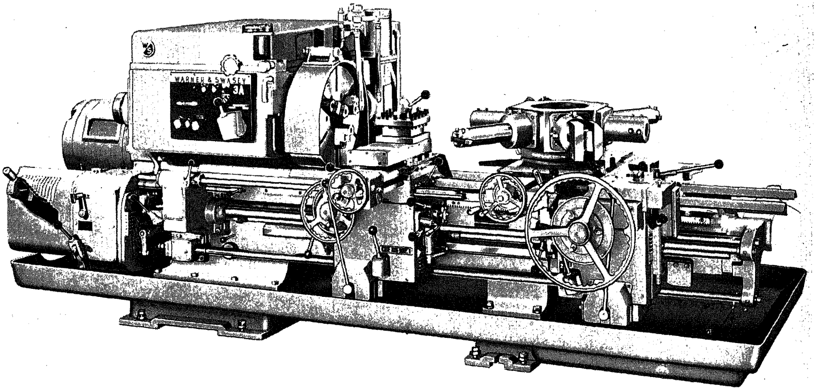


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.343X

Figure 10-5.—Saddle type chucking machine.

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 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 preselector 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

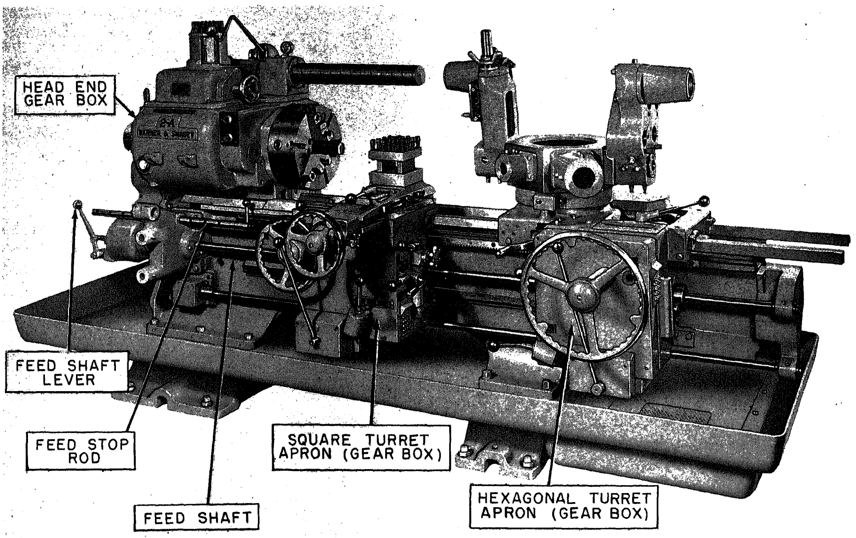


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.165X

Figure 10-6.—Saddle type turret lathe feed train.

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.

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

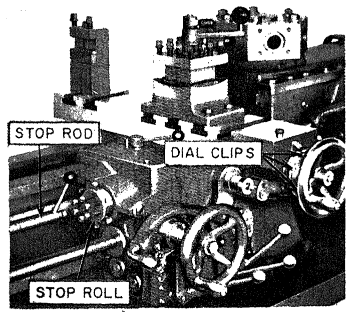


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.166X

Figure 10-7.—Typical longitudinal feed stop arrangement for cross slide.

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 other 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 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 the 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.

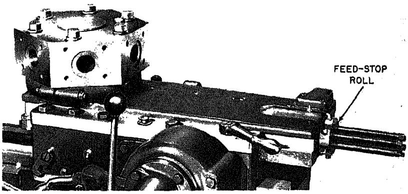


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.168X

Figure 10-9.—Hexagonal turret feed-stop roll on a ram type machine.

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

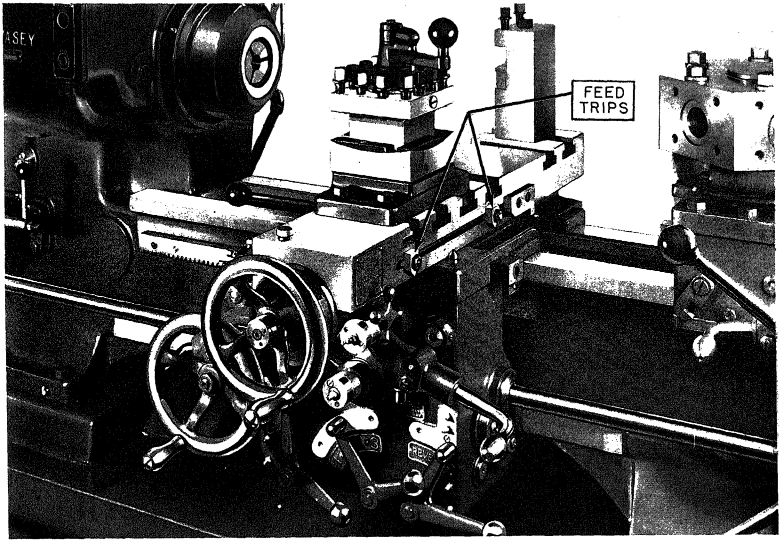


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

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 other end of the roll, where they will be in a convenient position. Selected 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 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.

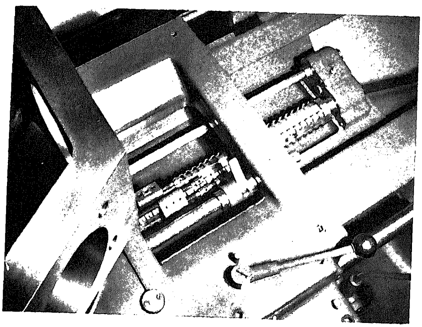


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.169X

Figure 10-10.—Hexagonal turret feed stops on a saddle type machine.

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 that on an engine lathe. It is bolted to the carriage and engaged 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

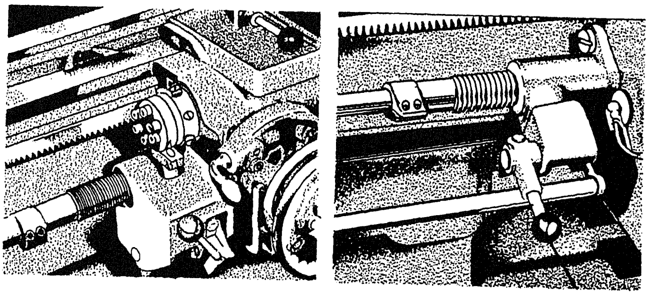


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.344X

lead screw threading attachment. 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 you can tell at a glance whether oil is being circulated to the areas where it is required.

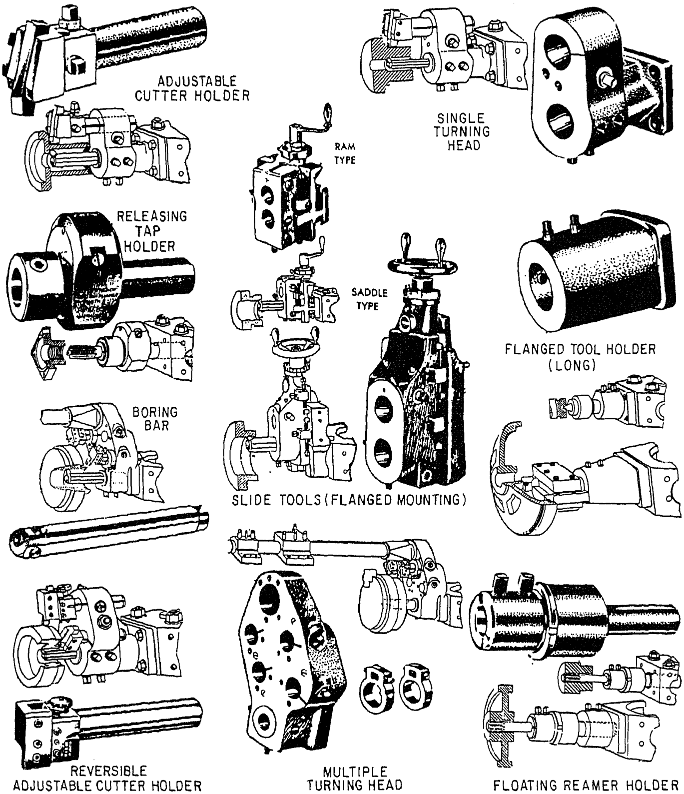


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

Figure 10-12.—Turret lathe chucking tools. 28.345X

Whenever you clean a lathe, use a cloth or a brush to remove chips. **DO NOT** use compressed air. 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 used on the hexagonal turret. In the following paragraphs which describe workholding devices, grinding and setting cutters, and various machining procedures, we do not specify 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 comment 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.

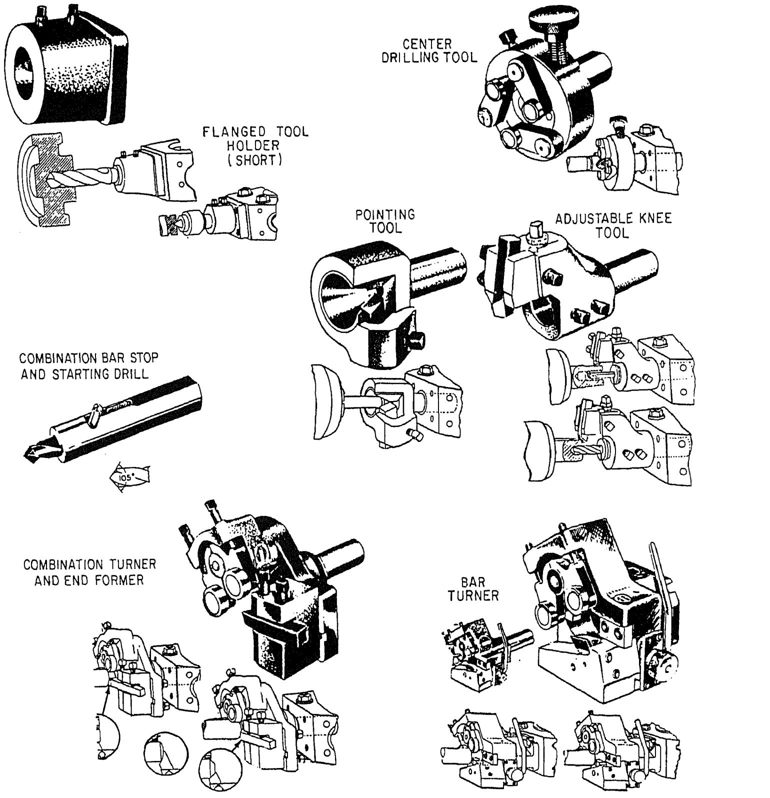


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

Figure 10-13.—Turret lathe bar tools. 28.346X

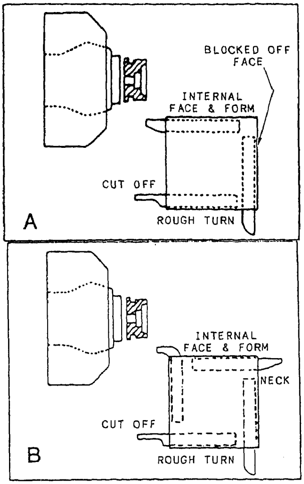


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.171X

Figure 10-14.—Square turret tool positions.

As a good turret lathe operator, your aim should be to tool and operate the machine 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.

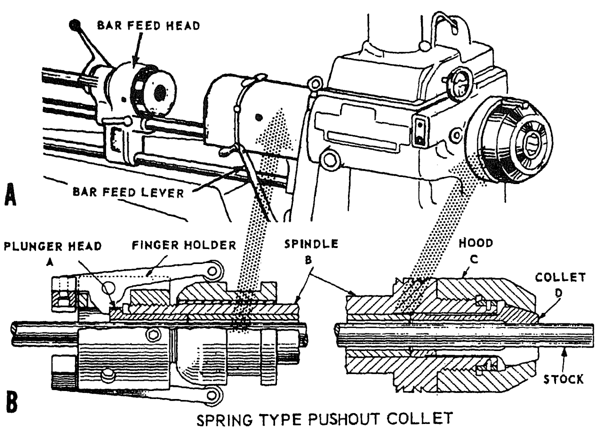


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.172X

- 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.

Holding the Work

Horizontal turret lathes are generally used for turning out duplicate machine parts rapidly in quantity. The workholding device must allow you to quickly place stock material in the machine. Moreover, 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, 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 feed head (fig. 10-15A) to advance the stock, you simultaneously loosen 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 collet D into the taper of the hood C, causing the collet to grip the stock firmly. Your one simple movement automatically sets the stock material into position for machining.

There are several variations of the spring-type collet, but they all depend on 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 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 of arbor, 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 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) universal chucks of the geared scroll, geared screw, or box type that have three jaws that move at the same time; (2) independent chucks, that have jaws that operate independently; and (3) combination chucks, that have jaws that may be operated either independently, or as a group.

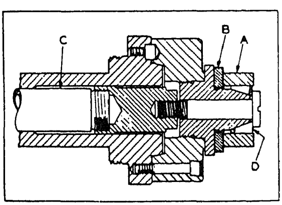


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

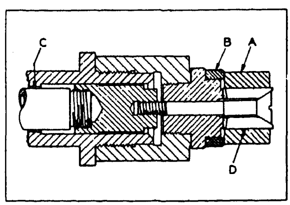


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

The 2-jaw chuck is used mostly for holding small or irregularly shaped work. 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; but it can be adapted to hold irregularly shaped work.

Figure 10-18 shows a 4-jaw combination chuck that 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 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 using 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 (indicated by the arrow).

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, the 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 you grind turret lathe tools are

- 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 a cutter wet, keep it 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

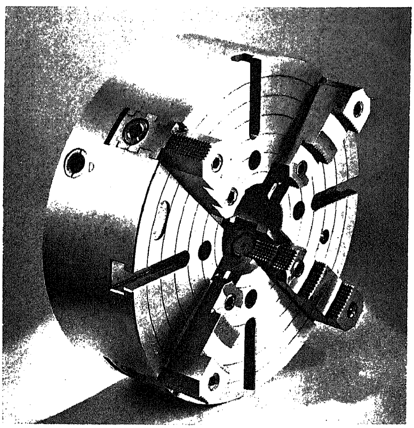


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.175X

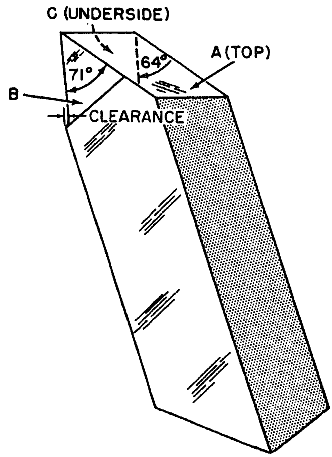


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.176X

shuold 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 grind a carbide-tipped cutter, always be sure that the pressure of the grinding is 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 of 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 surface 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 different because of the difference in tool point location.

CONTROLLING CHIPS.—You can control chips 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 that 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).

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 the 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

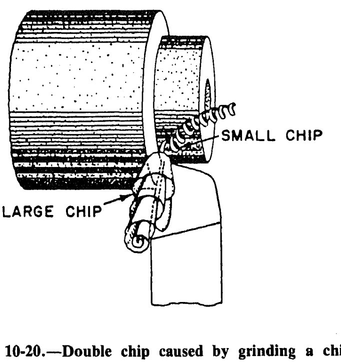


Figure 10-20.—Double chip caused by grinding a chip breaker groove too close to the cutting edge.

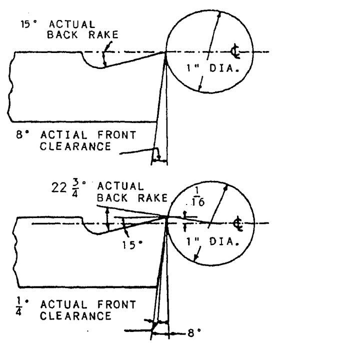


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.178X

Figure 10-21.—Keep cutters on center.

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 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 the 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.

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

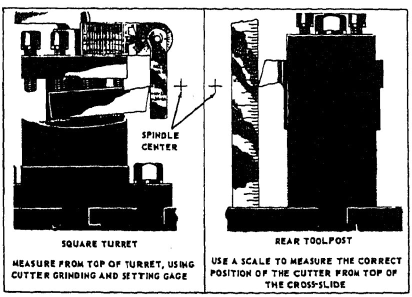


Figure 10-22.—Setting square turret and "reach over" toolpost cutters on center.

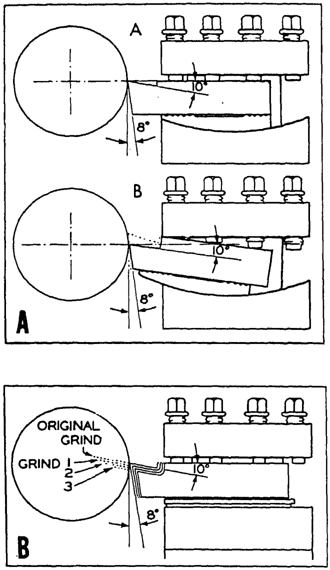


Figure 10-23.—A. Use of rockers. B. Use of shims.

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. This will allow you to account for 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 provides 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 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 position of the cutter into radial line with the center of the turning head, which is in alignment with the center of the spindle.

SETTING BAR TURNER CUTTERS.—Bar turners are held on the hexagonal turret and combine in one unit a cutter holder and a backrest that travel with the cutter and support 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 rollers remaining in fixed position. The universal bar turner is 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.

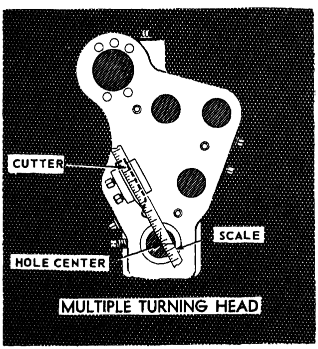


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.181X

Figure 10-24.—Setting an overhead turning cutter on center.

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

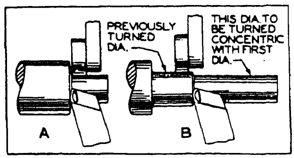


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.182X

Figure 10-25.—Rollers. A. Behind cutter. B. Ahead of cutter.

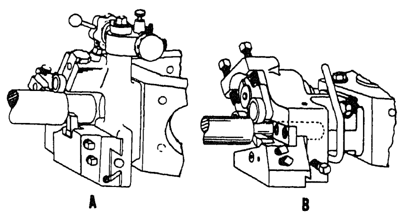


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.183X

Figure 10-26.—A. Universal bar turner. B. Single bar turner.

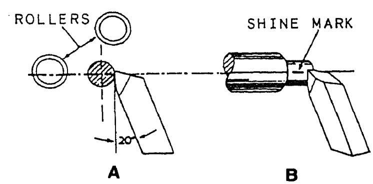


Figure 10-27.—Rubbing a shine mark to establish a center. A. Roll jaws out of position. B. Shine mark on the turned portion.

and 20° from the perpendicular bisector, 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 figure 10-27B.

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. Position the rollers endwise and adjust them 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 them 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 of the workpiece will be accurate to ±0.001 inch.

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.

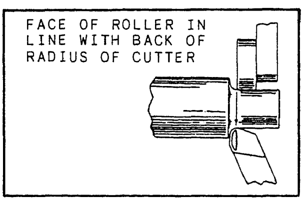


Figure 10-28.—Rollers aligned with the cutter.

- 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 than 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 be cutting at well below its usual 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 to provide greater accuracy by keeping the work from overheating and becoming distorted. Figure 10-29 shows the correct and incorrect ways to apply 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

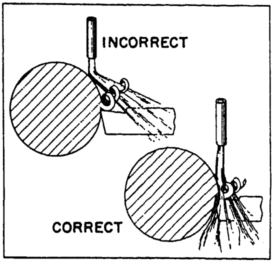


Figure 10-29.—Correct and incorrect ways to apply coolant.

STAINLESS STEEL—Soluble oil 1 to 5 ratio, or mineral lard oil

ALUMINUM—Soluble oil 1 to 25 ratio, or dry

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 just 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 STUB BORING BAR is held either at a right angle 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. 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 the 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 to adjust 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 to 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 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.

Figure 10-30 shows how to center a vertical slide tool-held boring cutter.

Forming

One of the fastest methods of producing a finished diameter or shape is by using a cutter with a cutting edge that matches the shape to be machined. This procedure is known as 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 made in the shop from forged blanks and ordinarily are mounted directly in the square turret or toolpost. These cutters are the least expensive to make. They have, however, the shortest production life.

DOVETAIL FORMING CUTTERS are cutters that may be either bought or made. They are attached by dovetails to 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 production life, are more easily set up, maintain their form after grinding, are more rigid, and can be operated under heavier feeds.

CIRCULAR FORMING CUTTERS (fig. 10-31) 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 a new cutting edge, move the cutter to a new cutting position by rotating the cutter about its axis.

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 way to cut 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

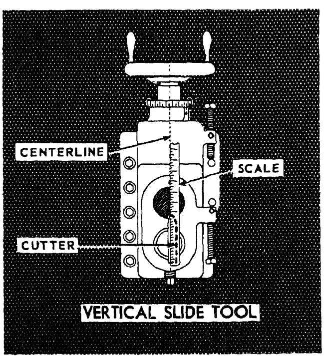


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.187X

Figure 10-30.—Setting a boring cutter on center.

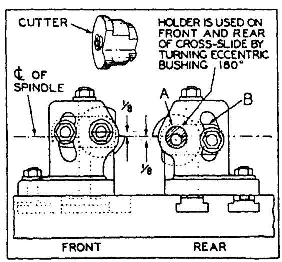


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.188X

Figure 10-31.—Circular forming cutter diagram.

the tap or die to follow the natural lead of the thread. 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.

Solid adjustable taps and dies should be used in place of collapsing taps and self-opening die heads only when lathe speed is low and when time required for a backing out is not important.

Collapsing taps (fig. 10-32) are used for internal threading. They are time-savers 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. The floating type die head should be used for heavy duty turret lathe work, for fine pitch threading, and for finishing rough-cut threads.

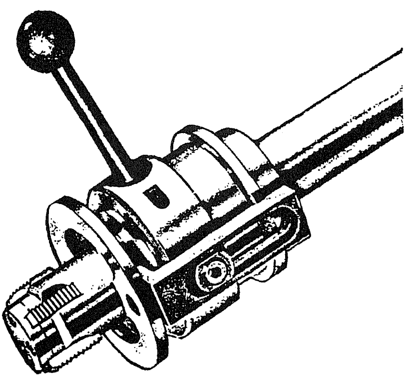


Figure 10-32.—Universal collapsing tap.

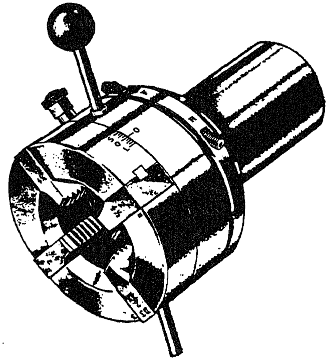


Figure 10-33.—Pull-off trip self-opening die head.

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

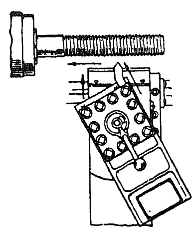


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.189X

Figure 10-34.—Compound cross-slide angular feed-in for thread cutting.

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.

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 types 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 under the following conditions:

1. The work is either self-supporting or is supported by a center rest so that chatter is prevented.

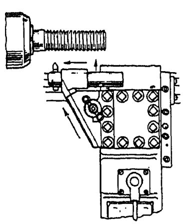


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.190X
Figure 10-35.—Angular feed-in with adjustable threading toolholder.

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 using the graduated dial and then can control the angle of taper accurately by using the taper guide bar.

Taper attachments are provided for the cross slide of most turret lathes, both ram and saddle type. These attachments can be quickly set to produce either internal or external tapers. They

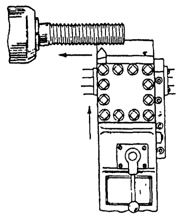
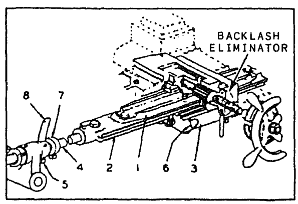


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.191X
Figure 10-36.—Straight-in feeding method of threading.



1. GUIDE PLATE
2. BASE PLATE
3. CARRIAGE PLATE
4. EXTENSION ROD
5. SETSCREW
6. BINDER SCREW
7. STOP COLLAR
8. LATCH

Photo courtesy of the Warner & Swasey Company, Solon, Ohio

28.192X
Figure 10-37.—Detail of a cross-slide taper attachment for a saddle-type machine.

do not interfere with normal operation when not in use. Most taper attachments are movable and can be quickly placed 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.

The guide plate (1) pivots on the base plate (2), which slides into carriage plate (3). When you plan to use the attachment, clamp the extension rod (4) to the machine with the setscrew (5), and loosen the binder screw (6). 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 accurate sizes when you use a taper attachment with other tools in a setup, remember these three things; (1) you must locate the attachment in the same position in relation to the cross slide each time you use it, (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 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

Regardless of the job, your aim as a good turret lathe operator is to tool up the machine and operate it so the job can be turned out as rapidly and as accurately as possible. The following examples show you how.

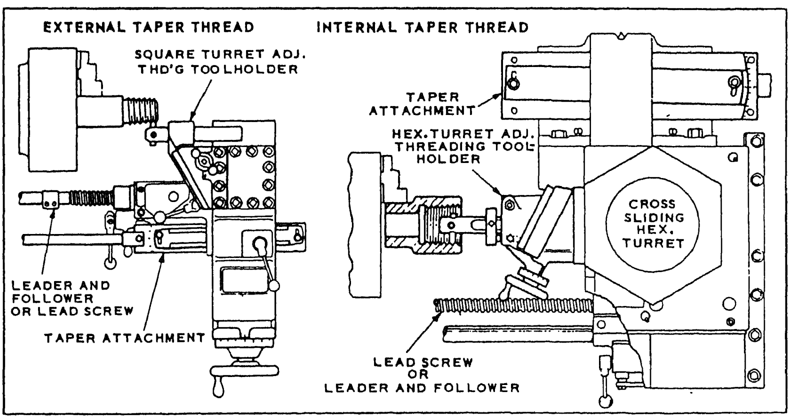


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

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. These 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.

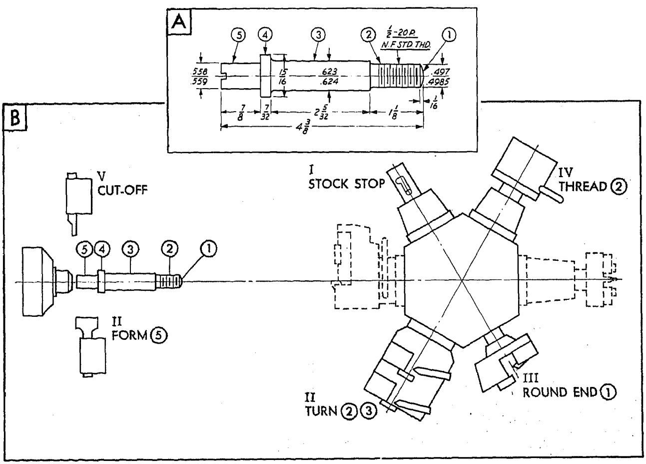


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

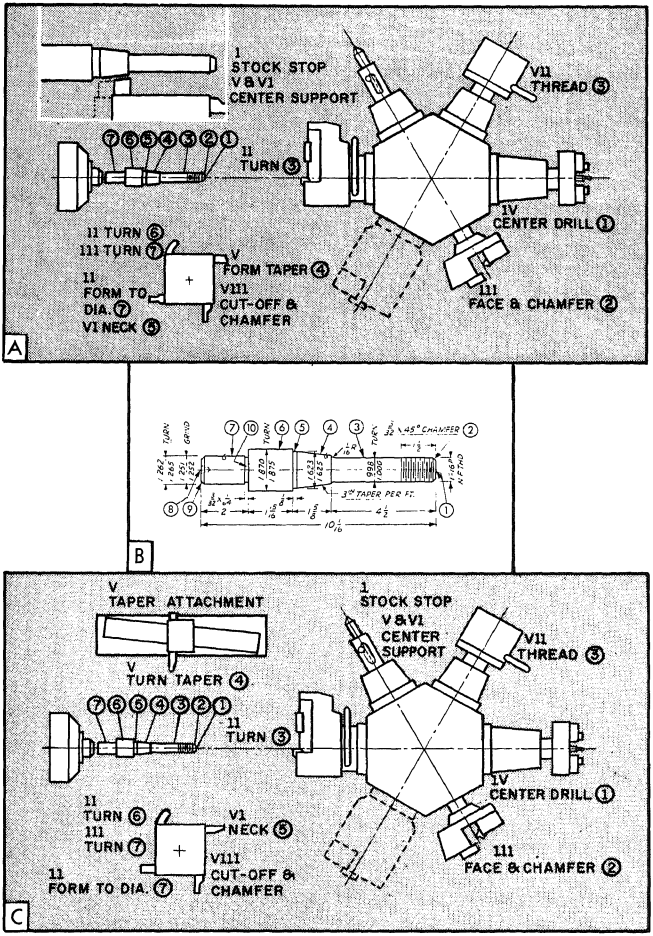


Photo courtesy of the Warner & Swasey Company, Solon, Ohio

126.11X

Figure 10-40.—A. Tooling setup for a taper stud—small lot production. B. A taper stud. C. Tooling setup for a taper stud—medium lot production.

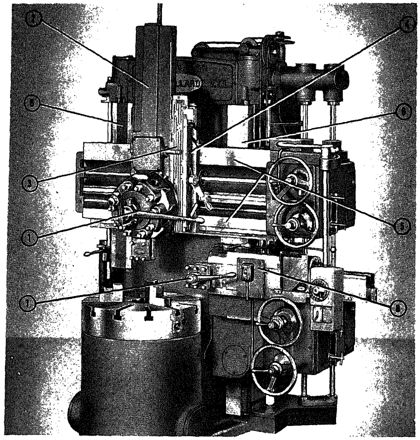
Figure 10-41 shows a simple setup for the second operation of the taper stud. The setup is the same for producing either a small or a medium size quantity.

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.

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



(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.

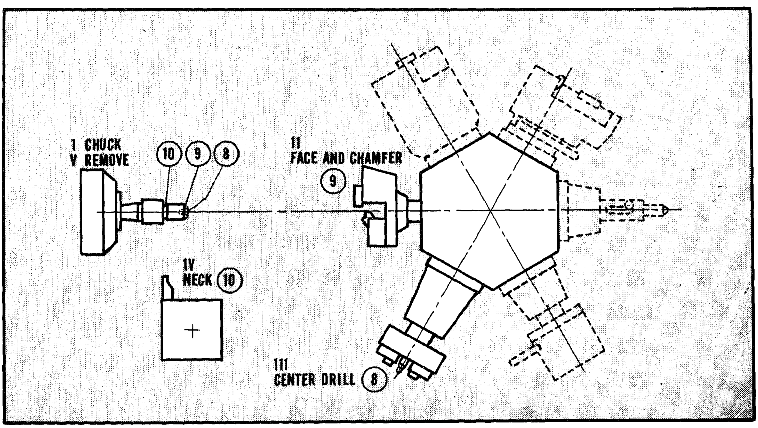
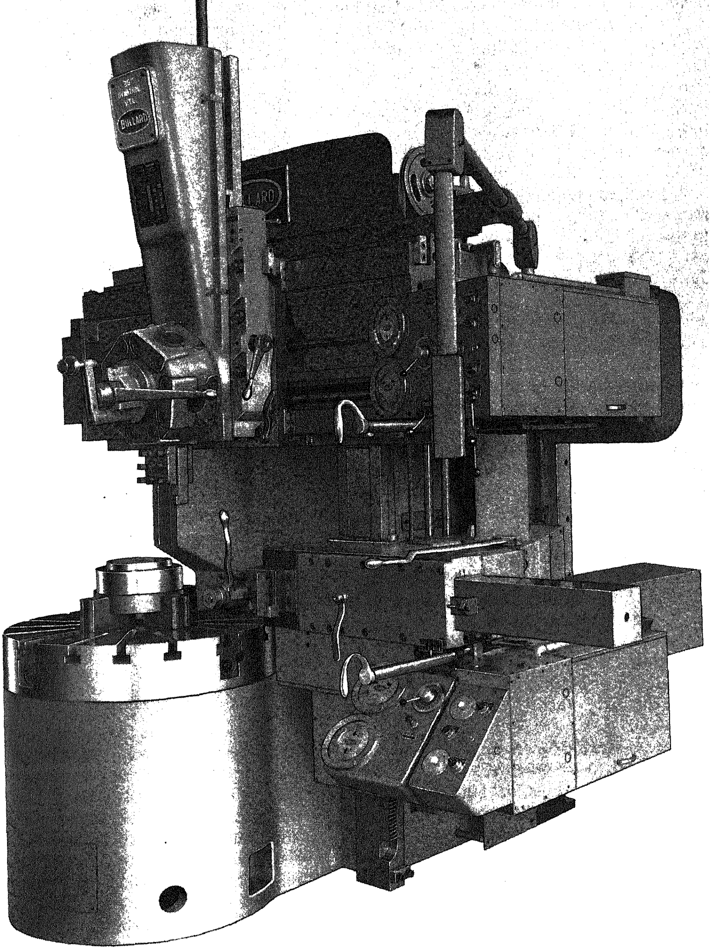


Photo courtesy of the Warner & Swasey Company, Solon, Ohio



28.349X

Figure 10-43.—A 36-inch vertical turret lathe.

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 you to feed main turret tools either vertically or horizontally, as compared to one direction on the horizontal turret lathe. Also, you can cut tapers 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 that 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, 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 setup and completion.

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 oddly 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 vertical turret lathes shown in figures 10-42 and 10-43 is in the method used to position the cutter to the work. The lathe in figure 10-42 has a handwheel for manually positioning the work. The lathe in figure 10-43 uses 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 of up to 32 threads per inch with a single point tool. 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 this 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, 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, 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 you machine 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

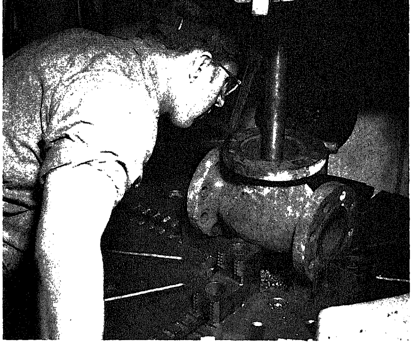


Figure 10-44.—Refacing a valve seat in a vertical turret lathe.

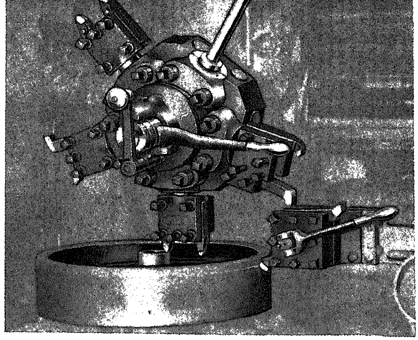


Figure 10-45.—Double tooling.

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 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 having to use special attachments. To machine angles greater than 30° and less than 60° from the vertical, engage both the horizontal feed and the vertical feed simultaneously and swivel the head. Determine the angle to which you swivel the head 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

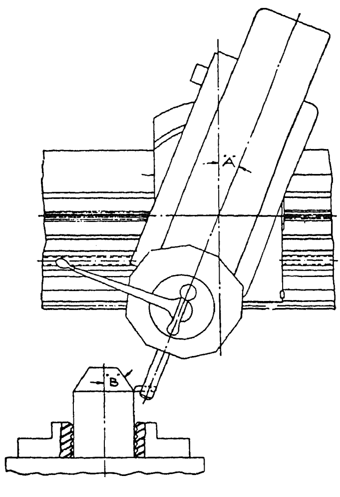


Figure 10-46.—Head setting for 30° to 45° angles.

determining the proper angle is $A = 90° - 2B°$. A sample problem from figure 10-46 follows:

Formula $A + 90° - 2B°$

Example $B = 35°$

Therefore $A = 90° - (2 \times 35°)$

$A = 90° - 70°$

ANGLE $A = 20°$

For angles between 46° and 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 $A = (2 \times 56°) - 90°$

$A = 112° - 90°$

ANGLE $A = 22°$

Whenever you turn a taper by using the main turret slide swiveling method, use great care to set the slide in a true vertical position after you complete the taper work and before you use 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.

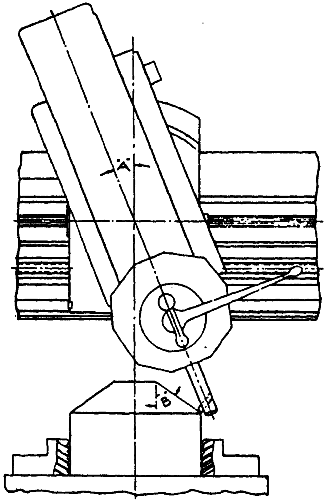


Figure 10-47.—Head setting for 45° to 60° angles.

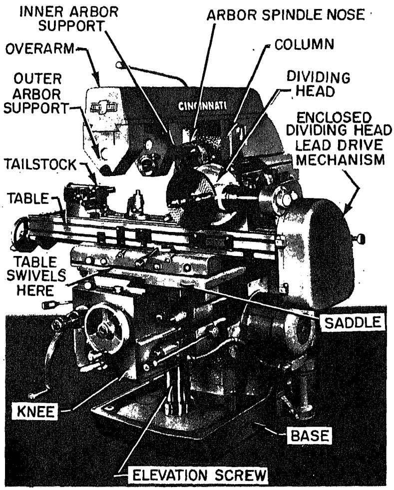
Unless you are alert to this, you may inadvertently cut a dimension undersize before you are aware of the error.

Still another way to cut 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.

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 a typical bed-type mill. 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 the classes with which you will be concerned are discussed in this chapter.



28.362X

Figure 11-1.—Universal milling machine.

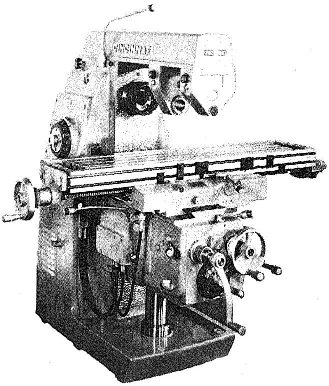
You must be able to set up the milling machine to machine flat, angular, and formed surfaces. Included in these jobs are the milling of keyways, hexagonal and square heads on nuts and bolts, T-slots and dovetails, and spur gear teeth. To set up a milling machine, you must compute feeds and speeds, select 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.

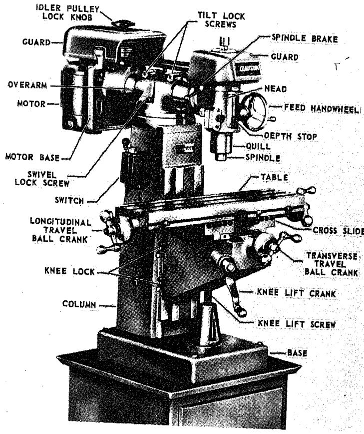
KNEE AND COLUMN MILLING MACHINES

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 a right angle to 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.



28.365X
Figure 11-2.—Plain Milling Machine.



28.364X
Figure 11-4.—Small vertical milling machine.

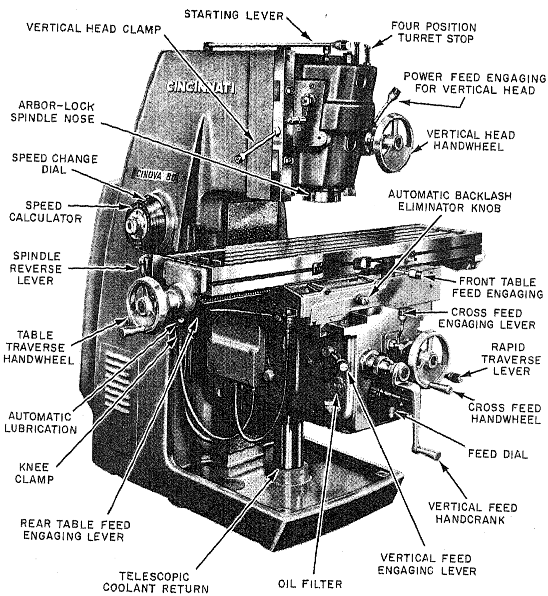


Figure 11-3.—Vertical spindle milling machine. 28.363X

a few of the features found on the other machines. You can move the table in three directions: longitudinally (at a right angle 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 and is made possible by the machine's rigid construction.

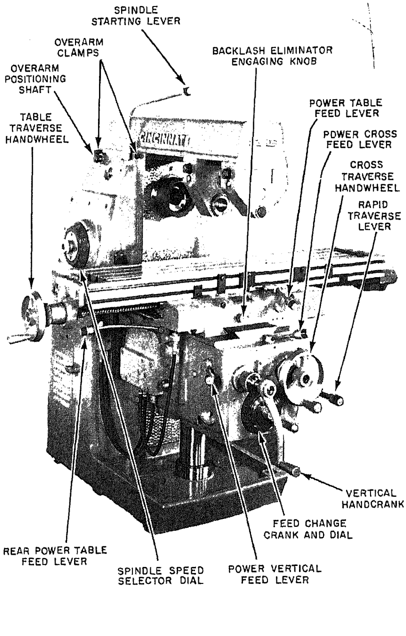
The VERTICAL SPINDLE MILLING MACHINE (fig. 11-3) has the spindle in a vertical position and at a right angle to the surface of the table. The spindle has a vertical movement, and the table can be moved vertically, longitudinally, and transversely. Movement of both the spindle and the table can be controlled manually or by power. The vertical-spindle milling machine can be used for face milling, profiling, die sinking,

Various small vertical spindle milling machines (fig.11-4) are also available for light, precision milling operations.

MAJOR COMPONENTS

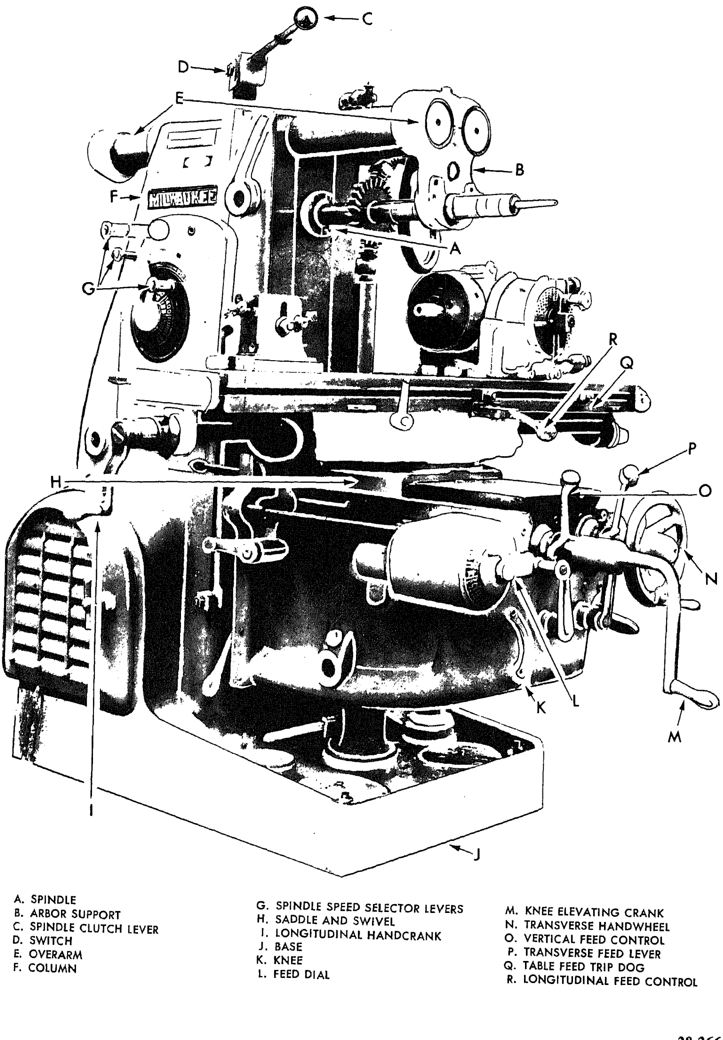
You must know the name and purpose of each of the main parts of a milling machine to understand the operations discussed later in this chapter. Keep in mind that although we are discussing a knee and a column milling machine you can apply most of the information to the other types.

Figure 11-5, which illustrates a plain knee and column milling machine, and figure 11-6, which illustrates a universal knee and column milling



28.365X

Figure 11-5.—Plain milling machine, showing operation controls.



28.366

Figure 11-6.—Universal knee and column milling machine with horizontal spindle.

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 you perform any machining operation that requires a coolant.

KNEE: The knee is the casting that supports the table and the saddle. The feed change gearing is enclosed within the knee. It is supported and can be adjusted by turning 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.

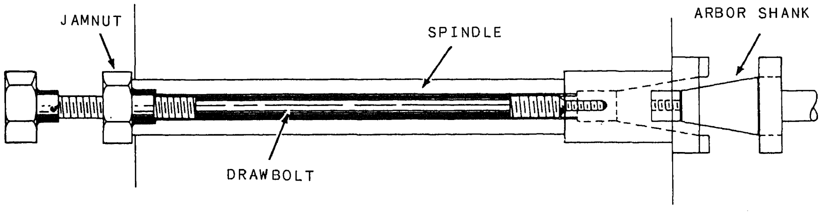
SADDLE and SWIVEL TABLE: The saddle slides on a horizontal dovetail (which is parallel to the axis of the spindle) on the knee. The swivel table (on universal machines only) is attached to the saddle and can be swiveled approximately 45° in either direction.

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 11-5, by positioning the feed selection levers as indicated on the feed selection plate. On machines such as the one in figure 11-6, 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 to position or align the work.

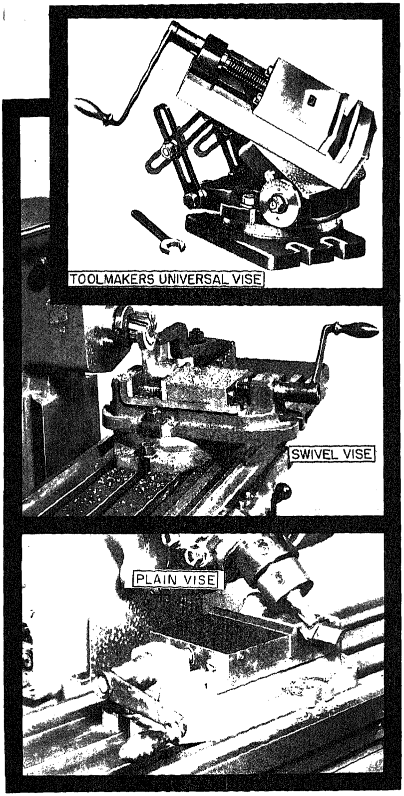
NOTE: For safety reasons, you must exercise extreme caution whenever you use the rapid traverse controls.

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 of 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 11-7. 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 may be a single casting that slides in dovetail ways on the top of the column (fig. 11-6) or it may consist of one or two cylindrical bars that slide through holes in the column, as shown in figure 11-6. To position the overarm on some machines, you 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 on an overarm such as the one shown as B, in figure 11-6, 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.



BROWN & SHARPE Manufacturing Company, North Kingstown, RI

28.199X

Figure 11-8.—Milling machine vises.

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 at any place you want on the overarm. Small arbor supports give additional clearance below the arbor supports when you are using small diameter cutters. However, small arbor supports can provide support only at the extreme end of the arbor. For this reason they are not recommended for general use. Large arbor supports can provide support near the cutter, if necessary.

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 is concerned. 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

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 used with milling machines in Navy ships includes workholding devices, spindle attachments, cutters, arbors, and any special tools needed for setting up the machines for milling. This equipment allows you to hold and cut the great variety of milling jobs you will encounter in Navy repair work.

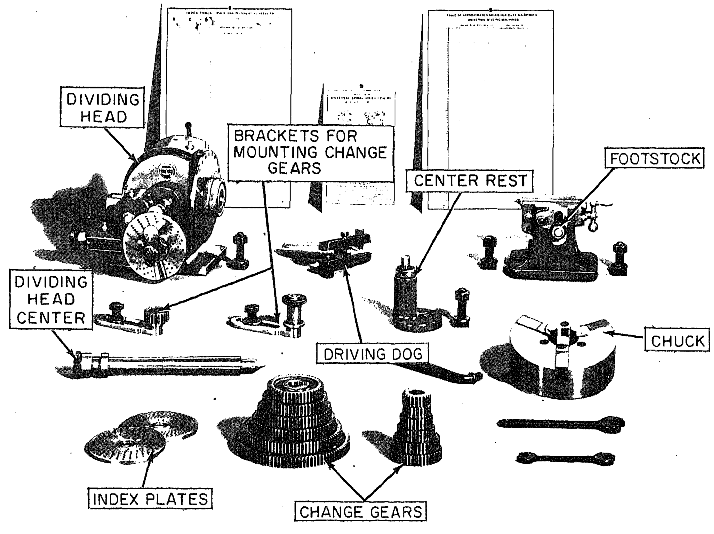
WORKHOLDING DEVICES

The following workholding devices are the ones that you will probably use most frequently.

vise provides the most support for a rigid workpiece. The swivel vise is similar to the flanged vise, but the setup is less rigid because the workpiece can be swiveled in a horizontal plane to any required angle. The toolmaker's universal vise provides the least rigid support because it is designed to set up the workpiece at a complex angle in relation to the axis of the spindle and to the surface of the table.

INDEXING EQUIPMENT

Indexing equipment (fig. 11-9) is used to hold and turn the workpiece so that a number of accurately spaced cuts can be made (gear teeth for example). The workpiece may be held in a chuck or a collet, attached to the dividing head spindle, or held between a live center in the dividing



BROWN & SHARPE Manufacturing Company, North Kingstown, RI

28.200X

Figure 11-9.—Indexing equipment.

index head and a dead center in the footstock. The center of the footstock can be raised or lowered for setting up tapered workpieces. The center rest can be used to support long slender work.

Dividing Head

The internal components of the dividing head are shown in figure 11-10. The ratio between the worm and the 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, which you can use to gauge partial turns of the worm shaft and to 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 a rotating shaft and you can adjust the crankpin radially for use in any circle of holes. You can also set the sector arms as a guide to span any number of holes in the index plate to provide a guide for rotating the index crank for partial turns. To rotate the workpiece, you can turn the dividing head spindle either 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 slightly below horizontal to slightly past vertical. As mentioned previously, most index heads have a 40:1 ratio. One well-known exception has a 5 to 1 ratio (see fig. 11-11). This ratio is made possible by a 5 to 1 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. Differential indexing (discussed later in this chapter) can be done on the 5 to 1 ratio dividing head by using a differential indexing attachment.

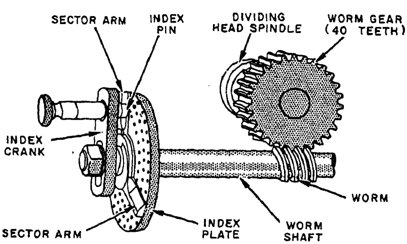


Figure 11-10.—Dividing head mechanism.

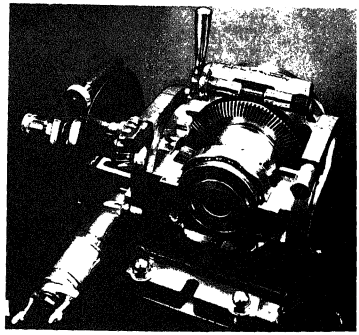
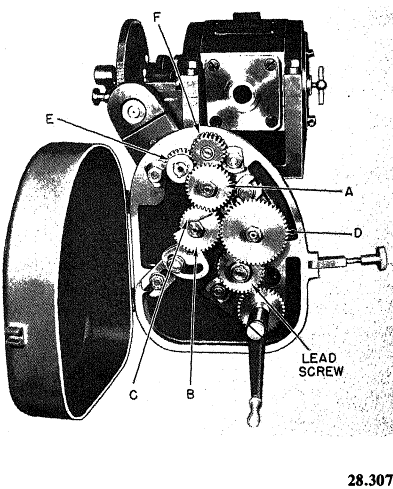


Photo courtesy of Kearney & Trecker Corporation, Milwaukee, Wis.

28.368X

Figure 11-11.—Universal spiral dividing head with a 5 to 1 ratio between the spindle and the index crank.



28.307X

Figure 11-12.—Enclosed driving mechanism.

milling machine by a driving mechanism to turn 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 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 11-12 illustrates the gearing arrangement used on most milling machines. The gears are marked as follows:

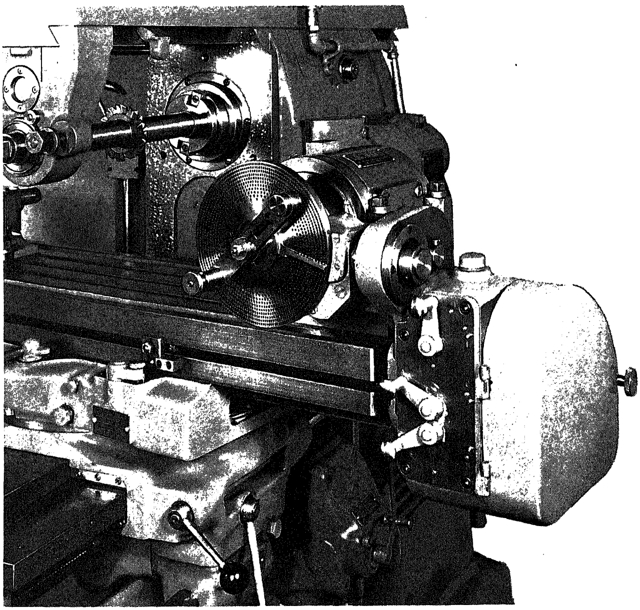
A = Gear on the worm shaft (driven)

B = First gear on the idler stud (driving)

E and F = Idler gears

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. 11-13). 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



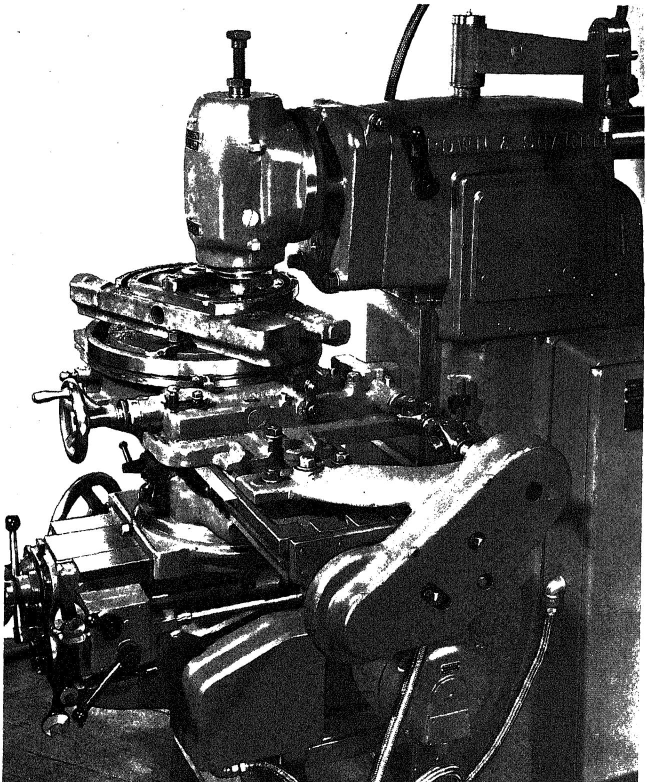
BROWN & SHARPE Manufacturing Company, North Kingstown, RI

126.27X

Figure 11-13.—The long and short lead attachment.

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 (fig. 11-14), 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. 11-14). An 80 to 1 worm and gear drive contained in the rotary table and index plate arrangement makes this device



BROWN & SHARPE Manufacturing Company, North Kingstown, RI

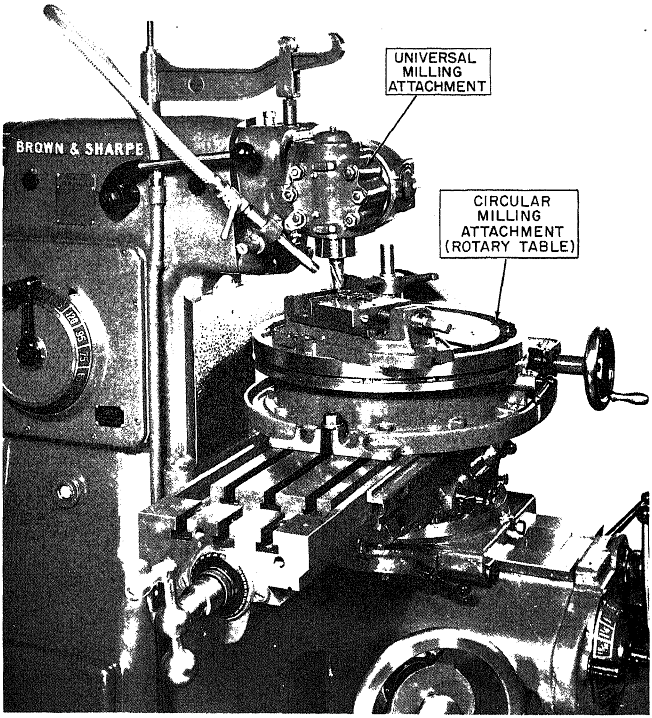
SPECIAL ATTACHMENTS

The universal milling (head) attachment, shown in figure 11-15, 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

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 of machine frequently is not available. Consequently, the machinist must devise other means for cutting slots. The slotting attachment



28.202X

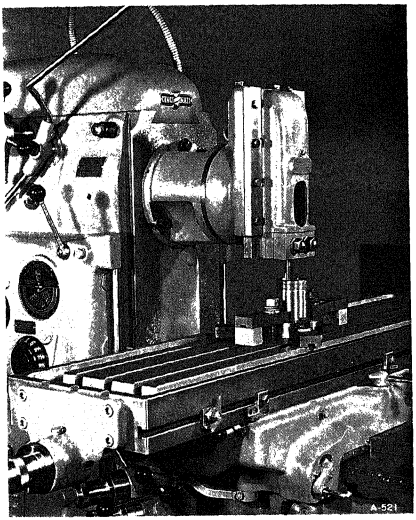
Figure 11-15.—Circular milling attachments (rotary table) and universal (head) attachment.

in figure 11-16, when mounted on the column and the 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 to from 0 to 4 inches.

INDEXING THE WORK

Indexing is done by the direct, plain, compound, or differential method. The direct and plain methods are the most commonly used; the compound and differential methods are used only when the job cannot be done by plain or direct indexing.



BROWN & SHARPE Manufacturing Company, North Kingstown, RI

28.369X

Figure 11-16.—Slotting a bushing using a slotting attachment.

DIRECT INDEXING

Direct indexing, sometimes referred to as rapid indexing, is the simplest method of indexing. Figure 11-17 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 pin, which is spring-loaded and moved in and out by a small lever. Rapid indexing requires that the worm and the worm wheel be disengaged so that the spindle can be moved by hand. Numbers that can be divided into 24 can be indexed in this manner. Rapid indexing is used when a large number of duplicate parts are to be milled.

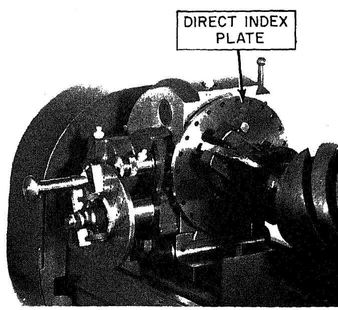
To find the number of holes to move the index plate, divide 24 by the number of divisions required.

Number of holes to move = 24/N where N = required number of divisions

Example: Indexing for a hexagon head bolt: because a hexagon head has six flats,

$$\frac{24}{N} = \frac{24}{6} = 4 \text{ holes}$$

IN ANY INDEXING OPERATION ALWAYS START COUNTING FROM THE HOLE ADJACENT TO THE CRANKPIN. During heavy cutting operations, clamp the spindle by the clamp screw to relieve strain on the index pin.



BROWN & SHARPE Manufacturing Company, North Kingstown, RI

28.209X

Figure 11-17.—Direct index plate.

PLAIN INDEXING

Plain indexing, or simple indexing, is used when a circle must be divided into more parts than is possible by rapid indexing. Simple 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 the worm wheel is 40 to 1 (40:1). One turn of the index crank turns the index head spindle 1/40 of a complete turn. Therefore, forty turns of the index crank are required to revolve the spindle chuck and the job one complete turn. To determine the number of turns or fractional parts of a turn of the index crank necessary to 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 required

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 using the indexing

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 circles of holes. Do this by multiplying or dividing the numerator and the denominator by the same number. For example, to index for the machining of a hexagon (N = 6):

$$\frac{40}{6} = \frac{40}{6} \times \frac{3}{3} = \frac{120}{18} = 6\frac{12}{18} = 6\frac{2}{3} \text{ turns}$$

The denominator 3 will divide equally into the following circles of holes, so you can use any plate that contains one of the circles.

Plate one: 15 and 18

Plate two: 21 and 33

Plate three: 39

To apply the fraction 2/3 to the circle you choose, convert the fraction to a fraction that has the number of holes in the circle as a denominator. For example, if you choose the 15 hole circle, the fraction 2/3 becomes 10/15. If plate 3 happens to be on the index head, multiply the denominator 3 by 13 to equal 39. In order not to change the value of the original indexing fraction, also multiply the numerator by 13

$$\frac{2}{3} \times \frac{13}{13} = \frac{26}{39}$$

The original indexing rotation of 6 2/3 turns full turns and 26 holes on the 39-hole circle.

When the number of divisions exceeds 40, you may divide both the numerator and the denominator 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. To use a 16-hole circle, divide the numerator and denominator by 10.

$$\frac{40/10}{160/10} = \frac{4}{16}$$

Turn 4 holes on the 16-hole circle.

It is usually more convenient to reduce the original fraction to its lowest terms and then multiply both terms of the fraction by a factor that will give a number representing a circle of holes.

$$\frac{40}{160} = \frac{1}{4}$$

$$\frac{1}{4} \times \frac{4}{4} = \frac{4}{16}$$

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 4/9 × 2/2 = 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.

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. Later 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: 24-25-28-30-34-37-38-39-4-42-43

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 a degree 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° (1/9 turn × 9°/turn), 1 hole in a 27-hole circle equals 1/3° (1/27 turn × 9°/turn), 3 holes in a 54-hole circle equal 1/2° (1/18 turn × 9°/turn). 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°

9 = 1 9 × 2 = 1 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 minutes/degree.

$$9^\circ \times 60 \text{ min/degree} = 540 \text{ min}$$

Therefore, open 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

540 = 10 × 2 = 20 (20-hole circle)

or 2 holes on the 20-hole circle.

COMPOUND INDEXING

Compound indexing is a combination of two plain indexing procedures. 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 33-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 have factors that will 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 you turn the index crank clockwise, turn the index plate 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 the index plate.

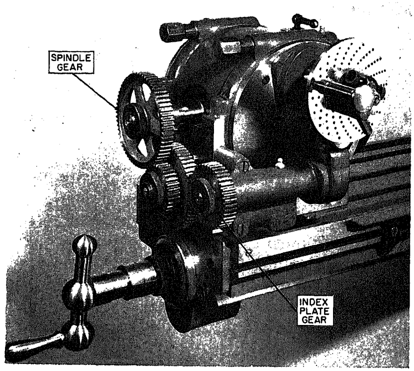
Figure 11-18 shows a dividing head set up for differential indexing. The index crank is turned as it is for plain indexing, thus turning the spindle gear and then the compound gear and the idler to drive the gear which turns the index plate. Specific procedures for installing the gearing and arranging 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



28.210X

Figure 11-18.—Differential indexing.

number or vice versa (depending on 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, you can get the desired number of equally spaced divisions by using either direct or plain indexing.

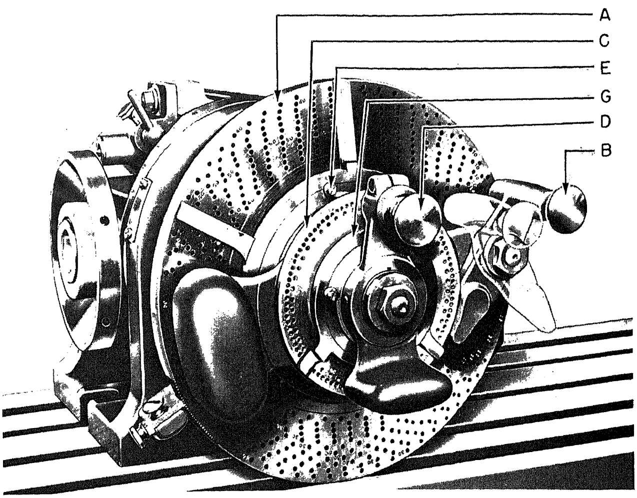
By using one or the other of these methods, you may index up to 2,640 divisions. To increase the range of divisions, use the high number index plates in place of the standard index plate. These high number plates have a greater number of circles of holes and a greater range of holes in the circles than the standard plates. This increases the range of possible divisions 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. 11-19) consists of a large index plate with sector arms and a crank and a small index plate with sector arms and a crank. The large index plate (A, fig 11-19) 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. 11-19) is mounted on the housing of the planetary gearing (G, fig. 11-19), which is built into the index crank (B, fig. 11-19) 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 the 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. 11-19) is rotated, the large index crank remains stationary but the main shaft that drives the work revolves in the ratio of 1 to 100. This ratio, superimposed on the 40 to 1 ratio between the worm and worm



126.28X

Figure 11-19.—The wide range divider.

wheel (fig. 11-20), causes the dividing head spindle to rotate in the ratio of 4,000 to 1. This means 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.

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 the crank. The dividing head manufacturer provides tables listing many of the settings for specific divisions that 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, use the manufacturer's manual or other reference for instructions on how to compute the required settings.

Adjusting the Sector Arms

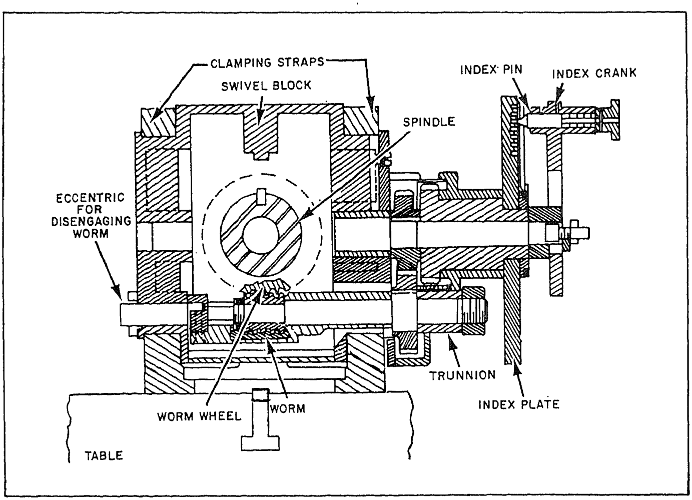
To use the index head sector arms, turn the left-hand arm to the left of the index pin, which is inserted into the first hole in the circle of holes that is to be used. Then loosen the setscrew (fig. 11-19E) and adjust the right-hand arm of the sector so that the correct number of holes will be contained between the two arms (fig. 11-21). 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 on the index pin.

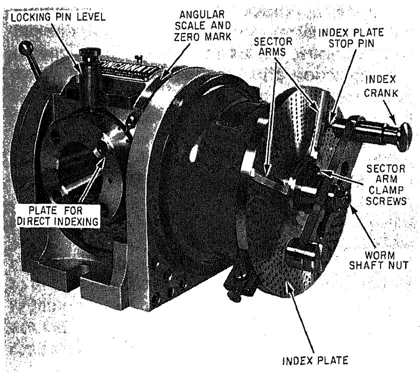
CUTTERS AND ARBORS

When you perform 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





28.371X

Figure 11-21.—Principal parts of a late model Cincinnati universal spiral index head.

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 each of the various operations you will perform. 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 that they perform. The most common cutters are: (1) plain milling cutters of various widths and diameters, used principally for milling flat surfaces that 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 a right angle to the axis of the cutter; and (4) forming cutters, used to produce surfaces with an irregular outline.

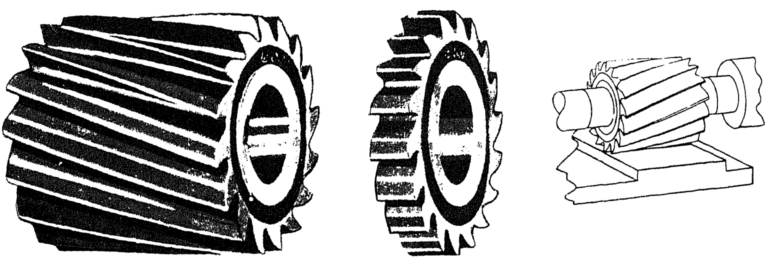
Milling cutters may also be classified as arbor-mounted, or shank-mounted. Arbor-mounted cutters are mounted on the straight shanks of arbors. The arbor is then inserted into the milling machine spindle. We will discuss the methods of mounting arbors and cutters in greater detail later in this chapter.

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.

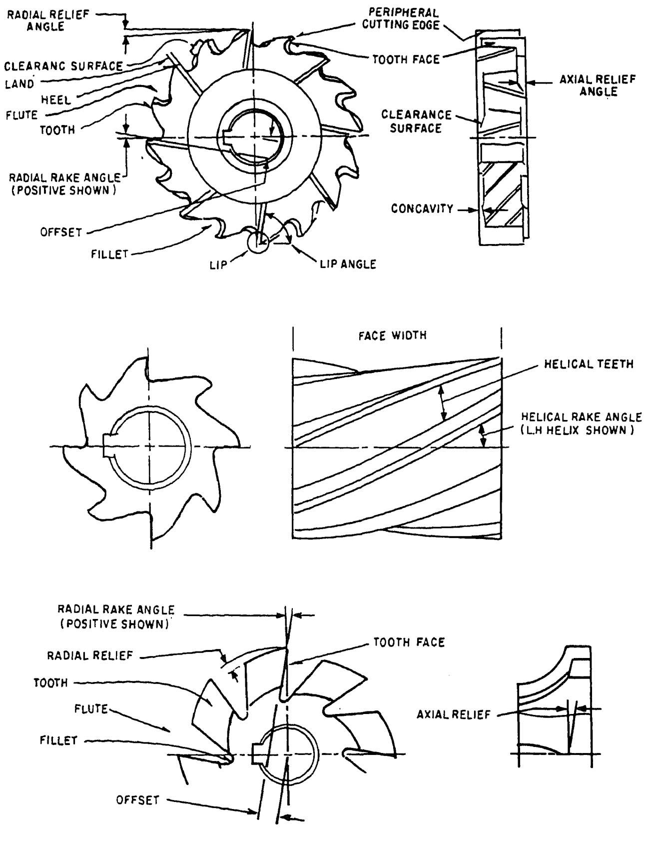
PLAIN MILLING CUTTER.—You will use plain milling cutters to mill flat surfaces that are parallel to the cutter axis. As you can see in figure 11-22, a plain milling cutter is a cylinder with teeth



28.372

Figure 11-22.—Plain milling cutters.

cut on the circumference only. Plain milling cutters are made in a variety of diameters and widths. Note in figure 11-23, that the cutter teeth may be either straight or helical. When the width is more than 3/4 inch, the teeth are usually helical. The teeth of a straight cutter tool are parallel to axis of the cutter. This causes each tooth to cut 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 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 and width of the cutter, and the diameter of the arbor hole in the cutter.

SIDE MILLING CUTTER.—The side milling cutter (fig. 11-24) is a plain milling cutter with teeth cut on both sides as well as on the 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. 11-25) 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.

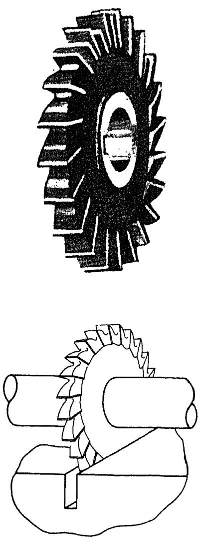


Figure 11-24.—Side milling cutter.

SIDE MILLING CUTTER (INTERLOCKING).—Side milling cutters whose teeth interlock (fig. 11-26) can be used to mill 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, with a face width usually less than 3/16 inch. This type of cutter usually has more teeth for a given diameter than a plain cutter. It is thinner at the center than at the outer edge to give proper clearance for milling

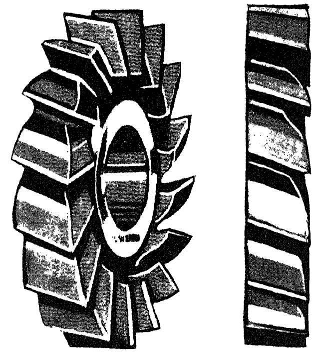


Figure 11-25.—Half-side milling cutter.

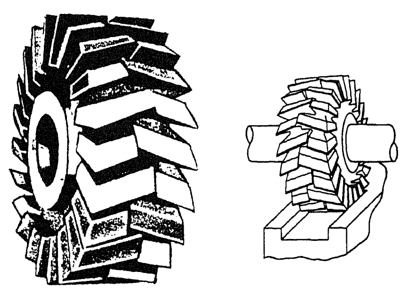


Figure 11-26.—Interlocking teeth side milling cutter.

deep slots. Figure 11-27 shows a metal slitting saw with teeth cut in the circumference of the cutter only. Some saws, such as the one in figure 11-28, have side teeth which achieve better cutting action, break up chips, and prevent dragging when you cut deep slots. For heavy sawing in steel, there are metal slitting saws with staggered teeth, as shown in figure 11-29. These cutters are usually 3/16 inch to 3/8 inch thick.

SCREW SLOTTING CUTTER.—The screw slotting cutter (fig. 11-30) is used to cut shallow slots, such as those in screw heads. This cutter has fine teeth cut on its circumference. It is made in various thicknesses to correspond to American Standard gauge wire numbers.

ANGLE CUTTER.—Angle cutters are used to mill surfaces that are not at a right angle to

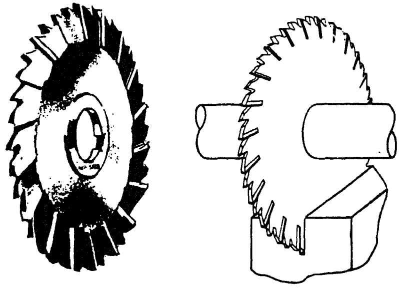


Figure 11-29.—Slitting saw with staggered teeth.

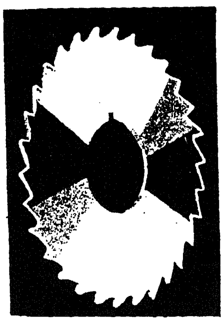


Figure 11-27.—Metal slitting saw.

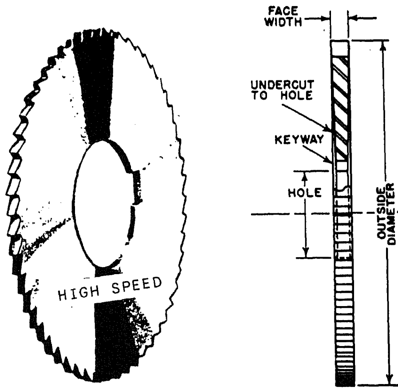


Figure 11-30.—Screw slotting cutter.

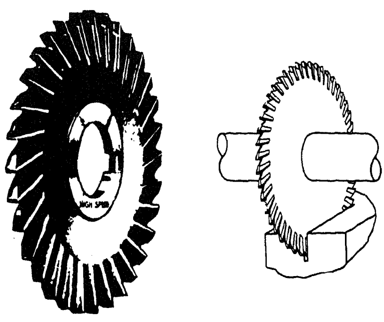


Figure 11-28.—Slitting saw with side teeth.

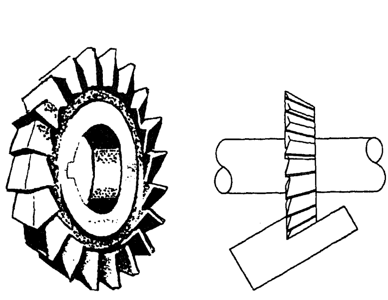


Figure 11-31.—Single angle cutter.

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 the type, hand, outside diameter, thickness, hole size, and angle.

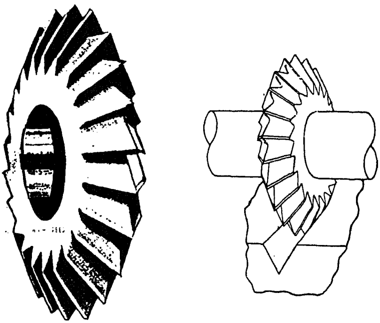


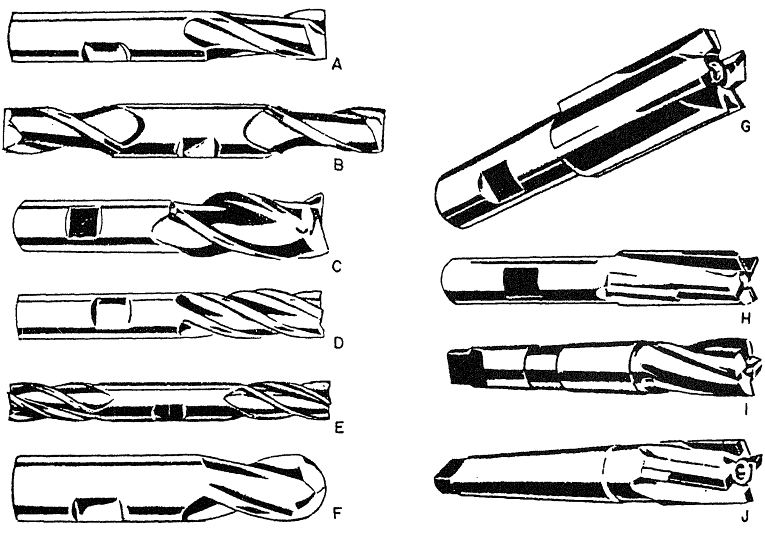
Figure 11-32.—Double angle cutter.

There are two types of angle cutters—single and double. The single angle cutter, shown in figure 11-31, 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. 11-32) 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 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 mill cutters may be the SOLID TYPE with the teeth and the shank as an integral part (fig. 11-33), or they may



(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.

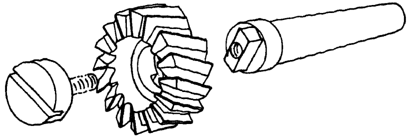
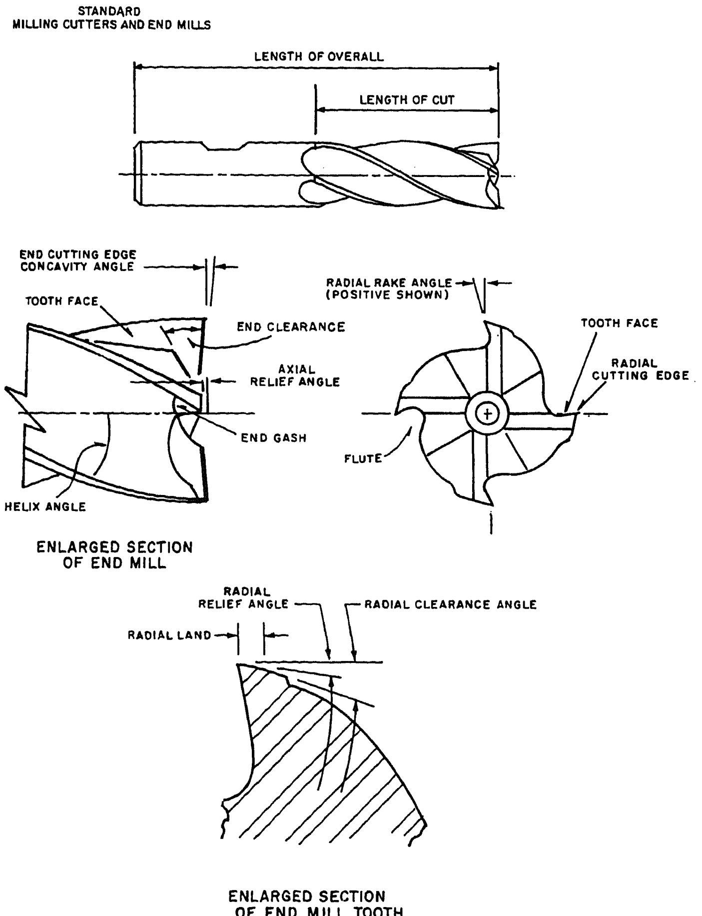


Figure 11-34.—Shell end mill.

be the SHELL TYPE (fig. 11-34) in which the cutter body and the shank or arbor are separate. End mill cutters have teeth on the circumference and on the end. Those on the circumference may be either straight or helical (fig. 11-35).

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 mill cutters. Common operations include the milling of slots, keyways, pockets, shoulders, and flat surfaces, and the profiling of narrow surfaces.

End mill cutters are used most often on vertical milling machines. However, they also are used frequently on machines with horizontal spindles. Many different types of end mill cutters are available in sizes ranging from 1/64 inch to 2 inches. They may be made of high-speed steel, may have cemented carbide teeth, or may be of the solid carbide type.

TWO-FLUTE END MILLS have only two teeth on their circumference. The end teeth can cut to the cutter. Hence, they may be fed 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 the cutter on one end only, or they may be the double-end type. (See fig. 11-33.)

MULTIPLE-FLUTE END MILLS have three, four, six, or eight flutes and normally are available in diameters up to 2 inches. They may be either the single-end or the double-end type (fig. 11-33).

BALL END MILLS (fig. 11-33) are used for milling fillets or slots with a radius bottom, for rounding pockets and the bottom of holes, and for all-around 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-flute 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. 11-34) have a hole for mounting the cutter on a short (stub) arbor. The center of the shell is recessed for the screw or nut that fastens the cutter to the arbor. These mills are made in larger sizes than solid end mills, normally 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, and usually have helical teeth.

FACE MILLING CUTTER.—Inserted tooth face milling cutters (fig. 11-36) 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, replaceable teeth made of carbide or any alloy steel.

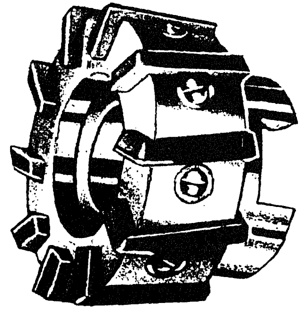


Figure 11-36.—Inserted tooth face milling cutter.

T-SLOT CUTTER.—The T-slot cutter (fig. 11-37) 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.

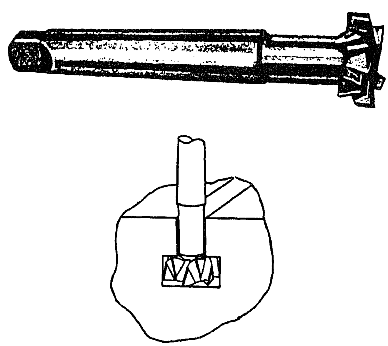


Figure 11-37.—T-slot cutter.

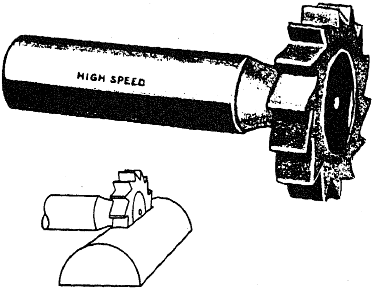


Figure 11-38.—Woodruff keyseat cutter.

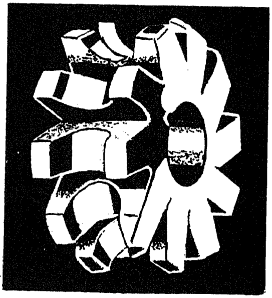


Figure 11-40.—Concave cutter.

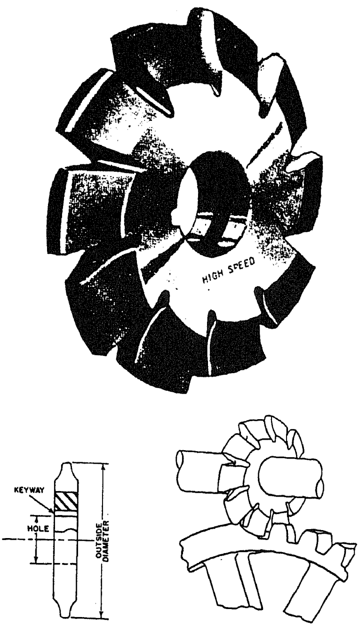


Figure 11-39.—Involute gear cutter.

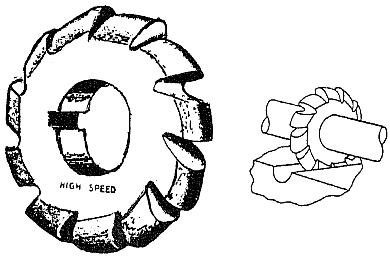


Figure 11-41.—Convex cutter.

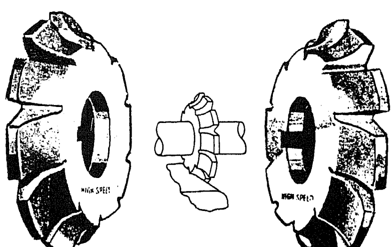


Figure 11-42.—Corner rounding cutter.

WOODRUFF KEYSEAT CUTTER.—A Woodruff keyseat cutter (fig. 11-38) is used to 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, and so on. Figure 11-39 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. 11-40) is used to mill a convex surface, and a convex cutter (fig. 11-41) is used to mill a concave surface.

CORNER ROUNDING CUTTER.—Corner rounding cutters (fig. 11-42) are formed cutters that are used to round corners up to one-quarter of a circle.

SPROCKET WHEEL CUTTER.—The sprocket wheel cutter (fig. 11-43) is a formed cutter that is used to mill teeth on sprocket wheels.

GEAR HOB.—The gear hob (fig. 11-44) Is a formed milling cutter with teeth cut like threads on a screw.

FLY CUTTER.—The fly cutter (fig. 11-45) 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. There will be times when you need a special formed cutter for a very limited number of cutting or boring operations. This will probably be the type of cutter you will use since you can grind it to almost any form you desire.

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.

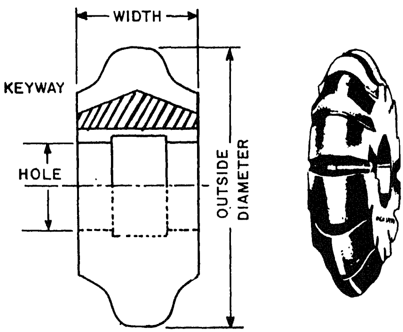


Figure 11-43.—Sprocketed wheel cutter.

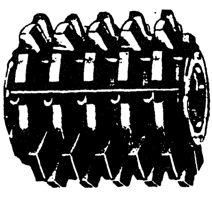


Figure 11-44.—Gear hob.

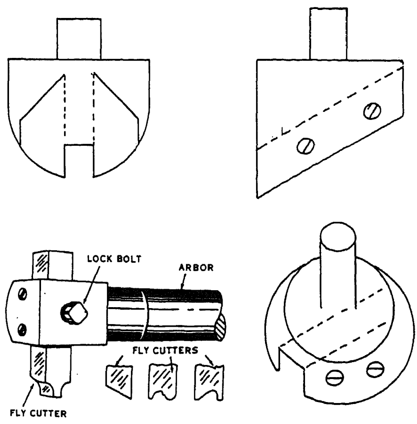


Figure 11-45.—Fly cutter arbor and fly cutters.

Naval Education and Training Command

NAVEDTRA 12204
May 1990
0502-LP-213-1100

Training Manual (TRAMAN)

Machinery Repairman 3 & 2

N.F.R.O.

RETURN TO GOV. DOCS. CLERK

DISTRIBUTION STATEMENT A: Approved for public release; distribution is unlimited.

Nonfederal government personnel wanting a copy of this document must use the purchasing instructions on the inside cover.

S/N 0502-LP-213-1100

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 11-47. 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 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 11-48, are called stub arbors. Stub arbors are used when it is impractical to use a longer 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. 11-49) 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 11-48,

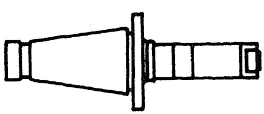


Figure 11-48.—Stub arbor.

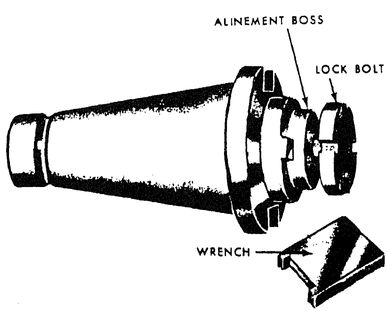


Figure 11-49.—Shell end mill arbor.

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 11-44. 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 11-50.

SCREW ARBOR.—Screw arbors (fig. 11-51) 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 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

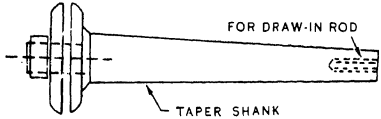


Figure 11-50.—Screw slotting cutter arbor.

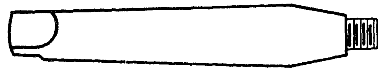


Figure 11-51.—Screw arbor.

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 11-52 shows a typical taper adapter. Some cutter adapters are designed to be used with tools that have taper shanks and a cam 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), 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 11-53, are similar to taper adapters except that they always have straight, rather than tapered holes. They are used to hold straight shank drills, end mills, and so on. 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,

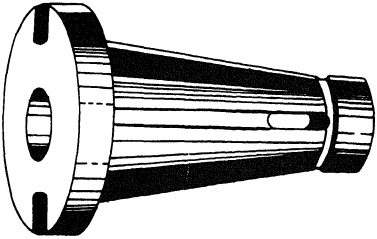


Figure 11-52.—Taper adapter.

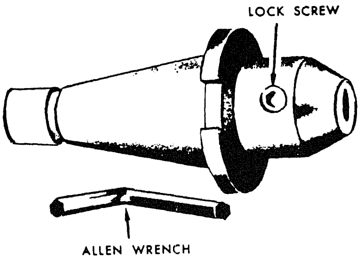


Figure 11-53.—Cutter adapter.

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. 11-54) 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 the floor. Use figure 11-7 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.

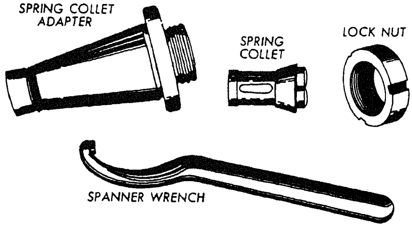


Figure 11-54.—Spring collet chuck adapter.

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.
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 striping 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 work to its required size by individually milling each of the flat surfaces on the workpiece. Plain milling cutters, such as the ones shown in figure 11-22, are used for plain milling. If possible, select a cutter that is slightly wider than the width of the 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 closely as possible to the milling machine column so that you can mount the cutter near the column. The closer you place the cutter and the 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 11-55.

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.

5. Mount the arbor in the spindle.

6. Clean and position the spacing collars and place them on the arbor so 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 closely as possible to the cutter. Make sure that the work and the vise will clear all parts of the machine.

8. Install the arbor nut and tighten it finger tight only.

9. Position the overarm and mount the arbor support.

10. After supporting the arbor, tighten the arbor nut with a wrench.

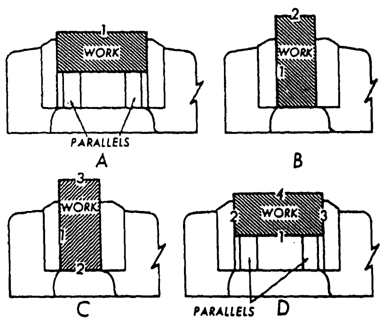


Figure 11-55.—Machining sequence to square a block.

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 the 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 form the vise.

To machine the second side, plate the work in the vise as shown in figure 11-55B. 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 11-55C 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 work 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 11-55D 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. Do this 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 11-56. You do face milling to produce flat 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 the cutter before you make the work setup. Mount the cutter by any means suitable for the cutter you have selected.

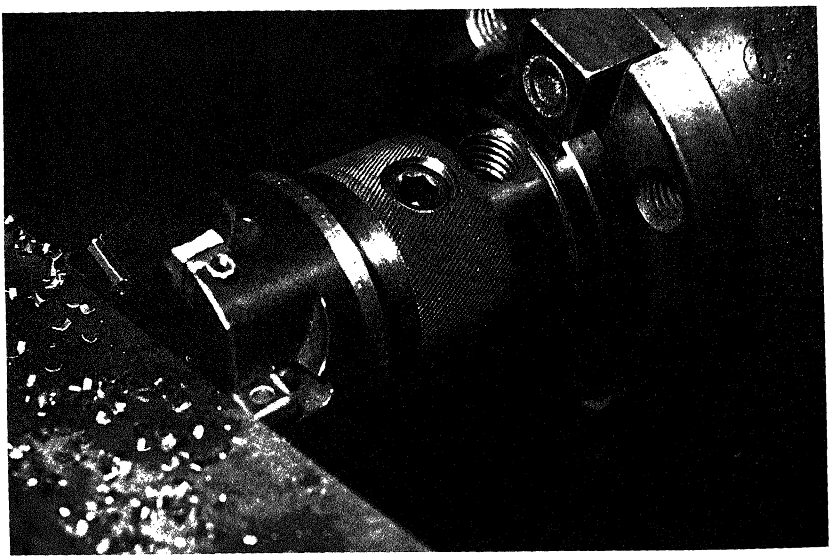
Work Setup

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 11-57. You can also use a machinist's square and a feeler gauge, as shown in figure 11-58.

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 11-56 so the thrust of the cutter is toward the solid vise jaw.



28.402

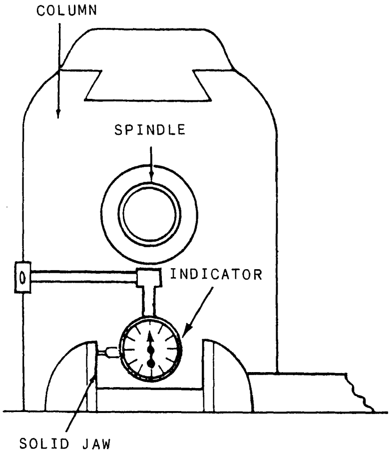


Figure 11-57.—Aligning vise jaws using an indicator.

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 even 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 11-59 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.

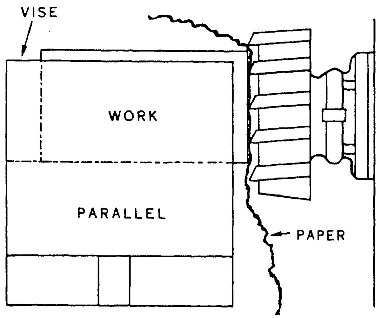


Figure 11-59.—Picking up the work surface.

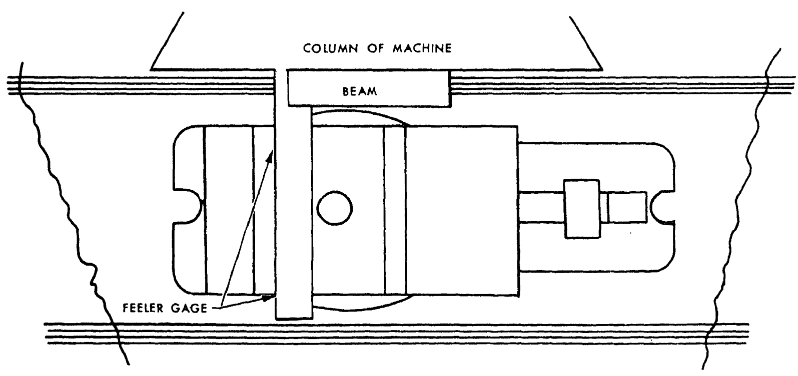


Figure 11-58.—Aligning vise jaws using a square.

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 onto the cutter.

12. Set the roughing depth of cut, using the graduated dial, and lock the saddle.

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 the saddle 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 form 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 the other end is ready for machining. Mill this end in the same manner as the first, but be sure to 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 11-60. 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.

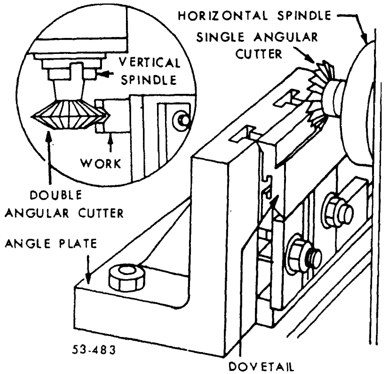


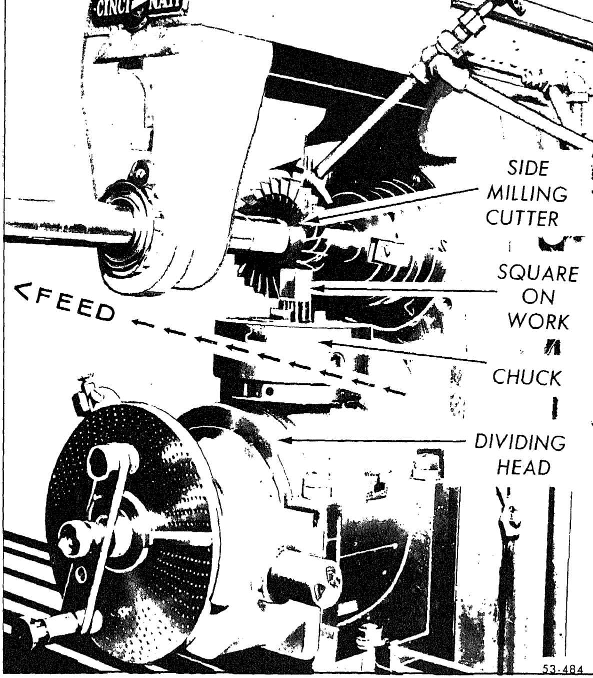
Figure 11-60.—Angular milling.

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 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 the cutter diameter is large enough so 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 11-61 and 11-62. The cutter



28.407

Figure 11-61.—Milling a square on work held vertically.

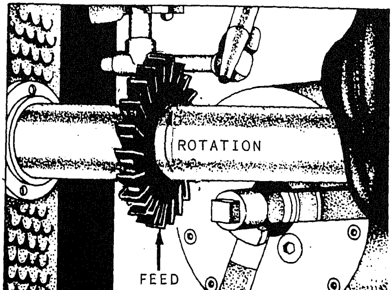


Figure 11-62.—Milling a square on work held horizontally.

shown in figure 11-61 rotates in a counterclockwise direction and the work is fed toward the left. The cutter shown in figure 11-62 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 the 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

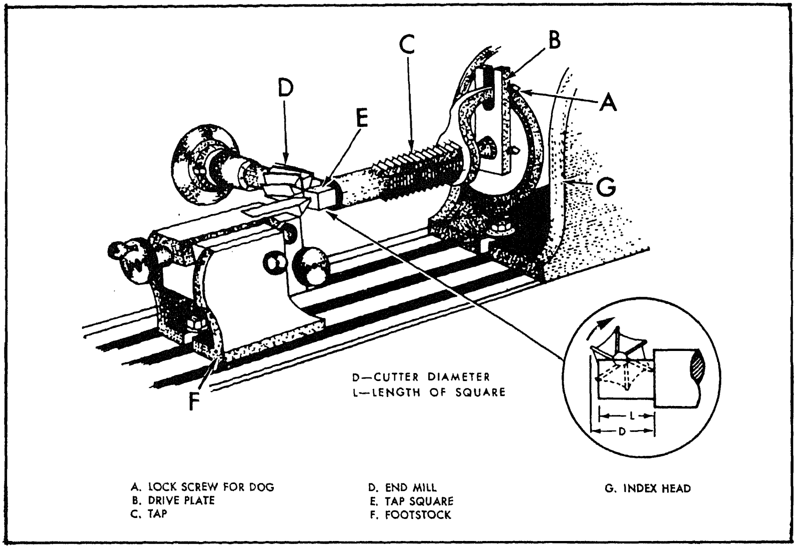


Figure 11-63.—Milling a square using an end mill.

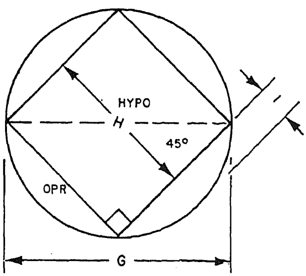


Figure 11-64.—Diagram of a square.

of the work that will tend to tighten the chuck on the index head spindle. When you mount work between centers, a dog rotates the work. The drive plate, shown in figure 11-63, 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 11-63A. 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. 11-63G) near the cutter edge of the table to ensure the cutter and the 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 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. 11-64) 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

Opposite side = Side of a square

Hypotenuse = Diagonal of square

45° = 90° bisected

$$H = G \times 0.707 \text{ or } \frac{\text{Opposite side}}{\text{Hypotenuse}} = \text{sine } 45°$$

The diagonal of a square equals the distance across the flats times 1.414. This is expressed as

$$G = H \times 1.414 \text{ or } \frac{\text{Hypotenuse}}{\text{Opposite side}} = \text{cosec } 45°$$

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

$$(I = \frac{G - H}{2})$$

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 11-65) is equal to the diagonal (the diameter of the stock) of the hexagon times 0.866 or

Opposite side = Largest hexagon that can be machined

Hypotenuse = Diagonal or diameter of round stock

$$H = G \times 0.866 \text{ or } \frac{\text{Opposite side}}{\text{Hypotenuse}} = \text{Sine } 60°$$

The diagonal of a hexagon equals the distance across the flats times 1.155, or

$$G = H \times 1.155 \text{ or } \frac{\text{Hypotenuse}}{\text{Opposite side}} = \text{cosec } 60°$$

The length of a flat is equal to one-half the length of the diagonal,

$$r = \frac{G}{2}$$

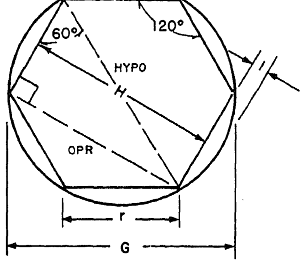


Figure 11-65.—Diagram of a hexagon.

We will explain two methods of machining a square or hexagon: machining work mounted in a chuck and machining work mounted between centers.

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 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 you have mounted the index head on the table, position the head spindle in the vertical position, as shown in figure 11-61. 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 will allow you to feed the work horizontally.

Then, tighten the work in the chuck to keep it from turning due to the cutter's thrust. Install the arbor, cutter, and arbor support. The cutter should be as close as practical to the column. Remember, this is done so the setup will be more rigid. Set the machine for the correct roughing speed and feed.

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 the 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 11-62 and 11-63. 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 when the work is held 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 into the spindle and align it with the footstock center.

3. Select and mount the desired end mill, preferably one whose diameter is slightly greater than the length of the flat you are to cut, as shown in figure 11-63.

4. Mount the work between centers. Make sure that the drive dog is holding the work securely.

5. Set the machine for roughing speed and feed.

6. Pick up the side of the work and set the graduated crossfeed dial at ZERO.

7. Lower the work until the cutter clears the footstock.

8. Move the work until the end of the work is clear of the cutter.

9. Align the cutter with the end of the work. Use a square head and rule, as shown in figure 11-66.

NOTE: Turn the machine off before aligning the cutter by this method.

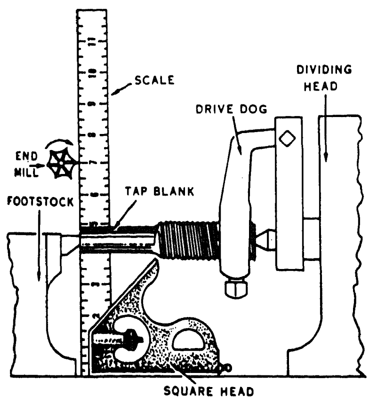


Figure 11-66.—Aligning the work and the cutter.

12. While feeding the work vertically, machine side 1. Lower the work to 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 to 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 to 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 in 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 to machine both flats.

1. Apply layout dye to both ends of the work.

2. Place the work on a pair of V-blocks, as shown in figure 11-67.

3. Set the scriber point of the surface gauge to the center height of the work. Scribe horizontal lines on both ends of the work, as illustrated in figure 11-67.

4. Mount the index head on the table with its spindle in the horizontal position.

5. Again, set the surface gauge scriber point, but to the centerline of the index head spindle.

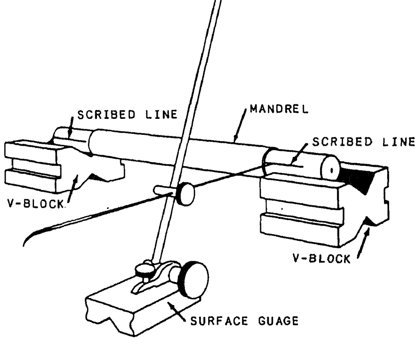


Figure 11-67.—Layout of the work.

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 gauge 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 gauge 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 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.

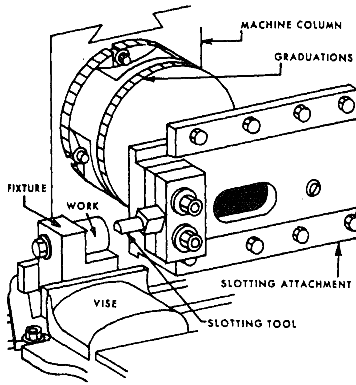


Figure 11-68.—Slotting attachment.

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 done with a milling machine attachment called a slotting attachment, as shown in figure 11-68. The slotting attachment is fastened to the milling machine column and driven by the spindle. This attachment changes the rotary motion of the spindle to a reciprocating motion much like that of a shaper. You can vary the length of the stroke within a specified range. A pointer on the slotting attachment slide indicates the length of the stroke. You can pivot the head of the slotting attachment and position it at any desired angle. Graduations on the base of the slotting attachment indicate the angle at which the head is positioned. The number of strokes per minute is equal to the spindle rpm and is determined by the formula:

$$\text{Strokes per minute} = \frac{\text{CFS} \times 4}{\text{length of stroke}}$$

The cutting tools used with slotting attachments are ground to any desired shape from high-speed steel tool blanks and are clamped to the front of the slide or ram. You can use any suitable means for holding the work, but the most common method 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.

Parting

Use a metal slitting saw for sawing or parting operations and for milling deep slots in metals and in a variety of other materials. Efficient sawing depends to a large extent on 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 style 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 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 into 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.

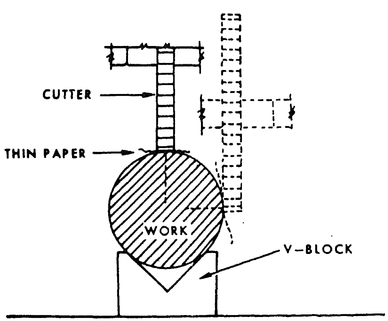


Figure 11-69.—Aligning the cutter using a paper strip.

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 11-69 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 to center the cutter over the centerline of the keyseat to be cut.

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 11-70. After the paper tears, lower the work to just below the bottom of the

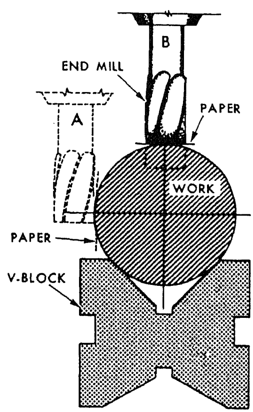


Figure 11-70.—Aligning an end mill with the work.

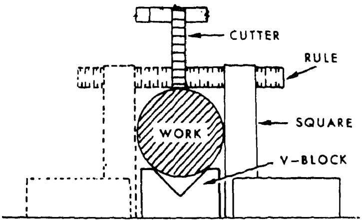


Figure 11-71.—Visual alignment of a cutter.

end mill. Then move the work a distance equal to the radius of the work plus the radius of the end mill to center the mill over the centerline of the keyseat to be cut. 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 11-70B. 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. You can determine the cutter rpm and the longitudinal feed in the same manner as you do for conventional milling cutters. Because of the higher speeds and feeds involved, more heat is generated, so flood the work and the cutter with coolant.

When extreme accuracy is not required, you can align the work with the cutter visually, as shown in figure 11-71. 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

Table 11-1.—Values for Factor (f) for Various Sizes of Shafts

DIAMETER OF SHAFT (INCHES)	WIDTH OF KEY IN 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

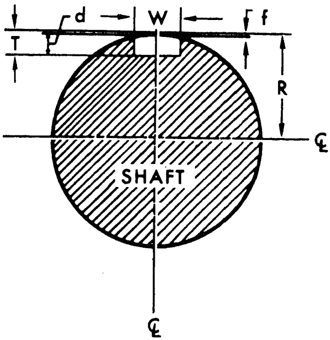


Figure 11-72.—Keyseat dimensions for a straight square key.

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 to measure the width of the cut.

Specifications for the depth of cut are usually furnished. When specifications are not available, you can determine the total depth of cut for a square keyseat by using the following formula based on dimensions shown in figure 11-72.

$$\text{Total depth of cut (T)} = d + f$$

where

$$d = \frac{W}{2} = \text{depth of the keyseat}$$

$$f = R - \sqrt{R^2 - \left(\frac{W}{2}\right)^2} = \text{height of arc}$$

W = width of the key

R = radius of the 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 the keyseats for accuracy with rules, outside and depth micrometers, vernier calipers, and go-no-go gauges. Use table 11-1 for 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 you do for milling straight external keyseats.

A Woodruff keyseat cutter (fig. 11-73) has deep flutes cut across the cylindrical surface of

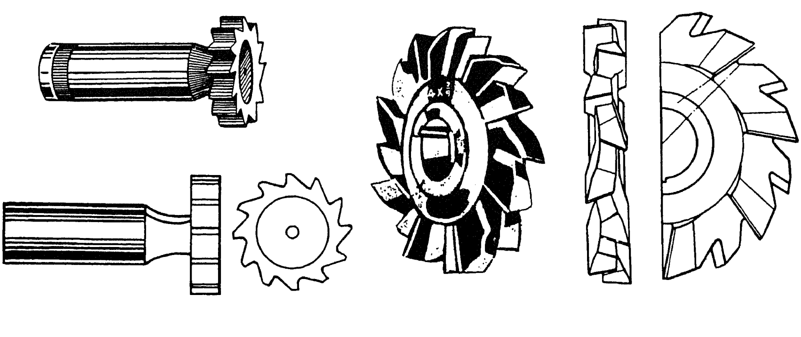
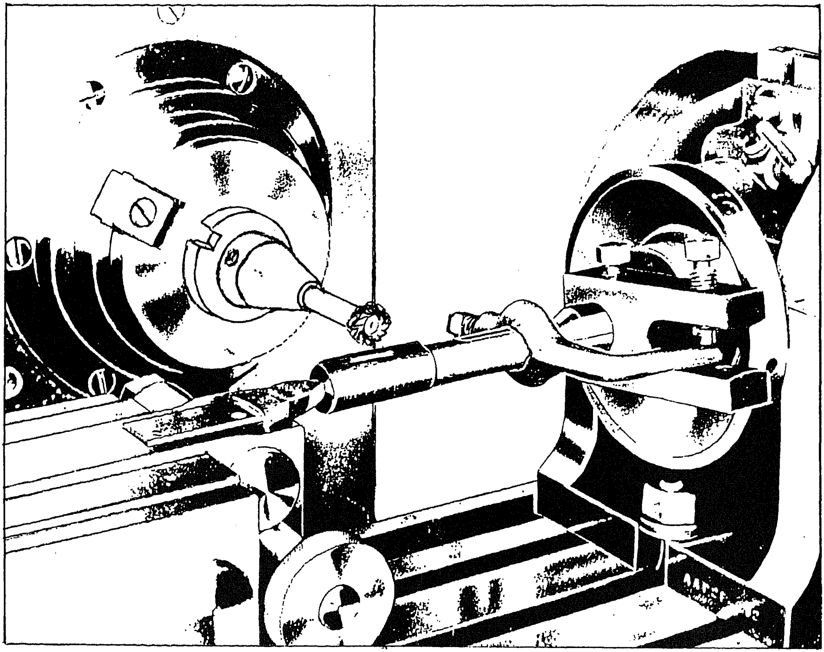


Figure 11-73.—Woodruff keyseat cutter.



28.416

Figure 11-74.—Milling a Woodruff keyseat.

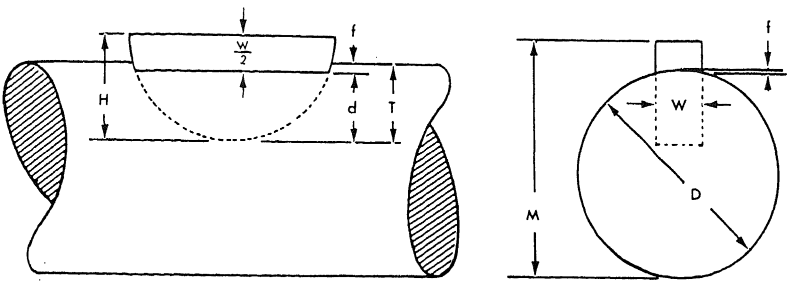


Figure 11-75.—Dimensions for a Woodruff keyseat.

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 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 their cutting action.

As discussed earlier, to mill a Woodruff keyseat in a shaft, you use a cutter that has the same diameter and 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 11-74.

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 the work with an outside micrometer to obtain the distance M in figure 11-75. Measure the correct micrometer reading over the shaft and the key. You can also determine distance M by using the formula

$$M = D + \frac{(W)}{(2)} - f$$

where

M = micrometer reading

D = diameter of the shaft

W = width of the key

f = height of the arc between the top of the slot and the top of the shaft.

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 the tooth depends on 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 11-76. The

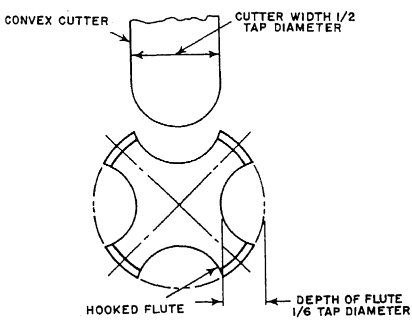


Figure 11-76.—Hooked tap flutes.

number of flutes is determined by the diameter of the tap. Taps 1/45 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 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 gauge 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 horizontal 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 gauge scriber.
8. Make sure that the surface gauge is still at center height.

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

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

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.

16. Set the machine for finishing speed and feed and finish machine all flutes to the correct size.

17. Remove the work, deburr it, and check it for accuracy.

Reamer Flutes

You may mill flutes on reamers with angular fluting cutters, but you normally use special formed fluting cutters. The advantages of cutting the flutes with a formed cutter rather than with an angular cutter are that the chips are more readily removed and the flute cutting teeth are stronger. Also, the teeth are 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 on 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 11-77.

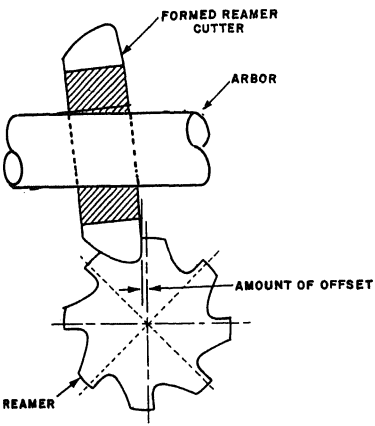


Figure 11-77.—Negative rake tooth.

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 *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, and so on.

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

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.

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.

5. Rotate the cutter until a tooth is in the vertical position.

6. Shut off the machine.

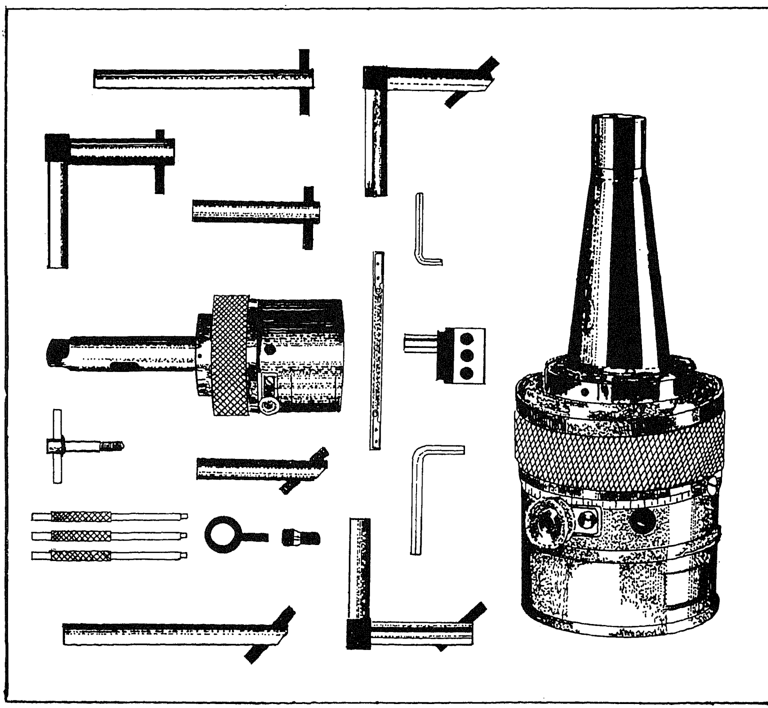
7. Move the table until the point of the footstock center is aligned with the tooth that is in the vertical position.

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.

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 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 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 11-45. 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 the 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. You determine the speeds used for drilling and reaming in the same manner as for drilling and reaming in the lathe or the drill press. The work is fed into the drill or reamer by either hand or power feed. If you mount the cutting tool in a horizontal position, use the transverse or saddle feed. If you mount a drill or reamer in a vertical position, as in a vertical type machine, 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 11-78 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 11-79 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 a drill press.

Fly cutters, which we discussed previously, can also be used for boring, as shown in figure 11-79. 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 on the same factors: hardness of the

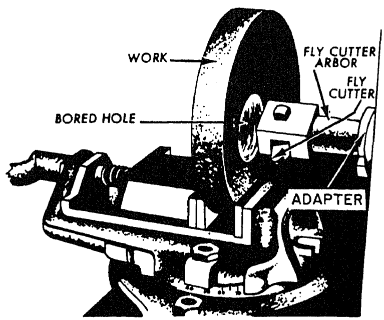


Figure 11-79.—Boring with a fly cutter.

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 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 that 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. 11-80) you can convert the horizontal spindle machine to a vertical spindle machine and can 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 more easily do jobs that 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 11-81. 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.

CIRCULAR MILLING ATTACHMENT

This attachment (fig. 11-82) 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 11-83, 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.

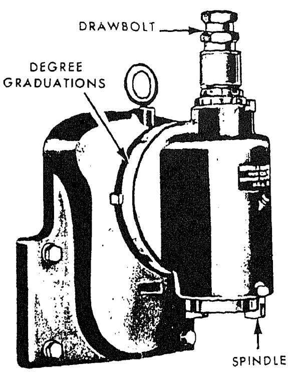


Figure 11-80.—Vertical milling attachment.

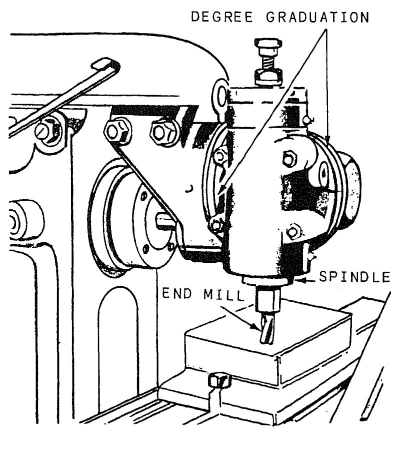


Figure 11-81.—High-speed universal milling attachment.

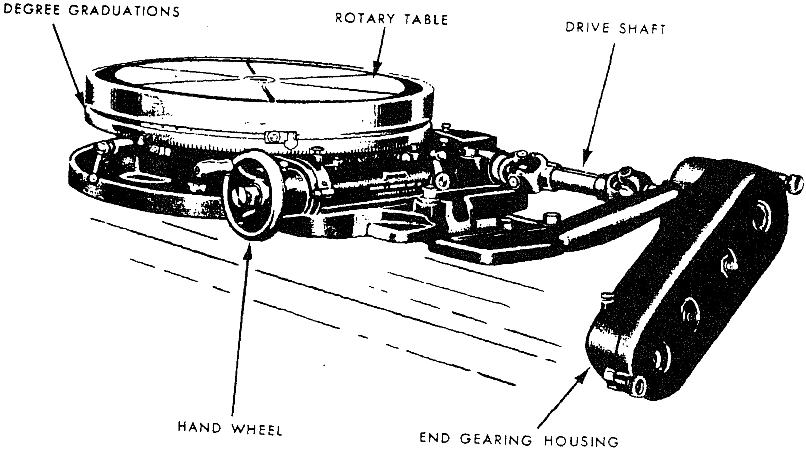


Figure 11-82.—Circular milling attachment with power feed. 28.423

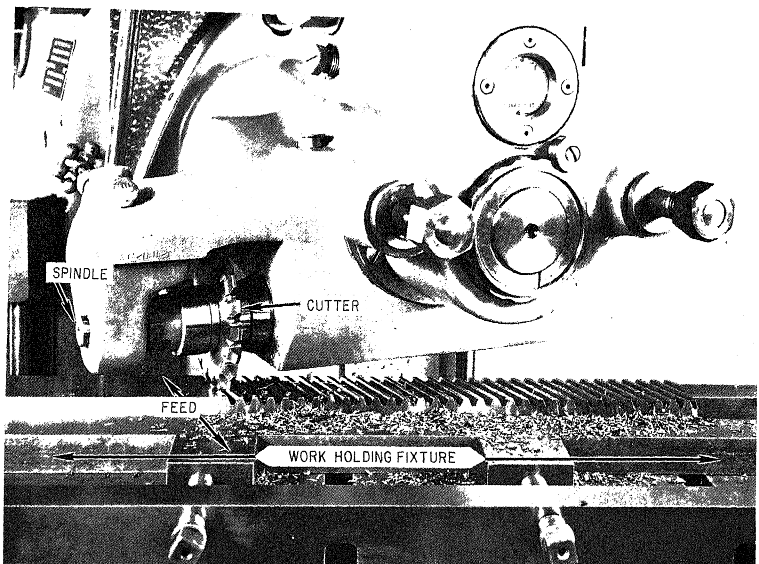


Figure 11-83.—Rack milling attachment. 28.424X

RIGHT-ANGLE PLATE

The right-angle plate (fig. 11-84) is attached to the table. The right-angle slot permits mounting the index head so the axis of the head is parallel to the milling machine spindle. With this attachment you can make work setups that 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. 11-85) are heavy-duty parallels that usually come in matched pairs. They are mounted on the table, and the index head is mounted on the blocks. This 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. 11-86) 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.

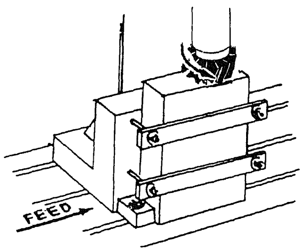


Figure 11-84.—Right-angle plate.

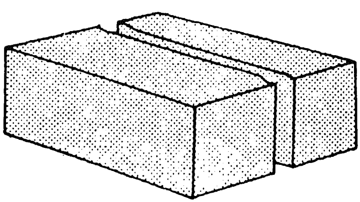


Figure 11-85.—Raising blocks.

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.

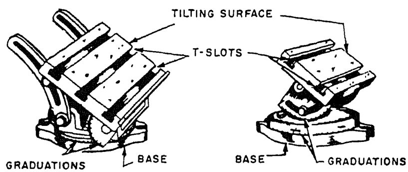


Figure 11-86.—Toolmaker's knees.

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

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 provide 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; therefore, 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.

Use the approximate values in table 11-5 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 an immediate readjustment.

By referring to table 11-6, you can determine the cutter revolutions per minute for cutters

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

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 not shown in table 11-6, determine the proper revolutions per minute of the cutter by using the formula:

$$\text{(a) rpm} = \frac{\text{Cutting speed} \times 12}{3.1416 \times \text{Diameter}}$$

$$\text{or rpm} = \frac{\text{fpm}}{0.2618 \times D}$$

where

rpm = revolutions per minute of the cutter

fpm = required surface speed in feed per minute

D = diameter of the 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?

$$\text{rpm} = \frac{45}{0.2618 \times 0.5}$$

$$\text{rpm} = 343.7$$

To determine cutting speed when you know the spindle speed and cutter diameter, use the following formula:

$$\text{fpm} \times 12 = \text{rpm} \times 3.1416 \times D$$

$$\text{fpm} = \frac{3.1416 \times \text{Diameter} \times \text{rpm}}{12}$$

$$\text{fpm} = 0.2618 \times D \times \text{rpm}$$

EXAMPLE: What is the cutting speed of a 2 1/4-inch end mill running at 204 rpm?

$$\text{fpm} = 0.2618 \times D \times \text{rpm}$$

$$\text{rpm} = 0.2618 \times 2.25 \times 204$$

$$\text{fpm} = 120.1$$

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 varies directly with the amount of metal being removed and can be regulated by adjusting the feed and the depth of cut. The feed and depth of cut are, therefore, interrelated, and depend on 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 on 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 is indicated 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 an 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:

$$\text{Chip load} = \frac{\text{feed rate (ipm)}}{\text{cutter speed (rpm)} \times \text{number of teeth in the cutter}}$$

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

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	.0C2
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	.008	.006	.005	.003	.002
Stainless steels	.006	.005	.004	.003	.002	.002
Monel metals.	.008	.007	.005	.004	.003	.002

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 flow freely on the work and cutter.

HORIZONTAL BORING MILL

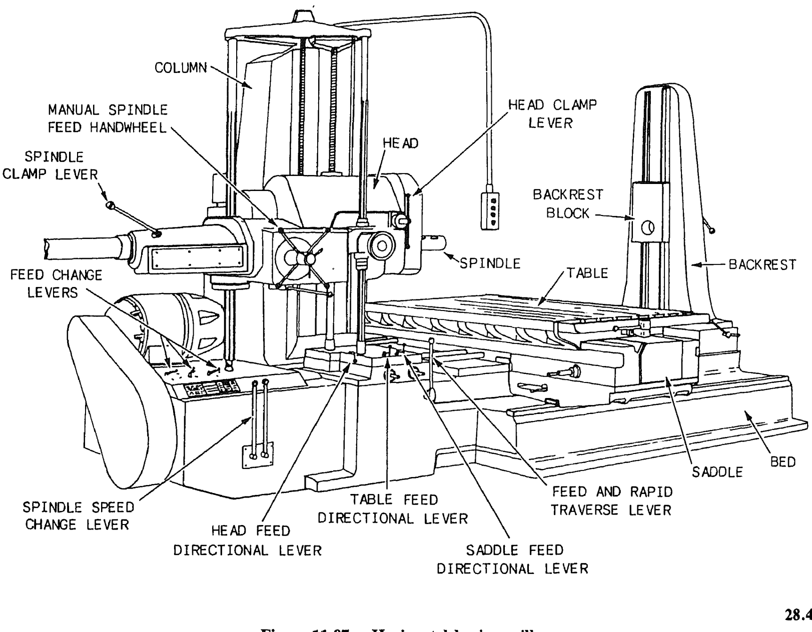
The horizontal boring mill is used for many kinds of shopwork, such as facing, boring, drilling, and milling. In horizontal boring mill

milling machine work; therefore, a detailed discussion of these operations will not be necessary in this section.

The horizontal boring mill (fig. 11-87) consists of four major elements.

BASE AND COLUMN—The base contains all the drive mechanisms for the machine and provides a platform that 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, the auxiliary spindle, and the mechanism for controlling them. The head also provides a station for mounting various attachments. The spindle feed and spindle hand feed controls are contained in the head, along with the quick



28.426

Figure 11-87.—Horizontal boring mill.

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.

SADDLE AND TABLE—A large rectangular slotted table is mounted on a saddle that can be traversed the length of the ways. 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 backrest blocks have an antifriction bearing, which the boring bar passes through and rotates within. The back rest blocks travel vertically 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, consider both of these factors 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 a reference.

It is important that holding clamps used to secure a piece of work be tight. If you use braces, place them so that they cannot come loose. Fasten blocks, stops, and shims 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 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 HEAD

The boring and facing head (fig. 11-88) is used for facing and boring large diameters. This attachment is mounted and bolted directly to the spindle sleeve and has a slide with automatic feed that 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. Whenever you use tools off center, 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, a right angle arm, a boring bar, and a boring bar holder that 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.

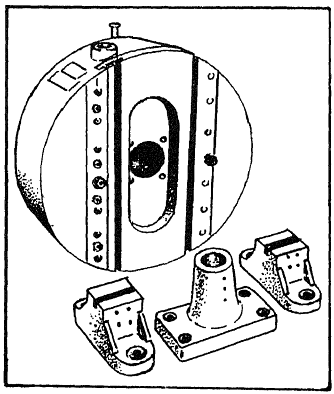


Figure 11-88.—Combination boring and facing head.

2. Disengage the overrunning spindle feed clutch to prevent inadvertent engagement of the spindle power feed while you mount the combination head on the machine. (If the slide is centered and locked, you may run the spindle through it for use in other operations without removing the attachment, but be sure to disengage the spindle overrunning clutch again before you resume use of the slide.

3. Set the spindle for the speed to be used.

4. Before you shift the spindle back-gear to neutral or make 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 you 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, hitting you 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 top 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 to 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 the 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 you mount 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 you operate the head 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. You may use standard milling machine tooling, held in the spindle by a drawbolt that extends through the spindle. A right angle milling attachment is shown in figure 11-89.

BORING MILL OPERATIONS

You should be able to perform drilling, reaming, and boring operations in a boring mill. 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 as they are in a radial

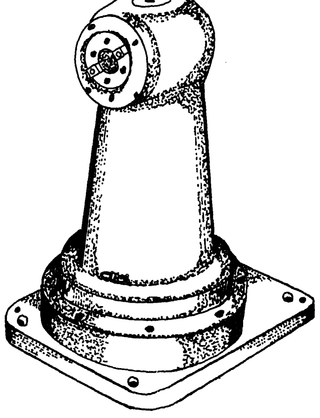
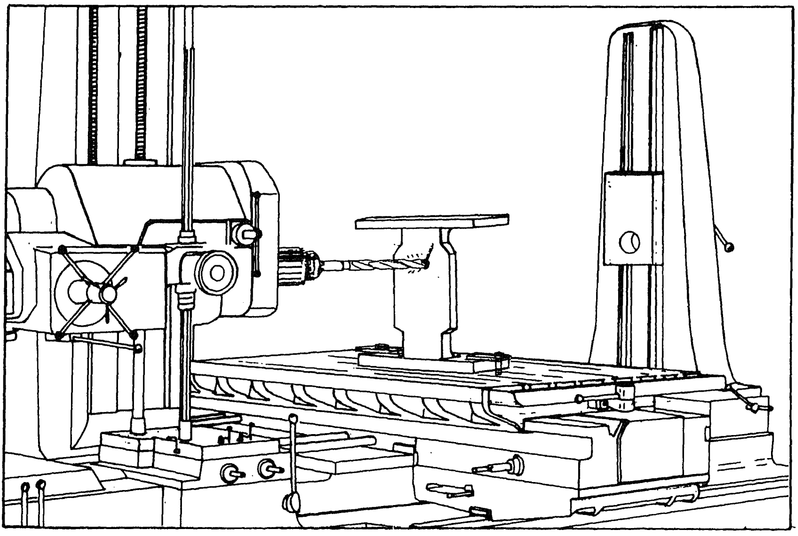


Figure 11-89.—Angular milling head.

machines. In the horizontal boring mill the tool is held in the horizontal position (fig. 11-90), while 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 while the work is held stationary. However, you may, from time to time, find an operation in which you need to hold the bar in



126.30

Figure 11-90.—Drilling in the horizontal boring mill.

a fixed position and move the table lengthwise to complete the operation. (See fig. 11-91.)

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.

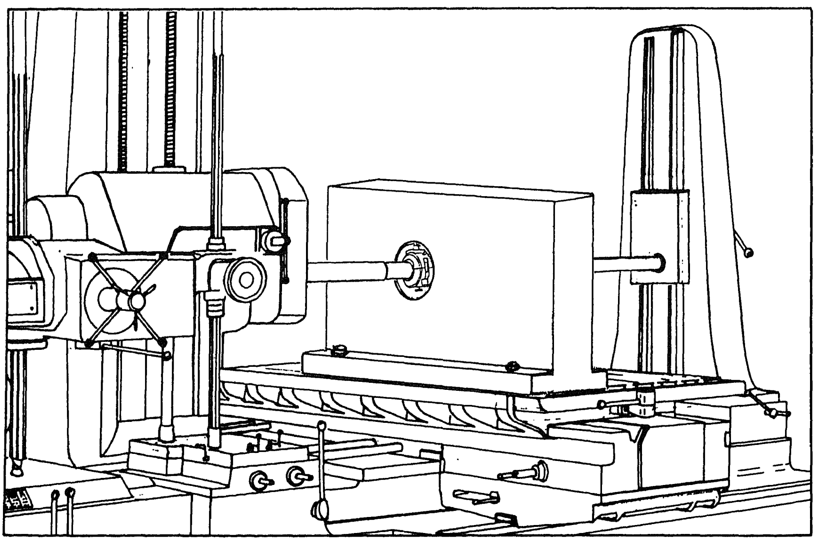
Some boring mills have a single spindle in the head while others have a secondary or auxiliary spindle that can be fitted with a precision head and used in some boring operations. This secondary spindle may also be used on light work such as drilling accurately spaced small holes.

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. *Wiped* means that the bearing has been damaged by being run under an abnormal condition, such as without sufficient lubrication. If it has wiped only slightly, it can probably be scraped to a good bearing surface and restored to service. If it is badly wiped, it will have to be rebabbitted and rebored, or possibly replaced.

When you receive a wiped bearing for repair, follow the procedure listed below as closely as possible:

1. Check the extent of damage and wear marks.
2. Take photos of the bearing to indicate the actual condition of the bearing and for future reference in the machining steps and reassembly.
3. Check the shell halves for markings. A letter or number should be on each half for proper identification and assembly. (If the shell halves are not marked, mark them before you disassemble the bearing.)
4. Inspect the outer shell for burrs, worn ends and the condition of alignment pins and holes.
5. Check the blueprint and job order to ensure that required information has been provided to you.
6. Ensure that the actual shaft size has not been modified from the blueprint.



28.280

Figure 11-91.—Boring bar driven by the spindle and supported in the backrest block.

ing 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 according to NAVSHIPS 9411.813.2. 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 you have properly aligned the bearing 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 required by blueprint specifications.

Oil is distributed through the bearing by oil grooves. These grooves may be of several forms; the two simplest are axial and circumferential. Sometimes circumferential grooves are placed at the ends of the bearings as a controlling device to prevent side leakage, but this type of grooving does not affect the distribution of lubricant.

When you machine grooves into a bearing, you must be careful in beveling the groove out into the bearing leads to prevent excess babbitt from clogging the oil passage. The type of grooves used in a bearing should not be changed from the original design, unless the change is warranted by continuous trouble traceable to improper lubricant distribution within the bearing.

On completion of all machining operations, it is the responsibility of both the repair activity and the ship's force to determine that the bearing meets 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 25 different threads, both standard and metric, is available.

To 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 direction 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 micro-adjustable 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 you have set up the gear train, 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 the 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 threading operation is completed.

9. To disengage the feed, place the thread lead driving gear lever in the 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 given the full respect that is due any machine tool.

- 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 on 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 your fingers at the point of contact of the cutter and the work. There is danger to your eyes from flying chips, so always protect your eyes with goggles and keep your eyes out of the line of cutting action.

SHAPERS, PLANERS, AND ENGRAVERS

In this chapter we will discuss the major types of shapers, planers, and pantographs (engravers), and their individual components, cutters, and operating principles and procedures. A shaper has a reciprocating single-edged cutting tool that removes metal from the work as the work 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 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 only on the forward stroke of the ram. The work is held in a vise or on the worktable, which moves at a right angle to the line of motion of the ram, permitting the cuts to progress across the surface being machined. A shaper is 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

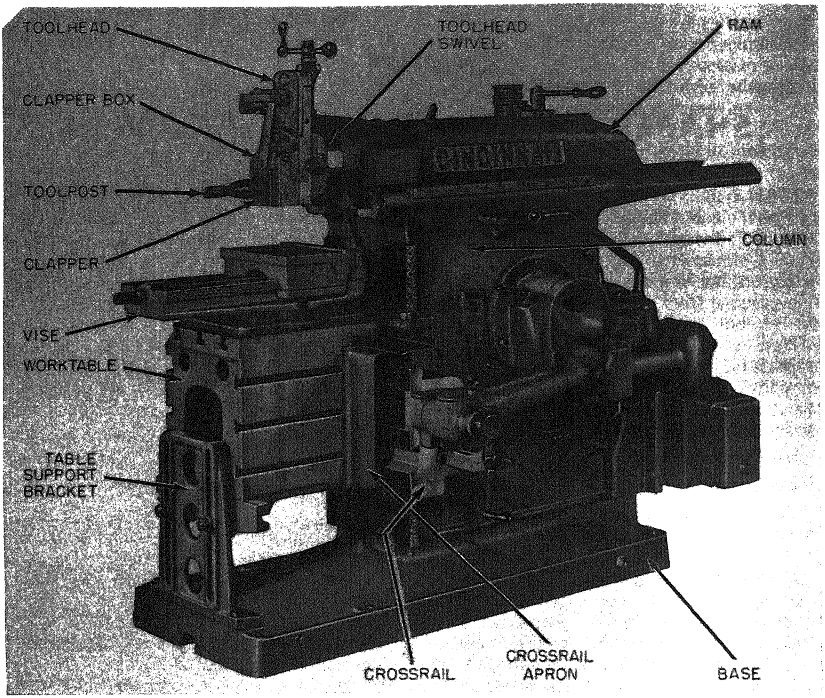
To perform the variety of jobs you will be required to do using the shaper, you must know the construction and operation of the main components. Those components are the main frame assembly, drive assembly, crossrail assembly, toolhead assembly, and table feed mechanism. (See fig. 12-2.)

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 receive the cross-rail.

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



28.219X

Figure 12-1.—Standard shaper.

motion of the ram. By using the adjustments provided, you can increase or decrease the length of stroke of the ram, and can 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 has a freesliding fit in the rocker arm. If you center the crankpin (and therefore the sliding block) on 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 the rocker arm to move. This motion is transferred to the ram through the ram linkage and starts the reciprocating motion of the ram. The distance the crankpin is set off center determines the length of stroke of the tool.

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 machines you are using.

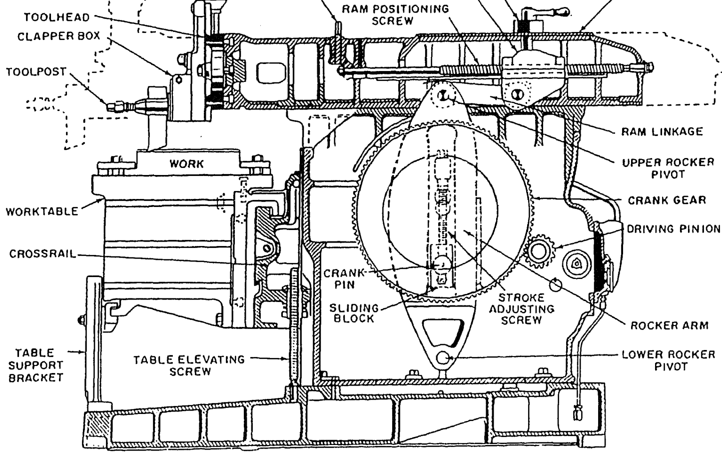
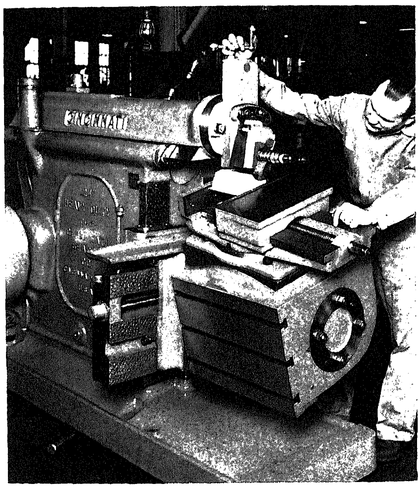


Figure 12-2.—Cross-sectional view of a crank type shaper.

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).



28.221X

Figure 12-3.—Swiveled and tilted table.

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 feed drive wheel rotates, the crankpin (which can be adjusted off center) causes the rocker to oscillate. The straight face of the pawl pushes on the back side of a tooth on the ratchet wheel, turning the ratchet wheel and the feed screw. The back face of the pawl is cut at an angle to ride over one or more teeth as it is rocked in the 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

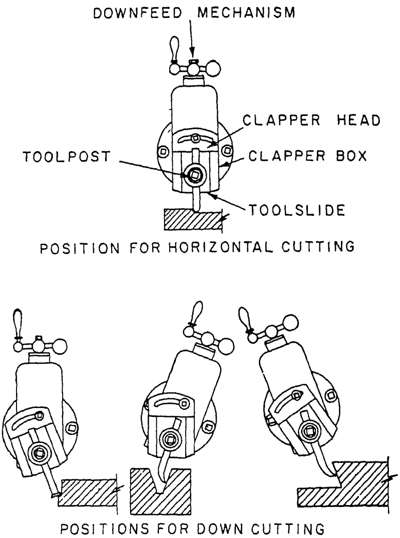
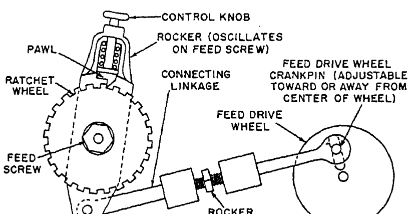
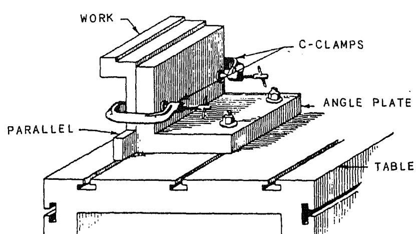


Figure 12-5.—Toolhead assembly in various positions.





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 have the jaws set either parallel with the stroke of the ram or at a right angle to the stroke. See that the vise is free from any obstruction that might keep the work from seating properly. Remove burrs and rough edges on the vise and chips left from previous machining before starting to work.

Work can be set on parallels so the surface to be cut is above the top of the vise. Shaper hold-downs 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 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 (fig. 12-7):

1. Right-hand, straight, and left-hand toolholders, which may be used for the majority of common shaper and planer operations.

2. Gang toolholders, which are especially adapted for surfacing large castings. With a gang toolholder you make multiple cuts with each

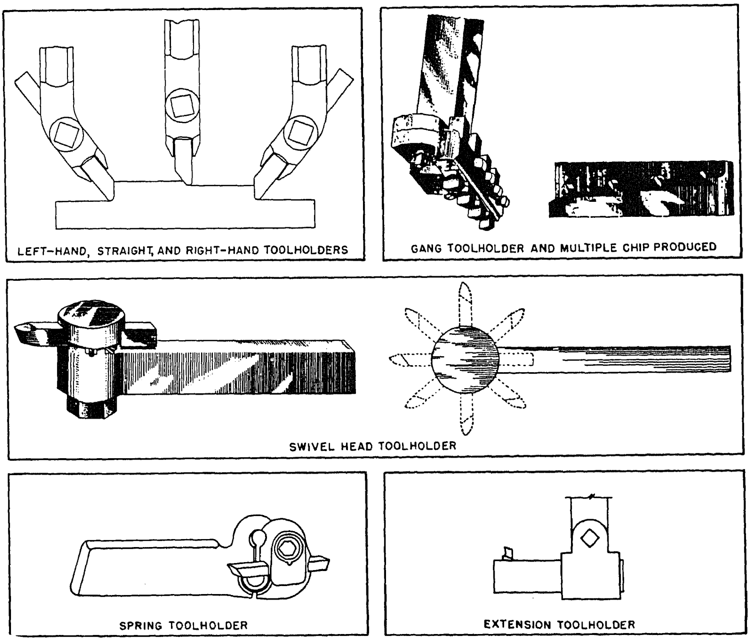


Figure 12-7.—Toolholders.

forward stroke of the shaper. Each tool takes a light cut and there is less tendency to "break out" at the end of a cut.

3. Swivel head toolholders, which 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, which have a rigid U-shaped spring that lets the holder cap absorb a considerable amount of vibration. A spring toolholder is particularly good for use with formed cutters, which have a tendency to chatter and dig into the work.

5. Extension toolholders, which are adapted for cutting internal keyways, splines, and grooves on the shaper. The extension arm of the holder can be adjusted to change the exposed length and the 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

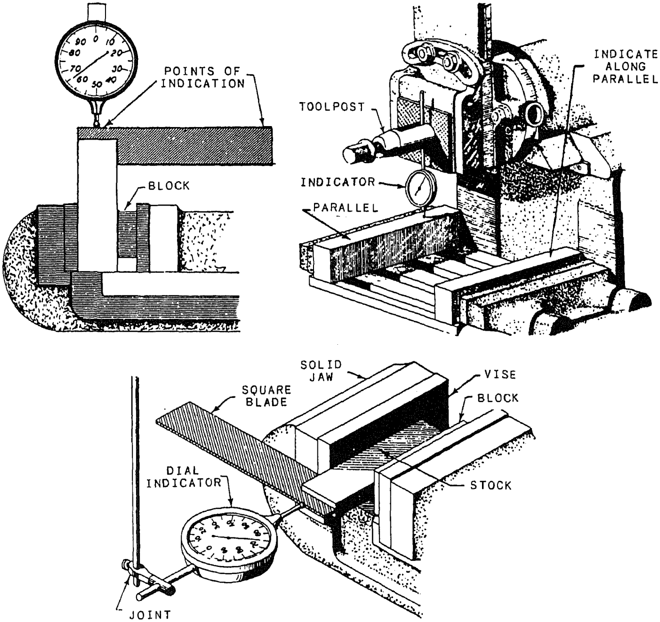
The shaper, like all machines in the machine shop, is not a dangerous piece of equipment if you observe good safety practices. You should read and understand the safety precautions and operating instructions posted on or near a shaper prior to operating it. Some good safety practices are listed here but are intended only to supplement those posted on the machine.

- Always wear goggles or a face shield.
- Ensure that the workpiece, vise, and setup fixture are 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 the front and the rear of the machine.
- Never stand in front of the shaper while it is in operation.
- Avoid touching the tool, the clapper box, or the workpiece while the machine is in operation.
- Never remove chips with your bare hand; always use a brush or a piece of wood.
- Keep the area around the machine clear of chips to help prevent anyone from slipping and falling into the machine.
- Remember: **SAFETY FIRST, ACCURACY SECOND, 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, you should take the following precautions:

- 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 the tool bit does not extend more than about 2 inches below the clapper box.
- Be sure the piece of work is 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 a 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.



28.226

Figure 12-8.—Squaring the table and the vise.

Speeds and Feeds

Setting up the shaper to cut a certain material is similar to setting up other machine tools, such as drill presses and lathes. First, you have to determine the approximate required cutting speed and then you have to determine and set the necessary machine speed to produce your desired cutting speed. On all of the machine tools we discussed in the previous chapters, cutting speed was directly related to the speed (rpm) of the machine's spindle. You could determine what spindle rpm to set by using one formula for all brands of a particular type of machine. Setting up a shaper is slightly different. You still relate cutting speed to machine speed through a formula, but the formula that you use depends on the brand of machine that you operate. This is because some manufacturers use a slightly different formula for computing cutting speed than others. To determine what specific formula to use for your machine, consult the operator's manual provided by the manufacturer.

The following discussion explains basically how the operation of a shaper differs from the operations of other machine tools. It also explains how to determine the cutting speeds and related machine speeds for a Cincinnati shaper.

Whenever you determine the speed of the shaper required to produce a particular cutting speed, you must account for the shaper's reciprocating action. This is because the tool only cuts on the forward stroke of the ram. In most shapers the time required for the cutting stroke is 1 1/2 times that required for 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 consumes 2/5 of the time. The formula for determining required machine strokes contains a constant that accounts for this partial time consumption by the cutting stroke.

To determine a cutting stroke value to set on the shaper speed indicator, first select a recommended cutting speed for the material you plan to shape from a chart such as the one shown in table 12-1.

After you have selected the recommended cutting speed, determine the ram stroke speed by using the formula shown below (remember, your machine may require a slightly different formula):

$$SPM = \frac{CS}{0.14 \times LOS}$$

Where: SPM = strokes of the ram per minute

CS = cutting speed in feet per minute

LOS = length of stroke in inches

0.14 = constant that accounts for partial ram cycle time and that converts inches to feet

When you have determined the number of strokes per minute, set it on the shaper by using the gear shift lever. A speed (strokes) indicator plate shows the positions of the lever for a variety of speeds. Take a few trial cuts and adjust the ram speed slightly, as necessary, until you obtain the desired cut on the work.

If after you have adjusted the ram speed, you want to know the exact cutting speed of the tool, use the formula:

$$CS = SPM \times LOS \times 0.14$$

The speed of the shaper is regulated by the gear shift lever. 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 the work and the 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.

Horizontal feed rates of 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. Therefore, when you select feeds, you must rely on past experience and common sense. Generally, for making roughing cuts on rigidly held work, set the feed 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 you reach a desirable feed rate.

Shaping a Rectangular Block

An accurately machined rectangular block has square corners and opposite surfaces that are parallel to each other. In this discussion, faces are the surfaces of the block that have the largest surface area; the ends are the surfaces that limit the length of the block; and the sides are the surfaces that limit 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 are machined in the first setup. The opposite face and end are machined in the second setup. The sides are machined in two similar but separate setups. For both setups, the vise jaws are aligned at a right angle to the ram.

To machine a rectangular block from a rough casting, proceed as follows:

1. Clamp the casting in the vise so a face is horizontally level and slightly above the top of the vise jaws. Allow one end to extend out of the side of the vise jaws enough so you can take a cut on the end without unclamping the casting. Now feed the cutting tool down to the required depth and take a horizontal cut across the face. After you have machined the face, readjust the cutting tool so it will cut across the surface of the end that extends from the vise. Use the horizontal motion of the ram and the vertical adjustment of the toolhead to move the tool across and down the surface of the end. When you have machined the end, check to be sure that it is square with the machined face. If it is not square, adjust the toolhead swivel to correct the inaccuracy and take another light finishing cut down the end.

2. To machine the second face and end, turn the block over and set the previously machined face on parallels (similar to the method used in step 1). Insert small strips of paper between each corner of the block and 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, machine the second face and end to the correct thickness and length dimensions of the block.

Type of metal	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

3. To machine a side, open the vise jaws so the jaws can be clamped on the ends of the block. Now set the block on parallels in the vise with the side 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 and make the cut.

4. Set up and machine the other side 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 you would to machine a flat surface parallel with the table.

2. Swivel the toolhead (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. Use the horizontal feed control to feed the work into the tool and to control the depth of cut (thickness of the chip). (Because the tool is fed manually, be careful to feed the tool toward the work only during the return stroke.)

4. Set up and machine the other side as described in step 3.

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 (fig. 12-9, view A). Figure 12-9 also shows holes of the same diameter as the keyway width and slightly deeper than the key drilled into the shaft. These holes are required to provide tool clearance at the

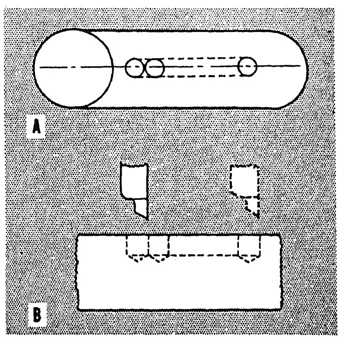


Figure 12-9.—Cutting a keyway in the middle of a shaft.

beginning and end of the cutting stroke. The holes shown in figure 12-9 are 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:

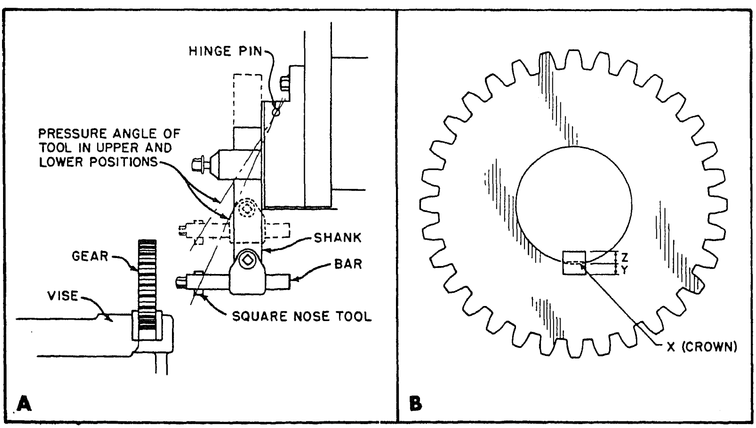
1. Lay out the centerline, the keyway width, and the clearance hole centers as illustrated in part A of figure 12-9. Drill the clearance holes.
2. Position the shaft in the shaper vise or on the worktable so that it is parallel to the ram. Use a machinist's square to check the centerline on the end of 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, to provide a keyway that is centered on the centerlines of the shaft.
3. Adjust the stroke and the position of the ram, so 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 the tool will remain in the keyway slot.) (See view B of fig. 12-9.)
4. Position the work under the cutting tool so that the tool's 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 you obtain the proper width.
5. Start the shaper and, using the toolhead slide, feed the tool down to the depth required, as indicated by the graduated collar.

Shaping an Internal Keyway

To cut an internal keyway in a gear, you will have to use extension tools. These tools 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 practical below the clapper block, rather than in the position shown by the dotted lines in view A of figure 12-10. The pressure angle associated with the toolholder in the upper position may cause the pressure of the cut to open the clapper block slightly and allow the tool to leave the cut.



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 view A of figure 12-10, center the tool within the layout lines in the usual manner, and make the cut to the proper depth while feeding the toolhead down by hand. Within the setup in an inverted position, center the tool within 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 view B of figure 12-10. In cutting a keyway in the gear, the downfeed micrometer collar is set to zero at the point where the cutting tool first touches the edge of the hole. The crown, X, is first removed from the shaft to produce a flat whose width is equal to the width of the key. Then the cut is made in the shaft to depth Z. The distance of "Y" plus "Z" is equal to the height of the key that is to lock the two parts together. (See fig. 12-10.).

Shaping Irregular Surfaces

You can machine irregular surfaces by using form ground tools and by hand feeding the cutting tool vertically while using power feed to move the work horizontally. An example of work that you might shape by using form tools is a gear rack. You can shape work such as concave and convex surfaces by using the toolhead feed. When you machine irregular surfaces, you have to pay close close attention because you control the cutting tool manually. Also in this work you should lay out the job before you machine it to provide reference lines. You should also take roughing cuts to remove excess material to within 1/16 inch of the layout lines.

You can cut **RACK TEETH** on a shaper as well as on a planer or a milling machine. During the machining operation, you may either hold the work in the vise or clamp it directly to the worktable. After you have mounted and positioned the work, rough out the tooth space in the form of a plain rectangular groove with a roughing tool, then finish it with a tool ground to the tooth's finished contour and size.

operations 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 the tool is centered on 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 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 the tool is properly aligned with the first tooth space that you 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 you have finished all of the teeth.

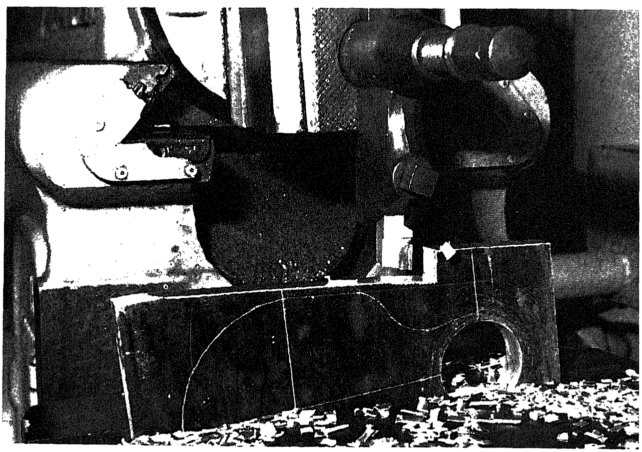
Irregular surfaces commonly machined on the shaper have both **CONVEX** and **CONCAVE** radii. On one end of the work, lay out the contour of the finished job. When you shape to a scribed line, as illustrated in figure 12-11, it is good practice to rough cut to within 1/16 inch of the line. You can do 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 of the line by the breaking of the chip. Starting at the right-hand side of the work, set the automatic feed so the horizontal travel is rather slow and, feeding the tool vertically by hand, take the finishing cuts to produce a smooth contoured surface.

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 done by tilting the vertical ram (fig. 12-12), which reciprocates up and down, to the required angle. Although different models of machines will have their control levers in different locations, 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 table, or held by special fixtures. The square hole in the center of a valve handwheel is an example of work that can be done on a machine of this type. The sides of the hole are cut on a slight angle to match the angled sides 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 between the straight and angular surfaces.

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, the reciprocating motion of planers is provided by the worktable (platen), while the cutting tool is fed at a right



28.227

Figure 12-11.—Shaping irregular surfaces.

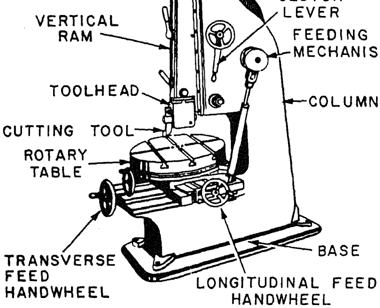


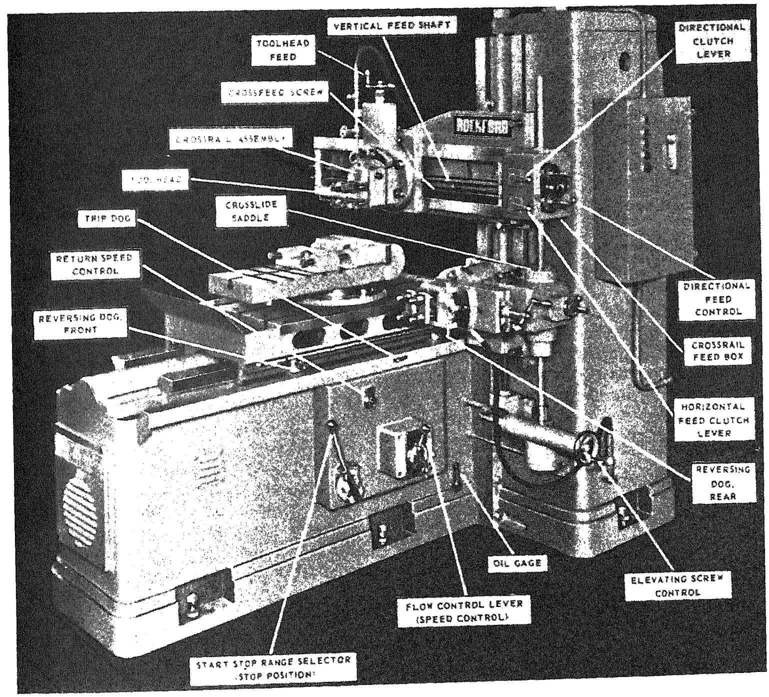
Figure 12-12.—Vertical shaper.

table makes a quick return to bring the work into position for the next cut. The size of a planer is determined 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 is that work that is too wide to pass



28.230X

Figure 12-13.—Open side planer.

between the uprights of a double housing machine may be planed.

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 the 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 that supports the entire piece of machinery. On the upper surface of the bed are the ways on which the planer table rides.

The table is a cast iron flat 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 usually a gear train or a hydraulic 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 support and carry the crossrail.

The crossrail serves as the rigid support for the toolheads. The vertical and horizontal feed screws on the crossrail enable you to adjust the machine for various size pieces of work.

The toolhead is similar to that of the shaper in construction and operation.

All sliding surfaces subject to wear are provided with adjustments. Keep the gibes adjusted to take up any looseness due to wear.

OPERATING THE PLANER

Before you operate a planer, be sure 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. You should, however, refer to the manufacturer's technical manual for the machine you are using for specific operating instructions. The following sections contain general information on planer operation.

Table Speeds

The table speeds are controlled by the start-stop lever and the flow control lever (fig. 12-13). Two ranges of speeds and a variation of speeds within each range are available. The speed range (LOW-MAXIMUM CUT or HIGH-MINIMUM CUT) is selected by using the start-stop lever, and the speeds within each range are varied by using the flow control lever. As the flow control lever is moved toward the right, the table speed will gradually increase 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 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 direction of feed (right or left, up or down) of the toolhead is controlled 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 lower clutch lever on the front of the feed box engages the horizontal feed of the toolhead. When the directional lever on the rear of the box is in the down position, the head is fed toward the left. When the directional lever is in the up position, the head is fed toward the right. Shifting the directional lever to the up position gives an upward feed.

The ball crank on top of the vertical slide (toolhead feed) is used to hand feed the toolslide up or down. A graduated dial directly below the crank indicates the amount of travel.

The two square-ended shafts at the end of the crossrail are used to move the toolhead by hand.

on the rear of the feedbox is in the center, or neutral, position, and then turn the shaft. The upper shaft controls vertical movement. The lower shaft controls horizontal movement.

Lock screws on both the cross-slide saddle and the vertical slide enable these slides to be locked in position after the desired tool setting is made.

The planer side head has power vertical feed and hand horizontal feed. The vertical feed, both engagement and direction, is controlled by a lever on the rear of the side head feed box. Vertical traverse is done by turning the square shaft that projects from the end of the feed box. Horizontal movement, both feed and traverse, is done by using 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 form the rear of the rail brace. To move 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 you do any machining.

On machines that have 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. The clamp nuts have the same use on all machines whether manual or power elevation is used.

Holding the Work

The various accessories used in planer or shaper work may make the difference between a superior job and a poor job. There are no set rules on the use of planer accessories for clamping down a piece of work—results will depend on your ingenuity and experience.

One way to hold down work on the worktable is by using clamps. The clamps are attached to the worktable by bolts inserted in the T-slots. Figure 12-14 illustrates a step block used with the clamps shown in figure 5-30. At some time you may have to clamp an irregularly shaped piece of work to the planer table. One way to do this is illustrated in figure 12-15; here an accurately machined step block is used with a gooseneck clamp. Figure 12-16 illustrates correct and incorrect ways to apply clamps.

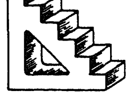


Figure 12-14.—Step block.

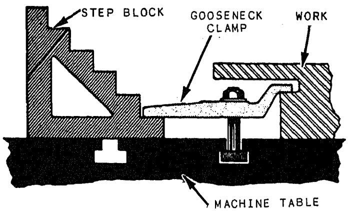


Figure 12-15.—Application of step block and clamp.

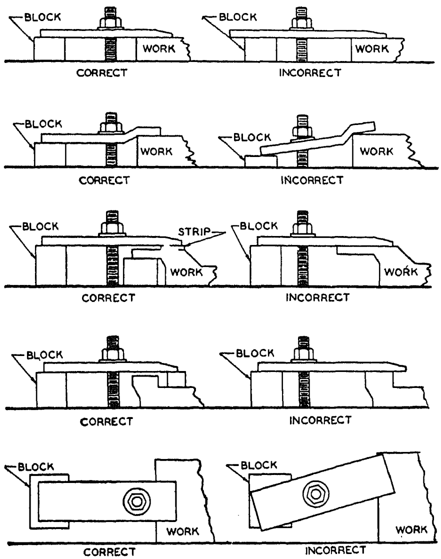


Figure 12-16.—Correct and incorrect clamp applications.

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 pad 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 use the planer as a surface grinder. Basically speaking, it is a matter of 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, be sure you have a thorough understanding of the material presented in chapter 13 of this manual.

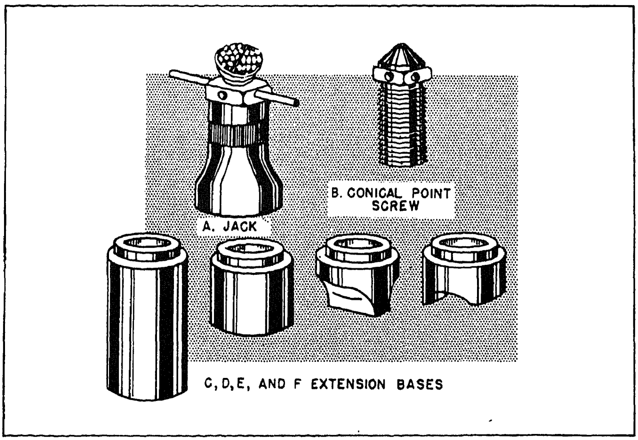
When you have completed the grinding job, you must clean the planer extensively, both inside and out. Filter or change the oil in the hydraulic system prior to further operation. The planer, unlike the surface grinder, has no built-in protection against the grinding particles left by the grinding operation.

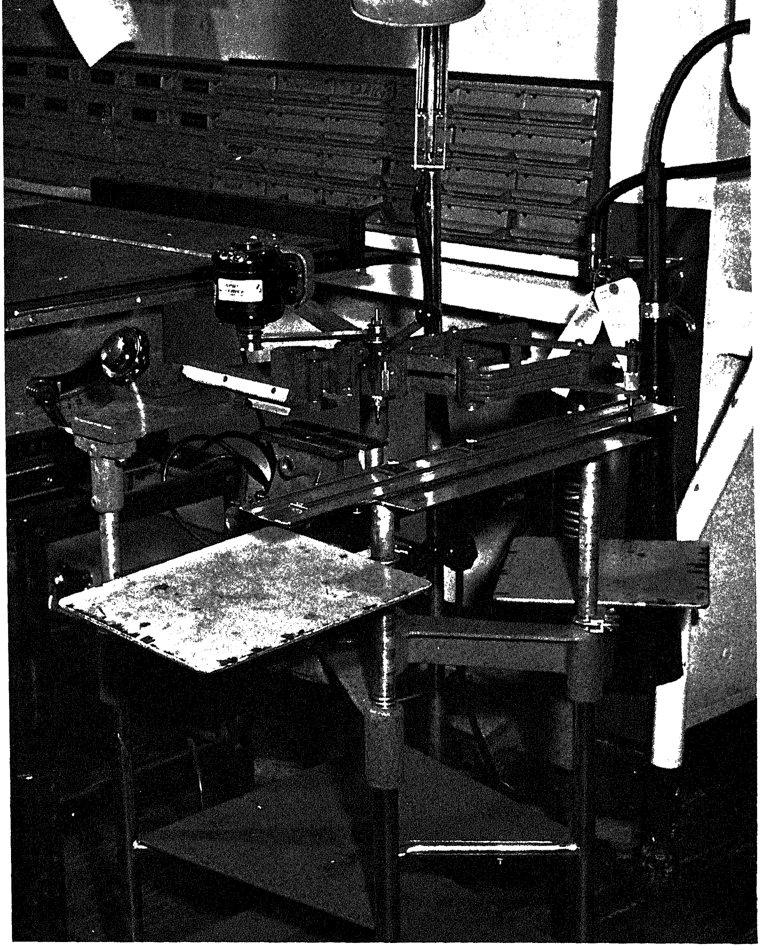
Observe the same safety precautions for the shaper as you do for the planer. Always observe standard machine shop practices.

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 that 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 that you may have to operate. Figure 12-18 shows one model that mounts on a bench or a table top and is used primarily for engraving small items. This particular machine is manufactured by the New Hermes Engraving Machine Corporation. 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 1/7 the size of the pattern.





28.332

Figure 12-18.—Engraving machine.

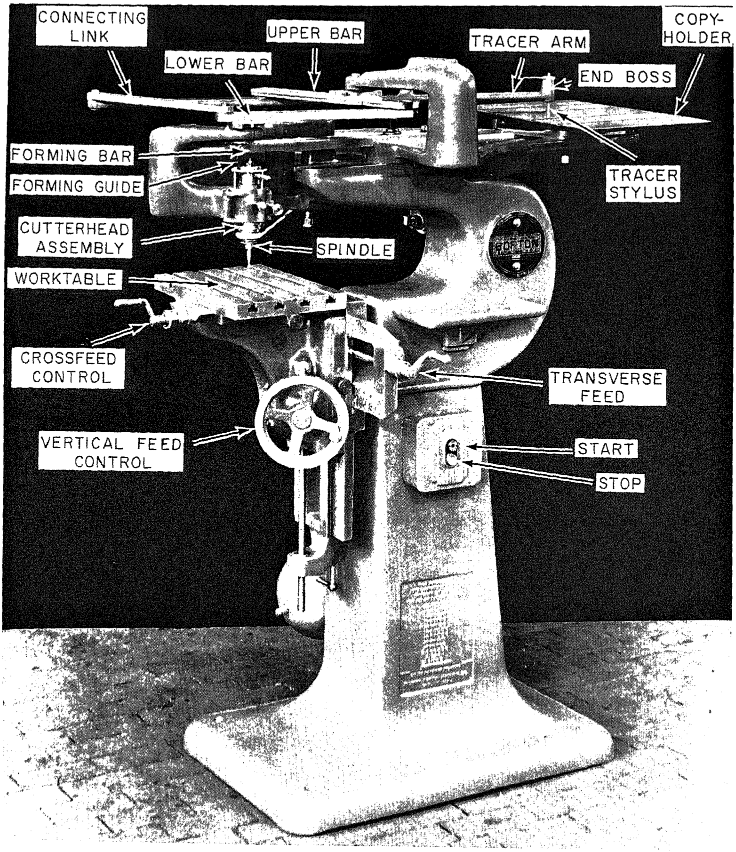
The Gorton 3-U pantograph (figure 12-19) is another engraving machine commonly used by the Navy. The principles of operation and setup procedures for the 3-U machine are similar to those for other models of pantograph type engraving machines. Because of the similarity in operating principles and setup 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



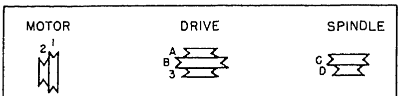
of surrounding machinery, the base should be set on rubber or cork pads.

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. It also has a 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 the spindle enable you to adjust the 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 limited vertical movement from 1/16 inch to 1/4 inch to prevent the cutter from breaking when it feeds into work. A plunger locks the spindle for flat surface engraving or releases it for floating



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

Gorton Pantographs made by FAMCO Machine since 1988

28.235X

Figure 12-20.—Spindle speeds.

is hinged to permit spindle removal from the side, making it unnecessary to disturb any work by lowering the table.

Worktable

The cast iron worktable of the 3-U pantograph engraver measures 8 inches by 12 inches and is flat and highly polished. It has four 3/8-inch T-slots cut parallel to its front edge for mounting a vise or table dogs to hold down a piece of work. Longitudinal feed can move the worktable 10 inches, while the cross feed can move the table 11 inches. Vertical feed of the worktable is 9 3/4 inches.

Copyholder

The copyholder is a steel casting 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 with an engraver is known by various terms—however, the Navy uses the 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.

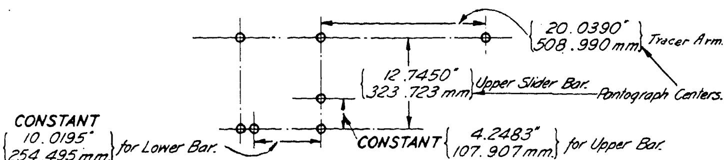
Copy is not self-spacing; therefore, you should adjust the spaces between the characters by inserting suitable blank spacers, which are furnished with each set of copy. Each line, when set in the copyholder, should be held firmly between the clamps.

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 were properly adjusted at 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 the work and the copy parallel 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 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 for engraving.

SETTING THE PANTOGRAPH

The correct setting of the pantograph is determined from the ratio of (1) the size of the



Upper Bar Constant = 12.7450" ÷ 3 (2nd. Reduction + 1). Lower Bar Constant = 20.0390" ÷ 2nd Reduction.

EXAMPLE: REQUIRED THE SETTINGS IN INCHES FOR REDUCING 4 TO 1.

For LOWER Slider Bar

Required Reduction 4.0) 20.0390" → 5.0097" (Tracer Arm Centers)

Lower Bar Constant 10.0195" − 5.0097" (Subtract from) = 5.0098"

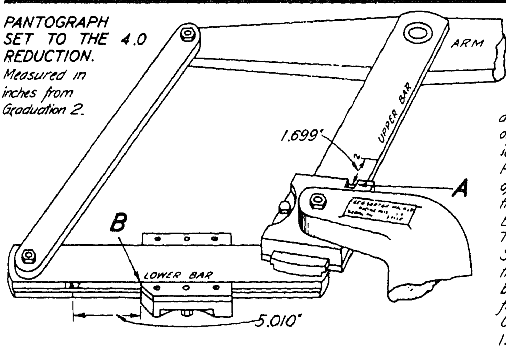
Distance to set Index Edge on Lower Slider Bar Head from Graduation 2. See below sketch.

For UPPER Slider Bar.

First divide the Upper Slider Bar Center distance 12.7450" by the Reduction Required plus a constant of 1.

1.0 + 4.0 (Reduction Required) = 5.0) 12.7450" (Upper Slider Bar Centers) → 2.5489"

Subtract from → 4.2483" (Upper Bar Constant) − 2.5489" = Distance → 1.6994" to set Index Edge on Upper Slider Bar Head from Graduation 2. See below sketch.



To set the Pantograph for any desired Special Scale of Reduction as per above Formula or as per Schedule of various Reductions given. Place the Bevelled Index Edges of the Sliders away from the Lines marked 2 on the Bars, the Distances required: THUS – As shown in the Sketch for the Reduction 4.0 the Lower Slider Block must be set as at 'B' 5.010" from the Line 2 and the Upper Slider Block as at 'A' 1.699" from its Line 2.

work to the size of the copy layout, or (2) 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:

$$10 \div 2 = 5 \text{ inches}$$

For this job, set the slider blocks at 5 inches.

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.

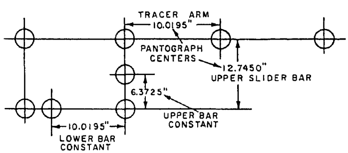
All settings are measured from the first reduction marking on the upper and lower arms. On the model 3-U pantograph, reductions are measured from the line marked 2 on the upper arm, and NOT the line marked 1. To accurately set special reductions use a hundredth-inch scale.

After you have set a special reduction, 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 an angle, recalculate the setting and reset the sliding blocks. If the point still runs off, loosen either of the slider blocks and tap it 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 the 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.



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"

STEP 3. THE RESULT IS THE DISTANCE TO SET INDEX EDGE ON UPPER SLIDER BAR HEAD FROM GRADUATION 1.

$$6.3725" - 5.098" = 1.2745"$$

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

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

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

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	182.81	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

Gorton Pantographs made by FAMCO Machine since 1988

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 you feed the cutter through the work. Since the 3-U engravers are fed manually, the rate of feed is subject to a wide variation by individual operations; this will affect the spindle speeds used.

Run the cutters at highest speeds possible without burning them, and remove stock with several light, fast cuts rather than one heavy cut at slower spindle speeds. When you cut steel and other hard materials, start with a slow speed and work up to the fastest speed the cutter will stand without losing its cutting edge. Sometimes you may have to sacrifice cutter life to obtain the smoother finish possible at higher speeds. With experience you will know when the cutter is running at its 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 all the way to the point. Stoning off the flat with a small fine oilstone will make the cutting edge keener.

You can make a cutter run almost perfectly by sharpening it in the spindle in which it will run. Most pantograph machines have a provision for removing the cutter spindle from the machine and placing it in a V-block toolhead on the cutter grinder. This will allow you to grind the cutter to the desired shape without removing it from the cutter spindle.

Grinding Single-Flute Cutters

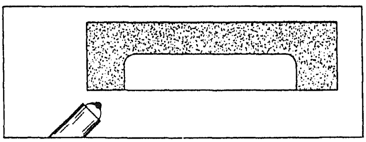
Before grinding cutters, true up the grinding wheel with the diamond tool supplied with the

Table 12-3.—Cutter Speeds

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

toolhead at approximately the same relation to 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, your maximum cut should be 0.001 to 0.002 inch. If the diamond fails to cut freely, turn it slightly in the toolhead to present an unused portion of the diamond to the wheel.

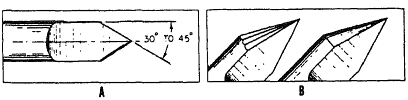
ROUGH AND FINISH GRINDING A CONICAL POINT.—Set the grinder toolhead to the desired cutting edge angle (fig. 12-24A). This angle usually varies from 30° to 45°, depending on the work desired. For most sunken letter or design engraving on metal or bakelite plates, a 30° angle is used. Now place the cutter in the toolhead and rough grind it to approximate size by swinging it across the wheel's face. Do not rotate the cutter while it is in contact with the face of the wheel but swing it straight across, turning it slightly **BEFORE** or **AFTER** it makes contact with the 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 of wheel marks.



Gorton Pantographs made by FAMCO Machine since 1988

28.238X

Figure 12-23.—Position of diamond for truing a grinding wheel.



Gorton Pantographs made by FAMCO Machine since 1988

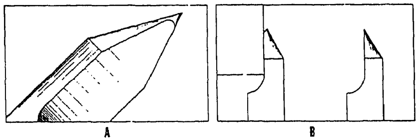
28.239X

Figure 12-24.—Grinding a conical point: (A) Cutter angle. (B) Rough and finished conical shape.

The next operation is to grind the flat to center. For very 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 THE 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.

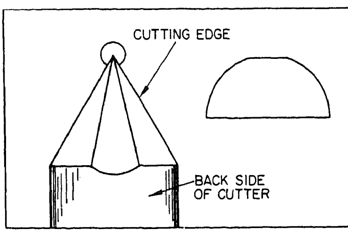
Gently feed the cutter into the face of the wheel. Do not rotate the cutter. Hold the back (round side) of the conical point against the wheel. Rock the cutter continuously across the wheel's face, without turning it, until you grind a flat that runs out exactly at the cutter point (fig. 12-26). Check this very carefully, with a magnifying glass



Gorton Pantographs made by FAMCO Machine since 1988

28.240X

Figure 12-25.—Grinding the flat. (A) Flat not ground to center. (B) Flat ground to center.



Gorton Pantographs made by FAMCO Machine since 1988

28.241X

Figure 12-26.—First operation in grinding clearance.

if necessary, to be sure you have reached the point with this flat. Be extremely careful not to go beyond the point.

The next step is to grind away the rest of the stock on the back of the conical side to the angle of the flat, up to the cutting edge. Rotate the conical side against the face of the wheel and remove the stock as shown in figure 12-24B. Be extremely careful not to turn the cutter too far and 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 work.

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

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

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

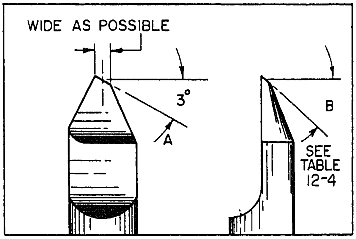
Table 12-6.—Clearance Angles for 3- and 4-Sided Cutters

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

when the point wears down, the cutter will immediately fail 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 approximately 3°; this angle causes the cutter to bite into the work like a drill when it is fed down. Angle B (fig. 12-27) varies, depending on the material to be engraved. Use table 12-4 as a guide in determining angle B.

Grinding Square-Nose Single-Flute Cutters

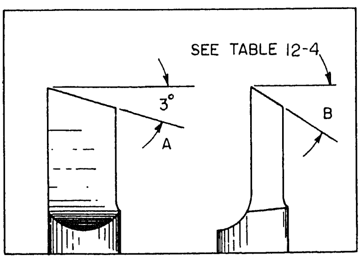
A properly ground square-nose single-flute cutter should be similar to the illustration in figure 12-28. When square-nose cutters are ground, they should be tipped off in the same manner as described in connection with figure 12-27. All square-nose cutters have peripheral clearance ground back of the cutting edge. After grinding the flat to center (easily checked with a micrometer), grind the clearance by feeding the cutter in the required amount toward the wheel and turning the cutter until you have removed all stock from the back (round side), up to the cutting edge. Table 12-5 provides information on chip clearance for various sized cutters.



Gorton Pantographs made by FAMCO Machine since 1988

28.242X

Figure 12-27.—A tipped off cutter.



Gorton Pantographs made by FAMCO Machine since 1988

28.243X

Figure 12-28.—Square-nose cutter with a properly ground tip.

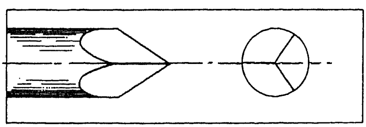
Grinding Three- and Four-Sided Cutters

Three- and four-sided cutters (see fig. 12-29) are used for cutting small steel stamps and for small engraving where a very smooth finish is desired. The index plate on the toolhead collet spindle has numbered index holes for indexing to grind three-and four-sided cutters.

Set the toolhead for the desired angle. Plug the pin in the index hole for the desired number of divisions and grind the flats. Now, without loosening the cutter in the toolhead collet, reset the toolhead to the proper clearance angle. Clearance angles are listed in table 12-6.

PANTOGRAPH ATTACHMENTS

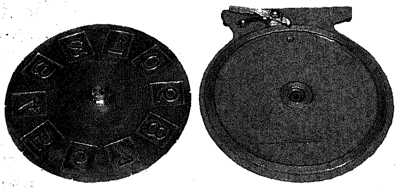
Some attachments commonly used with the pantograph engraving machine are: copy dial holders, indexing attachments, forming guides and rotary tables. The use of these attachments extends the capabilities of the pantograph engraving machine from flat, straight line engraving to include circular work, cylindrical work, and indexing.



Gorton Pantographs made by FAMCO Machine since 1988

28.244X

Figure 12-29.—Three-sided cutter.



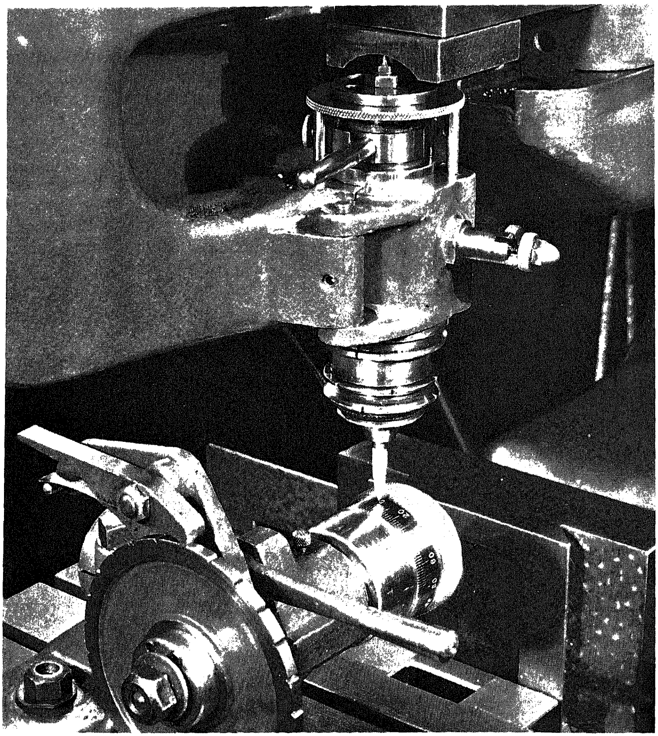
Gorton Pantographs made by FAMCO Machine since 1988

28.245X

Figure 12-30.—Copy dial holder and plate.

The copy dial holder shown in figure 12-30 is used instead of the regular copy-holder when a circular copy plate is used. This holder has a spring-loaded indexing pawl, which is aligned with the center pivot hole. This pawl engages in the notches in a circular copy plate 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



Gorton Pantographs made by FAMCO Machine since 1988

28.246X

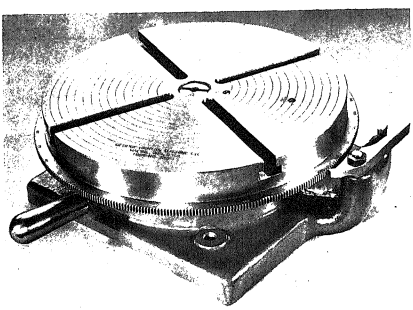
for any number of divisions available on the plate. Figure 12-31 shows a micrometer collar being held for graduation and engraving.

A forming guide (sometimes called a radius plate) is used to engrave cylindrical 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.

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, the following setup can be used to engrave 26 similar workpieces with a single letter.



Gorton Pantographs made by FAMCO Machine since 1988

28.247X

Figure 12-32.—A rotary table.

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 the plate will be steady while you trace each character.
3. Set the machine for the required reduction and speed, and adjust the worktable so the spindle is in position over the workpiece.
4. Clamp the first workpiece in place on the worktable. (The aligning stops, step 1, ensure 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 you have finished the first piece, remove it from the machine. Do not change the alignment of the aligning stops (step 1), the worktable, or the 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 workpieces, indexing the plate to the next character, engraving, and removing the work, until you have finished the job.

ENGRAVING A GRADUATED COLLAR

To engrave a graduated collar, as shown in figure 12-31, use a forming guide and indexing attachment. You can also use the circular copy plate to speed up the numbering process. After you have engraved each graduation, index the work to the next division until you have finished the graduating. When you engrave numbers with more than one digit, offset the work angularly by rotating the work so the numbers are centered on the required graduation marks.

ENGRAVING A DIAL FACE

Use a rotary table and a circular copy plate to engrave a dial face, such as the one 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 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 the center of the dial is directly under the point of the cutter.

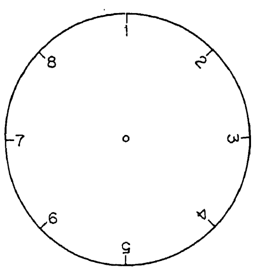


Figure 12-33.—A dial face.

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 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 mechanism.

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, rotate the table 45°; engrave this mark and continue until the circle is graduated. You will now be back to the starting point.

NOTE: Do not move the circular copy plate 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 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 you have engraved all the numbers. If two (or more) digit numbers are required, offset the dial as previously described.

10. To engrave the numbers shown on the left side of the dial in figure 12-33, rotate the copy plate to the required number and then, using the cross feed 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.

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 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 several classes of each type of grinder. The SURFACE grinder may have either a rotary or a reciprocating table, and either a horizontal or vertical spindle. *Cylindrical* grinders may be classified as plain, centerless, or internal grinders. The *tool and cutter* grinder is basically a 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 the 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.

INFEED 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.

$$\text{fpm} = \frac{(\text{cir.} \times \text{rpm})}{12}$$

$$\text{fpm} = \frac{\pi \times D \times \text{rpm}}{12}$$

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 provides a margin of safety. Usually, the wheel will have better cutting action at a lower speed than that listed by the manufacturer as a maximum speed.

One method of determining the proper wheel speed is to set the wheel speed between the minimum and maximum speeds recommended by the wheel's manufacturer. 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).

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 the material of which the work is made, heat treatment, wheel and work speed, and condition of the machine. Roughing cuts 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 fluids used in other machine tool operations. They are 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

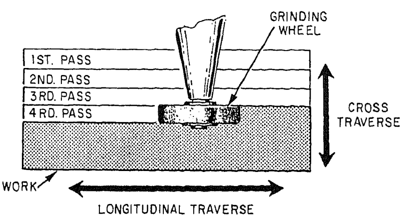


Figure 13-1.—Surface grinding a flat workpiece.

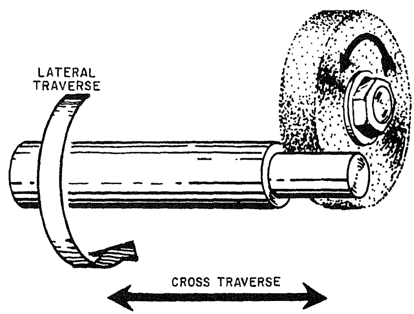


Figure 13-2.—Surface grinding a cylindrical workpiece.

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 property of a cutting fluid is its lubricating ability. In grinding, however, the primary property is the cooling ability, with the lubricating ability 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 of water to one part of oil. This mixture will generally 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. Therefore, using mineral oil allows you to select a finer grit wheel and requires fewer wheel dressings.

When you select 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 chips from sticking to the work

- 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 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 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.

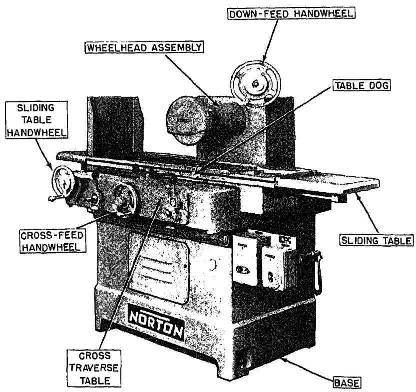
- It should be safe, particularly in regard to fire and accident hazards.

- 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



28.249X

Figure 13-3.—Surface grinder (planer type).

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.

The base is 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 table. 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 table in any position and regulate the speed of movement of the table within limits. These valves provide a constant pressure in the hydraulic system, allowing you to stop the feed without securing the system.

CROSS TRAVERSE TABLE

The ways on which the cross traverse table are mounted are parallel to the spindle of the wheelhead unit. This allows the entire width of the workpiece to be traversed under the grinding wheel.

Power feed is provided by a piston in a power cylinder fastened to the cross traverse table. 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 (discussed below); 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 limit 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.

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 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 power feed of the table is similar to that of the cross traverse table. During manual traverse, a pinion turned by a handwheel engages a rack attached to the bottom of the sliding table.

During manual operation of the sliding table, table stop dogs limit the length of stroke. When power feed is used, table reverse dogs reverse the direction of movement of the table 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 by turning 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 movement is not usually power fed because the depth of cut is quite small and any large movement is needed only in setting up the machine. The adjusting mechanism is quite sensitive; the depth of cut can be adjusted in amounts as small as 0.0001 inch.

WORKHOLDING DEVICES

Since 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 non-magnetic 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. For either chuck, work will not remain in place unless it contacts 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.

A 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 deep scratches and low spots caused by wear. If you remove the chuck from the grinder, be sure to regrind the chuck table when you replace the chuck to ensure that the table 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

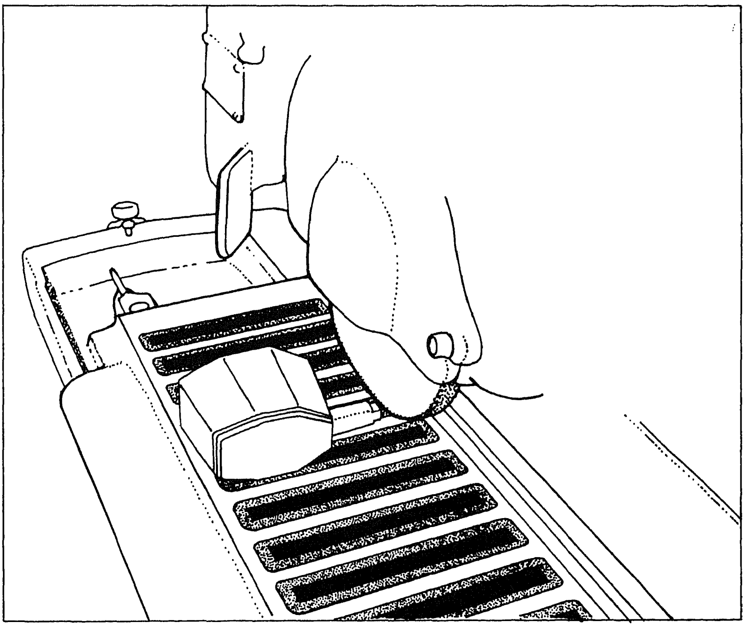


Figure 13-4.—Magnetic chuck used for holding a tool grinding jig.

a depth of cut that does not exceed 0.002 inch. Use ample coolant to help reduce heat and flush away the grinding chips.

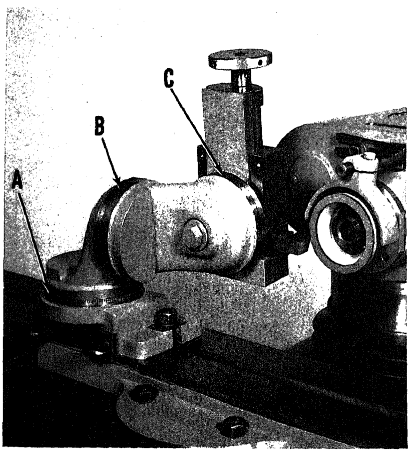
Universal Vise

The universal vise (fig. 13-5) can be used for setting up work, such as lathe tools, so 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 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 the one mounted 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.



28.251

Figure 13-5.—Universal vise (mounted on a tool and cutter grinder). (A) Base swivel; (B) Intermediate swivel; (C) Vise swivel.

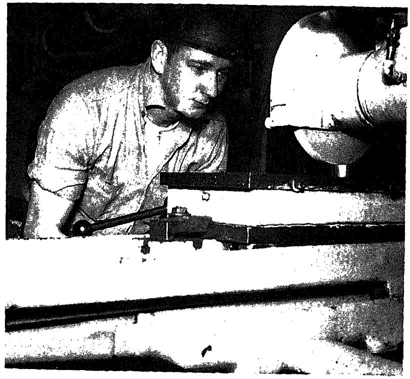


Figure 13-6.—Grinding a spacer on a surface grinder.

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 the sliding table will move the work clear of the wheel at each end of the stroke. If you will be using power traverse, set the table reverse dogs.
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 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 graduated 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 the work speed.

10. Stop the longitudinal and cross traverses 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. For this job, 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 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.

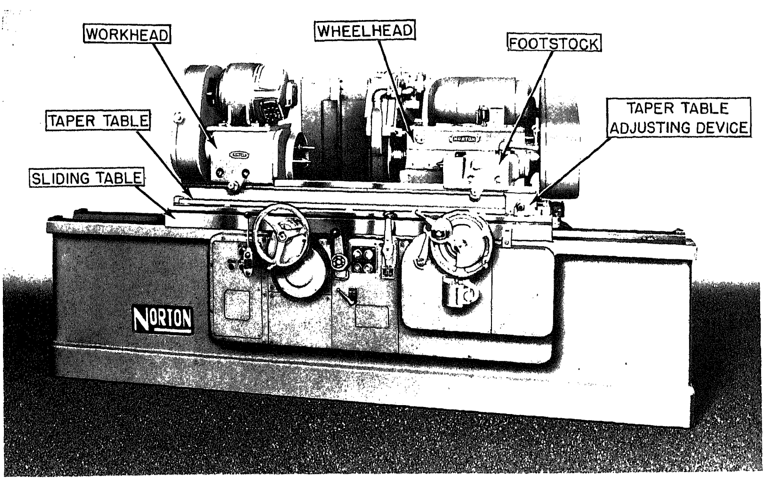
The universal vise can be used on a surface grinder for very accurate grinding of lathe cutting tools such as threading tools. For example, to grind an Acme threading tool, set the vise swivel at 14 1/2° from parallel to the table. Set the intermediate swivel to the clearance angle. Set the base swivel so 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 it 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 the angles of the tool bit are formed properly.

When you use either method for grinding tool bits, check the tool bit occasionally with an appropriate gauge until you have obtained the correct dimensions. To save time, rough grind the tool bit to approximate size on a bench grinder before you set the tool bit in the jig.

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 the surface grinder, 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.



28.252

Figure 13-7.—Cylindrical grinder (with workhead and footstock mounted).

As in the surface grinder, the base of this machine contains a hydraulic power unit and a coolant system. Longitudinal ways support the sliding table. Horizontal ways (at right angles to the longitudinal ways) permit the wheelhead to move 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 the longitudinal ways. 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 for the workpiece to meet the requirements of the job.

A chuck, a center, or a faceplate can be used to mount 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 to provide a way to set the work for grinding relatively large taper angles.

WHEELHEAD

The wheelhead of a cylindrical grinder moves on the 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 the 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 you plan to grind an angle or a taper.)
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 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).

a run-in disengage the power traverse, stop the workhead motor and wheelhead rotation, and check the workpiece for taper. Make any changes required. (If you are using the taper table attachment and an adjustment is necessary at this point, dress the wheel again).

We have not provided specific information on how to set the various controls and speeds because there is a variation for each machine. Check the manufacturer's technical manual for your machine for this information.

TOOL AND CUTTER GRINDER

The tool and cutter grinder (fig. 3-8) has a combination of the features of the plain cylindrical grinder and the planer type surface grinder. 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 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.

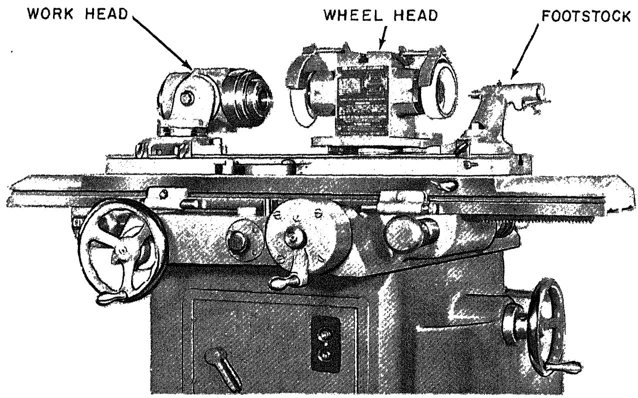


Figure 13-8.—Tool and cutter grinder (workhead and footstock).

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 it 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 and 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.

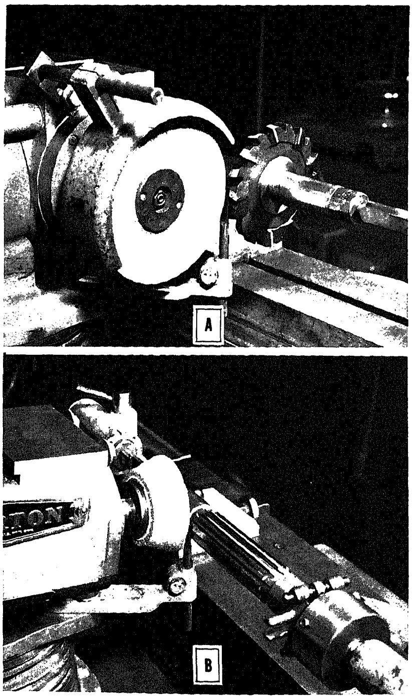


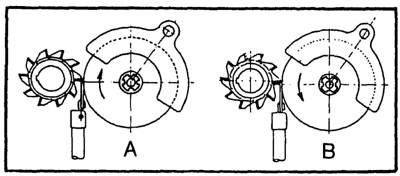
Figure 13-9.—Tool grinding setups on a tool and cutter grinder. (A) Straight wheel grinding a milling cutter. (B) Cup wheel grinding a reamer.

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.

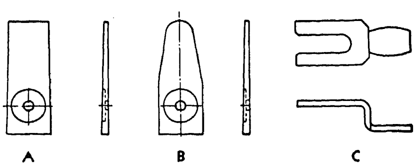
When you use the straight wheel, set the spindle parallel to the table. When you use a flaring cup wheel, turn the spindle at an angle of 89° to the table. This provides the necessary clearance for the trailing edge of the grinding wheel as it is traversed along the cutter.

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



28.257X

Figure 13-10.—Direction of wheel rotation. (A) Toward the cutting edge. (B) Away from the cutting edge.



126.46X

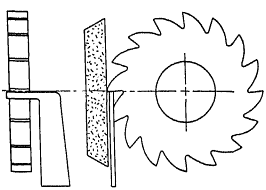
Figure 13-11.—Typical tooth rest blades.

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 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. As a result, succeeding teeth have less metal removed and the teeth gradually increase in size.

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.



126.47X

Figure 13-12.—L-shaped tooth rest blade.

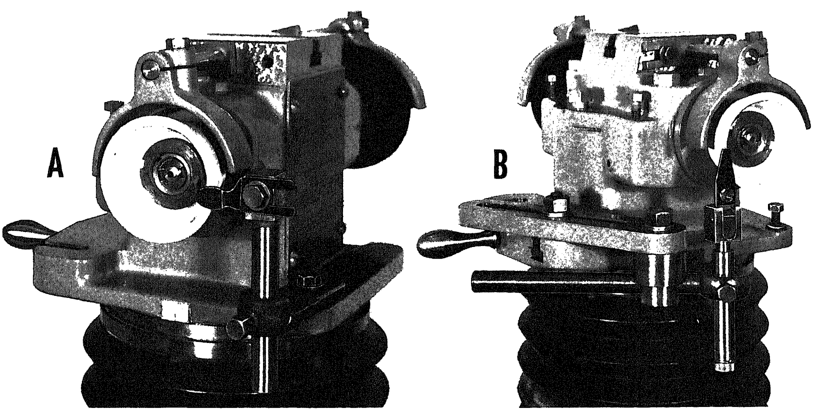
To ensure a good cutting edge on the cutter, there must be a good finish on the clearance angle; therefore, you will occasionally need to dress the grinding wheel. Use the wheel truing attachment for this operation and for the initial truing and dressing operation on the wheel.

Tooth Rest Blades and Holders

Tooth rest blades are not carried in stock, so they must be made in the shop. Once you understand the requirements for the blades, you will be able to readily fabricate various shapes to suit the types of cutters you will 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 cutters, end mills, straight-fluted reamers, or any straight-fluted cutter. The rounded tooth rest blade (B in fig. 13-11) is used for helix cutters, 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. You can make other shapes of tooth rest blades to fit the specific type of cutter or the cutter grinder you are using.

Holders for the tooth rest blades may be either plain or universal. Figure 13-13A shows a tooth



126.48X

Figure 13-13.—(A) Tooth rest blade in a plain holder. (B) Tooth rest blade in a universal holder.

rest blade in a 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 will provide this strength. Figure 13-14 shows a typical cutter tooth and the angles produced by grinding.

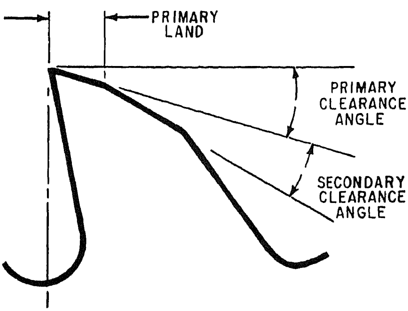
The primary clearance angle is the angle ground when a cutter requires sharpening. The number of degrees in the primary clearance angle varies according to the diameter of the cutter and the material being cut. A large diameter cutter requires less clearance than a small cutter. Cutters used to cut hard materials such as alloy and tool steels require less clearance than cutters used to cut softer materials such as brass and aluminum.

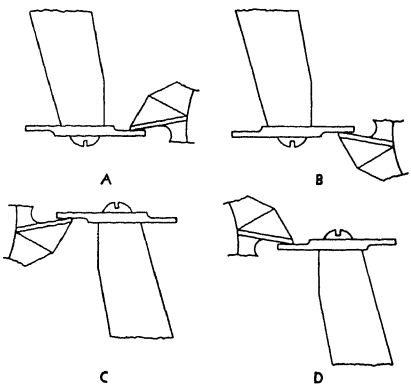
The primary clearance angles range from 4° for a large cutter to 13° for a smaller cutter. Some manufacturers of tool and cutter grinders have charts that can assist you in determining the correct clearance angle. The width of the primary land (the surface created when the primary clearance angle is 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 (heel) of the land 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 clearance angle is normally 3° to 5° greater than the primary clearance angle.

You obtain the desired clearance angle 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 to then raise or lower the wheel head the proper distance to give the desired clearance angle.

When you use the straight wheel, bring the center of the wheel and the center of the work into the same plane by using the centering gauge (fig. 13-15) or by using a height gauge. Then, fasten the tooth rest to the machine table and adjust the tooth rest to the same height as the center of the work. Raise or lower the wheelhead a predetermined amount to give the correct clearance angle. To determine the amount to raise or lower the wheelhead, multiply the clearance angle (in degrees) by the diameter of the *wheel* (inches) and then multiply this product by the constant 0.0087.





126.49X

amount to raise or lower the wheelhead, multiply 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, you can sharpen practically any cutter. In fact, you can sharpen most cutters by 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)

The following is a somewhat detailed explanation of how to sharpen a plain milling cutter with helical teeth. We have provided the detail because

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 the wheel guard.

6. Use a dressing stick to thin the cutting face of the wheel to not more than 1/8 inch. True the wheel, using a diamond truing device.

7. Using the centering gauge, 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 the mandrel 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 gauge on top of the wheelhead and the tip of the gauge directly in front of the cutting face of the wheel, adjust the tooth rest blade to gauge 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

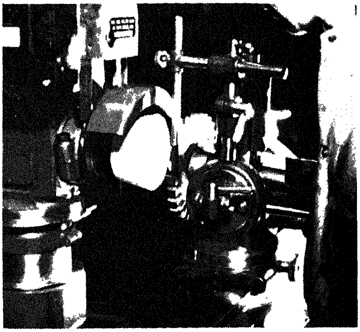


Figure 13-16.—Grinding the side teeth of side-milling cutter.

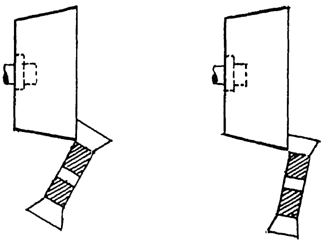


Figure 13-17.—Changing clearance angle by swiveling the cutter in a vertical plane.

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 tooth 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).

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

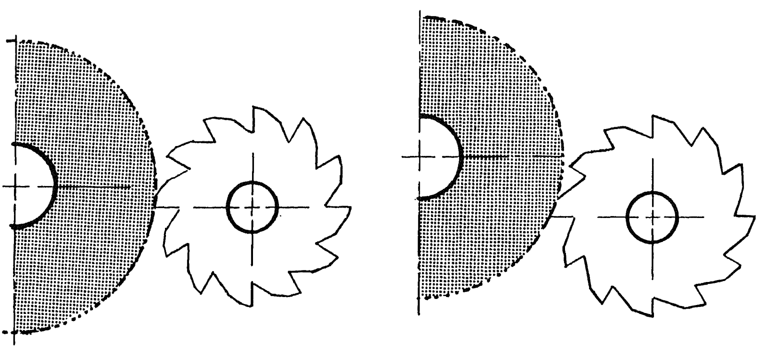


Figure 13-18.—Changing the clearance angle by raising the grinding wheel.

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).

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. Then turn the cutter over and grind all of the teeth on the other side.

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.

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).

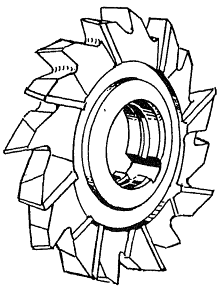
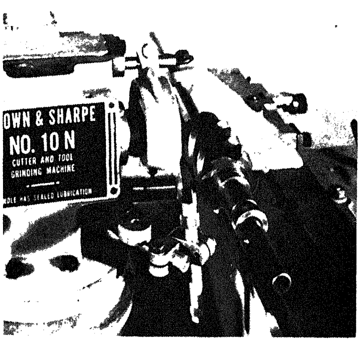


Figure 13-19.—Staggered-tooth side milling cutter.



BROWN & SHARPE Manufacturing Company, North Kingstown, RI

28.434X

Figure 13-20.—Tooth rest mounted on the wheelhead in grinding a helical-tooth cutter.

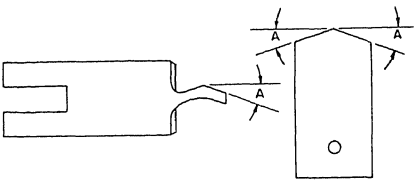


Figure 13-21.—Tooth rest blades for staggered tooth cutters.

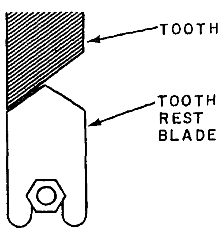


Figure 13-22.—Resting the face of a tooth on its corresponding side of the tooth rest blade.

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 tooth rest. Continue alternating teeth on each pass until you have sharpened all the teeth.

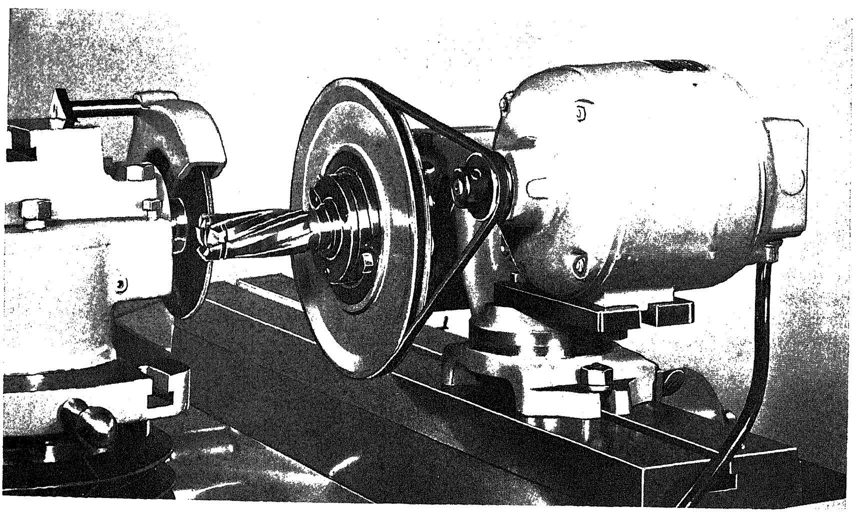
ANGULAR CUTTERS

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 gauge 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

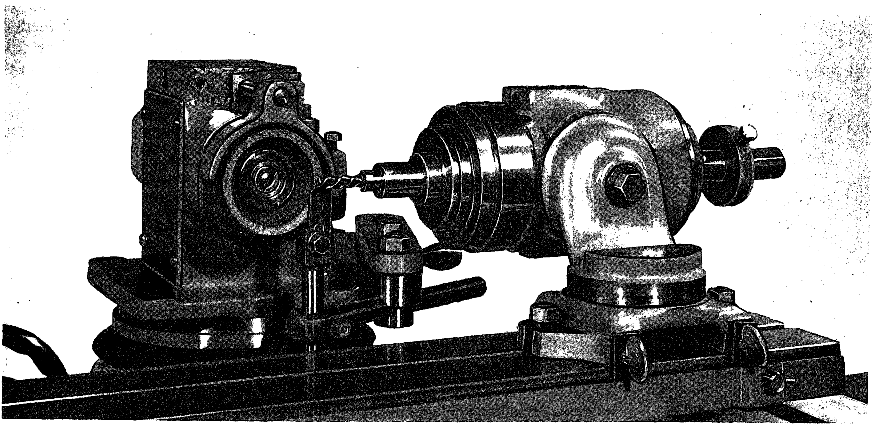
You may salvage a damaged end mill by cutting off the damaged portion with a cylindrical grinding attachment, as shown in figure 13-23. When you salvage an end mill in this manner, use a coolant if possible to avoid removing the temper at the end of the cutter. Be sure to relieve the center of the end in the same way as on 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)



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.

of the cutter. You must, therefore, 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 end mill in a universal workhead.

2. Swivel the wheelhead to 89°.

3. Bring the cutting edge of a tooth into the same horizontal plane as the wheelhead spindle axis by using a centering gauge. Place the gauge on top of the wheelhead and raise or lower the wheelhead sufficiently to place the blade of the gauge on the tooth's 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 you grind the corners of the teeth as when you grind the faces 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.

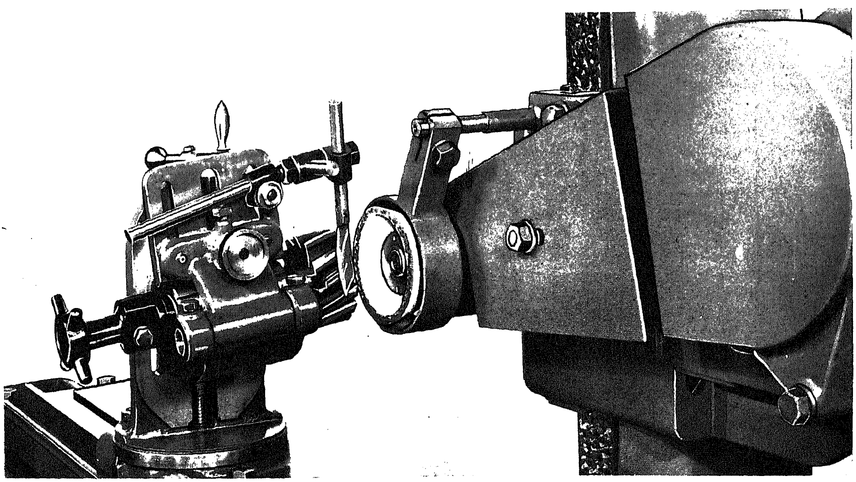
To sharpen the end teeth of a shell end mill (fig. 13-25), mount the cutter on an arbor set in a taper shank mill bushing. Then insert the bushing into the taper shank mill bushing sleeve held in the universal workhead. To obtain the desired clearance angle, swivel the workhead in the vertical plane 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. 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 faces.

The involute cutter (fig. 13-26) will serve as an example. Since the teeth of these cutters have a specific shape, the only correct way to sharpen them is to grind their faces. An important part of grinding the teeth is ensuring that the teeth are uniform, that is, that they all have the same thickness from the back face to the cutting face. You can provide this uniformity by grinding the back faces of all new cutters before you use them. Grind only the back faces, since the cutting faces are already sharp and ready to use. Once the teeth are uniform, they should remain uniform through repeated sharpenings because you will be taking identical cuts on the cutting faces whenever you sharpen the cutter.

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 a stud and line up the cutting face of a tooth with the attachment centering gauge. 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



126.53X

Figure 13-25.—Grinding the end teeth of a shell end mill.

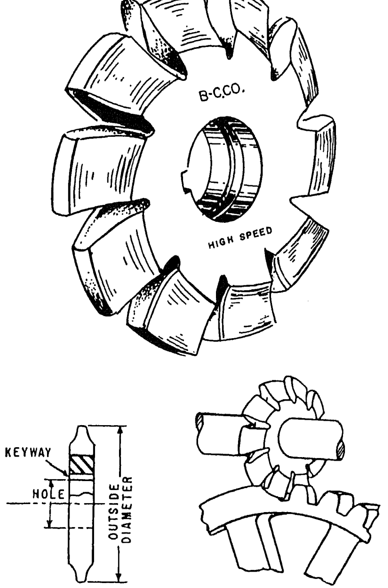


Figure 13-26.—Involute gear cutter.

wheel. Once you have made this adjustment, 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 you have ground all of the teeth 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 gauge, 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, you may sharpen formed cutters by

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, take 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 tooth 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 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 bores, 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. Consequently, it is not often found in too many shops. These machines are usually self-contained hones with a 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, the 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

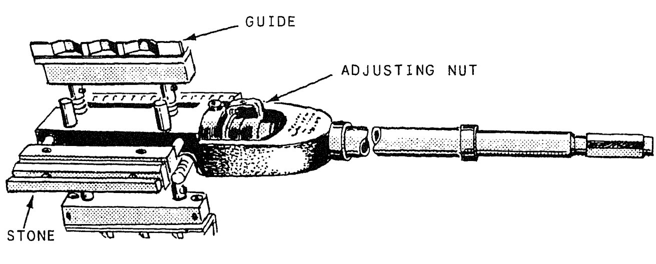


Figure 13-27.—Portable hone.

of the bore.

STONE SELECTION

The honing stone is made somewhat like a grinding wheel, with grit, a bond, and air voids. 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 action 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 with 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, use a coarse grit stone such as a 150-grit to bring the base to within 0.0002 to 0.001 inch of the finish size. Then use a finer grit stone to obtain a smooth finish.

Specific recommendations for stone selection are available from the hone manufacturer.

STONE REMOVAL

Honing does not change the axial location of a hole. The center line of the honing tool aligns itself with the center line 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.

Thus all taper and out-of-roundness are taken out before any stock is removed from the larger selection 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, maintain both the rotating and reciprocating motions so that every part of the bore is covered before any grit repeats its path of travel.

If a bore will require honing to correct taper or out-of-roundness, leave about twice as much stock 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, you should leave from 0.0002 to 0.001 inch 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 the workpiece, finish hone to correct any minor distortion and to produce the desired finish.

Honing produces a crosshatch finish. The depth of cut depends on the abrasive, speed, pressure, and 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.

METAL BUILDUP

Metal buildup is a rapid and effective method of applying practically any metal to a base material. This is used to restore worn mechanical equipment, to salvage mismachined or otherwise defective parts, and to 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 will discuss the thermal spray systems and the contact electroplating process.

Additional information on metalizing is contained in Mil Std 1687(SH) *Thermal Spray Process* and in NAVSHIPS 0919-000-6010, *Instructions for Metalizing Shafts or Similar Objects.*

Additional information on electroplating is contained in MIL-STD-2197(SH), *Brush Electroplating on Marine Machinery* and in NAVSHIPS 0900-LP-038-6010, *Deposition of Metals by Contact (Brush-on Method) Electroplating.*

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 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. Navy shipyards, Intermediate Maintenance Activity (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.)

NOTE: The thermal spray process is NOT authorized in the repair of submarine components (MIL-STD-1687A(SH)).

In this chapter we will 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 the use of thermal spraying in the applications listed below, but the 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.

● 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. Operators and personnel in the immediate vicinity must wear ear muffs and properly fitted soft rubber ear plugs.

● 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 them 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 are performed only by qualified personnel. Potential operators who

spray process used in the qualification testing. For each process, the operator must prepare test specimens for visual, microscopic, bend, and bond tests using qualified procedures developed for that particular coating and thermal spray process. In addition, the operator is responsible for setting up the spraying equipment (gun-to-work distance, air, fuel gas, and so on) as required by the spraying procedure.

A potential operator who fails one or more initial qualification test may be permitted one retest for each type of test that he or she failed.

Certified operators retain their certification as long as they do not let 6 months or more time pass between their uses of the thermal spray process. Operators who let their certification lapse may requalify by satisfactorily completing the qualification tests. Complete information regarding certification is contained in MIL-STD-1687.

TYPES OF THERMAL SPRAY

The two types of thermal spray discussed in this chapter are wire-oxygen-fuel spray and powder-oxygen-fuel spray.

Wire-Oxygen-Fuel Spray

The wire-oxygen-fuel spray process is suitable for all purpose use. It offers variable, controlled

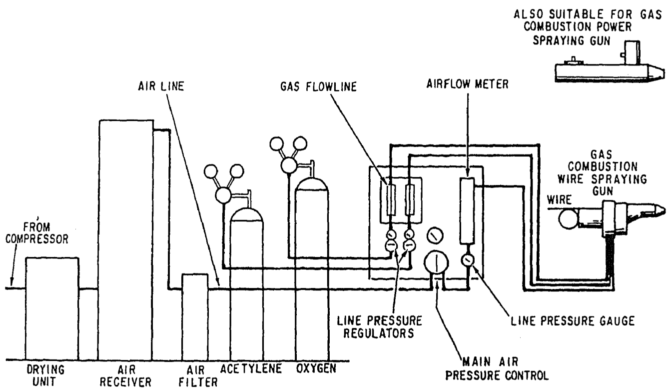


Figure 14-1.—Typical installation for combustion gas spraying.

installation.

The type 12E Flame Spray Gun (fig. 14-2) can spray metalizing wires, such as aluminum, zinc, copper, Monel, nickel, and so forth, in wire sizes ranging from 3/16-inch down to 20 gauge using acetylene, propane, natural gas, manufactured gas, or MPS as the fuel gas. The wire is drawn through the gun and the nozzle by a pair of wire feed drive rollers, 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, such as on steam valves, stanchions, exhaust manifolds, deck machinery, and equipment foundations.

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 a 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 Thermal Spray Gun will spray metal, ceramic, cement and exothermic powders.

Exothermic coating composites are materials that produce an exothermic (heat evolved)

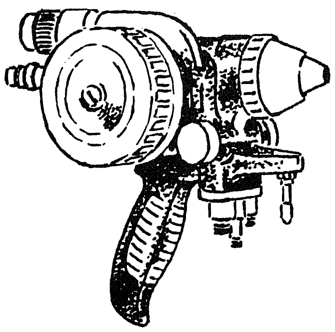


Figure 14-2.—Type 12E spray gun.

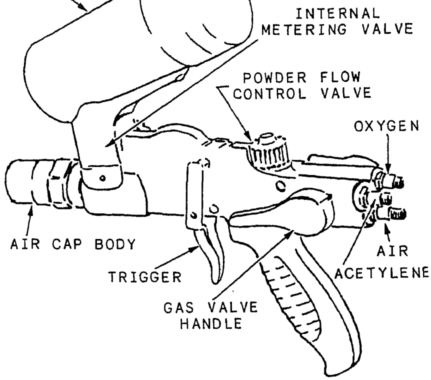


Figure 14-3.—Type 5P gravity feed oxygen-fuel powder spray gun.

reaction from their chemical creation. 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, they react to form nickel aluminide and produce a great deal of heat. Nickel and aluminum, for example, combine to produce nickel aluminite and 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.

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 feed 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 SURFACES

We cannot overemphasize the importance of proper surface preparation. An improperly tion is often the most critical part of the job, it is frequently given the least attention. Quite often, preparation is inadequate simply either because proper preparation is inconvenient or because the necessary equipment is not available. Great emphasis is placed on preparation because even the best and most elaborate surface preparation is still the cheapest part of the job. To help ensure a quality job, be sure to use the required equipment and prepare the surface carefully and thoroughly.

Preparing the surface involves three distinct operations: (1) cleaning, (2) undercutting, and (3) surface roughening.

Cleaning

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 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 that it is not so strong that it attacks the base material; do NOT leave any residue film on the base surfaces. METCO-Solvent Trichloroethane O-T-620 and Toluene TT-548 are suitable solvent cleaners. Because of the flammable and 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 for surface roughening operations the abrasive blasting equipment that you use for general cleaning operations.

HEAT CLEANING.—Clean porous materials that have been contaminated with grease or oil with a solvent 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

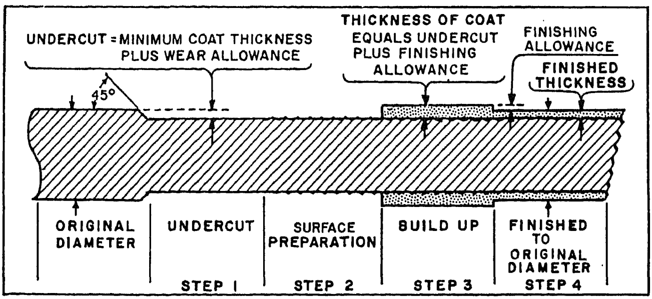


Figure 14-4.—Major steps in restoration of dimensions with thermal spray.

cut should be either 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 shoulders without a fillet or radius may produce stress risers. A stress riser is a spot on a part where stresses have been set up that may cause the part to fail. When you prepare 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 shooting 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 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 will contaminate it. If, for any reason, the surface becomes contaminated, you must thoroughly clean and degrease it. The cleanliness and roughness greatly carefully control cleanliness to ensure adequate bond strength for the service to which the part will be subjected. Two methods of surface roughening are (1) abrasive blasting and (2) macroroughening, for restoring dimensions greater than 1/2 inch where exothermic materials cannot be used.

ABRASIVE BLASTING.—Prior to thermal spraying, condition the surfaces to be coated by abrasive blasting. 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. The required amount of surface roughness is related to the configuration (size and shape) of the part. Where part configuration permits, a roughness of 200-300 microinches is desired. When distortion can occur, such as with thin walled sheet metal parts, reduce the 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. Keep chilled iron grit dry during storage and use. Do not use grit designated for coating preparation 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 basted 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 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, and then to blast further for a short period.

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 medium 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 for use 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 so 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 that the metal mask 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 done 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

thread in the substrate.

APPLYING THE COATING

Applying the coating consists of three distinct procedures: Masking, spraying the coating, and applying a sealant to the coating.

Masking for Spraying

You can use tapes, liquid-masking compounds, silicon 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 the adhesion of sprayed material. Approved masking compound will not run or bleed at the edges.

You may also use masking compound to protect the spray booths and other equipment which is subject to overspray, such as rotating spindles, chucks, lathes, and the like. When you use masking compound for this purpose, be sure to clean the surfaces on a regular schedule and reapply the compound since it will eventually dry out and the sprayed material will then stick to the substrate. For instances when you cannot protect holes, slots, keyways, or other types of recesses by tapes or shields, use inserts of carbon, metal, or rubber. Install these inserts before you begin abrasive blasting and spraying, and leave them in place throughout the thermal spray operation. Remove them after you complete the surface finishing but before you begin applying the final sealer.

Spraying the Coating

Spray the component using the specifications (gun-to-work distance, rotational or linear speed of the gun to the work piece, air, fuel, gas, primary and secondary pressures, and power output) contained in the approved procedure for the material being sprayed.

you expect more than 15 minutes, but not over 2 hours to elapse from the time that you finish preparing the surface until you begin the spraying operation, or if the part must be removed to another location, you must protect the prepared surface from oxidation, contamination, and finger marks. 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. Take temperature readings with a contact pyrometer. Do NOT use temperature sticks or similar devices in the thermal spray area. If you preheat with a gas flame, do not apply the flame 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 a difference in the expansion rates 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 on the area being sprayed.

In general, keep the direction of the metal spray as close as possible to a 90° angle to 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 a right angle 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.

in excess of that required for finished dimensions on the surface to provide for machining or grinding. To help ensure a proper buildup, 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 the work with a spray of water or other liquid.

Applying the Sealant

To prevent corrosive attack or fluid leakage, sprayed coatings must be treated with a sealant. The particular sealant selected will depend on the maximum use temperature of the component and the purpose of sealing the coatings. Apply the sealant after spraying and before finish machining. For severe applications, apply a sealant 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

FINISHING THE SURFACE

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 built up on each other. This structure, which by its relatively low coefficient of friction and high oil-retaining qualities makes sprayed metal ideal for all bearing surfaces, creates a problem in finishing. Experimentation and research indicate that if you understand and appreciate the characteristics of sprayed metals, you can machine and grind them in the toolroom or on the production line with less trouble than you have with 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 with a similar metal in its solid or wrought form. As a result, crumbly chips similar to those from cast iron will occur regardless of porous.

A grinding wheel operator will tend to use the grain and grade of wheel he or she uses 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 are available 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 specifications are more 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 coatings sprayed over a substrate. Grinding and finishing operations should take this 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 are cautioned to grind slowly enough and apply sufficient coolant to avoid local overheating of such materials. Where a thin coating has been applied over a relatively soft substrate, the finishing operations must be done in a way to avoid loads on the coating that could seriously deflect it.

Requirements

Thermal sprayed coatings differ enough from the same materials in wrought form that different grinding wheel and finishing tool recommendations are almost always required. Therefore, the choice of tools and wheels should NOT be based on experience with the parent material in wrought or cast form. Selection of the

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.

Wheels with coarse grain and low bond strength are used to grind sprayed coatings to prevent loading the wheel. Wet grinding is usually recommended over dry grinding if the proper wheel is used. 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.

Always consult and follow the coating manufacturer's finishing recommendations when you select the finishing technique, including the proper tool, feeds and speeds.

Remove masking materials before you begin surface finishing, and finish the part to the dimensions required by the specification or drawing.

Where finishing difficulties do arise even though you have followed proper finishing techniques, review the spraying operation itself. It is quite obvious that if, for instance, particles pluck out, 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. Using the wrong gun-to-work distance and spraying at the wrong angle to the substrate surface are typical faults which may affect the structure of the coating adversely and cause finishing difficulties.

Machining

The sprayed coating stream has an appreciable area (approximately 3/8 to 1/2 inch in diameter). 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

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 chunks, 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.)

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 in finishing than it is in service. The procedures described above minimize the machining stresses.

Machining sprayed metal is not difficult. Carbide tools are necessary for the harder materials. A tungsten carbide tool bit, sharpened for cast iron, will be satisfactory. Since the sprayed coating contains hard oxides, even the softer sprayed metals which can easily be cut with high-speed steel tools, have an abrasive action on the tool tip. High work speed, slow traverse and light infeed are required. When it is necessary to hold a dimension to a tight tolerance, you must take tool bit wear into account. Carbide tools have been found to be more satisfactory than softer tools for machining most sprayed metals.

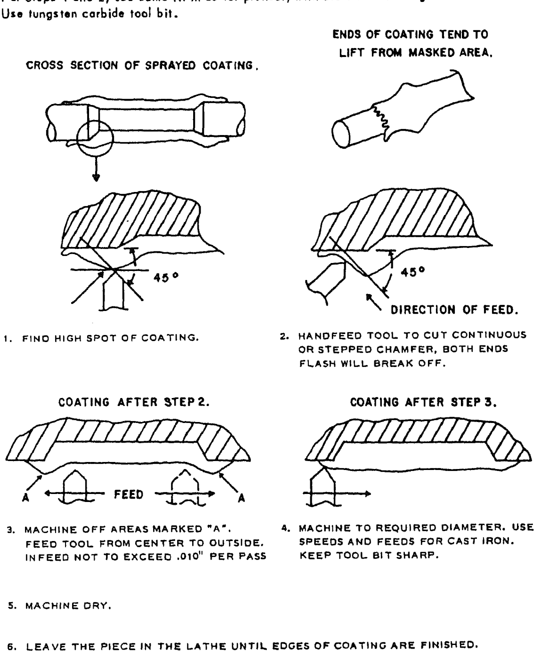


Figure 14-5.—Finishing machining of a thermal spray coating.

Figure 14-6 illustrates proper tool configuration for machining sprayed materials. Do not follow the usual rules that apply to the use of carbide tools for heavy machining work since they do not apply to machining sprayed materials. For instance, when you machine 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°.

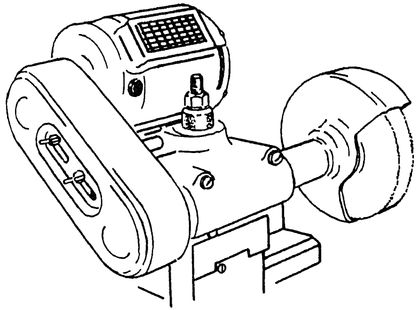


Figure 14-7.—Lathe grinder for dry grinding of thermal spray coating.

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 sealants before grinding prevents contamination of the pores of the sprayed coating and also helps to provide a cleanly ground surface instead of a surface with the particles smeared or drawn into feathers.

Always use heavy grinding equipment with carefully trued concentric wheels. (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, you must pay attention to the special problems resulting from the structure of sprayed materials as discussed earlier. Remember that

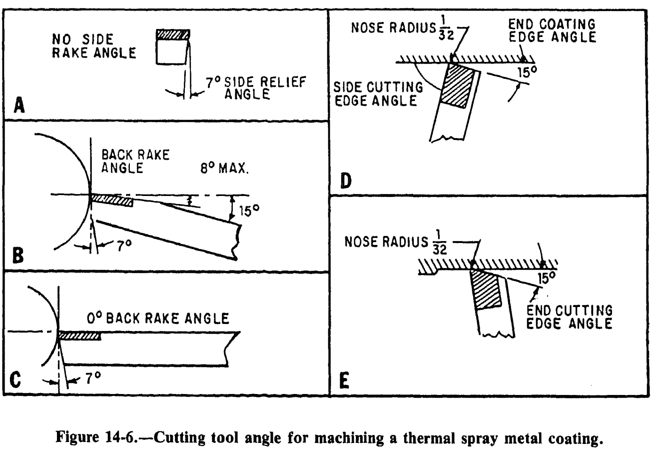


Figure 14-6.—Cutting tool angle for machining a thermal spray metal coating.

structurally as those in solid form and that you need to use the different wheels, feeds, speeds, and so on suggested in the coating manufacturer's recommendations.

The softer sprayed materials, particularly the sprayed metals, tend to "load" a wheel. The use of wheels with relatively coarse grain and low bond strength is necessary for such materials so that the wheel will break down before loading.

Thoroughly clean ground surfaces after you grind them whenever the surface is to be used as a journal surface or a surface that will mate to another machined part. This procedure is emphasized because the porous structure of most sprayed coatings are more inclined to retain mentioned earlier, sealing prior to grinding will ensure clean final surfaces.

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 the anode lead of a d.c. power pack. The cathode lead of the power pack is connected to

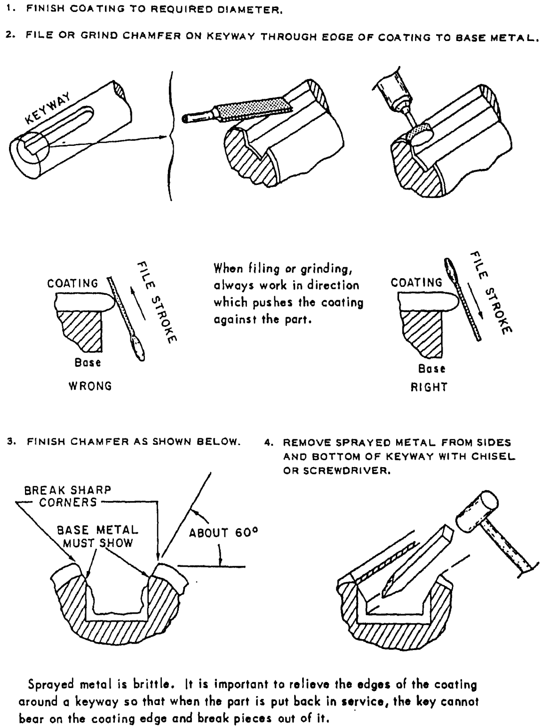
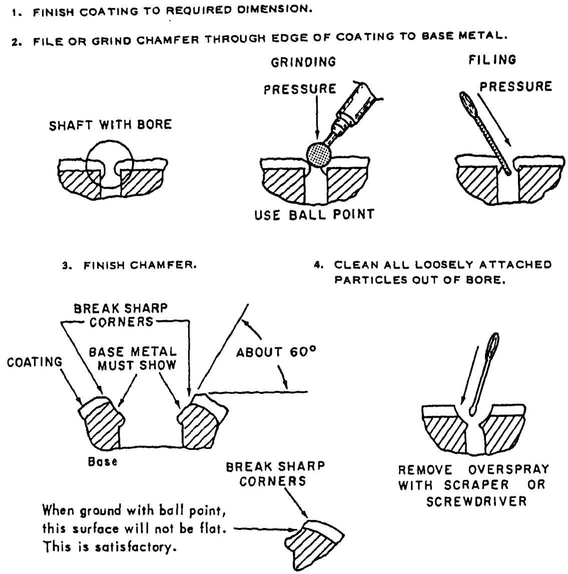


Figure 14-8.—Finishing keyways.



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.

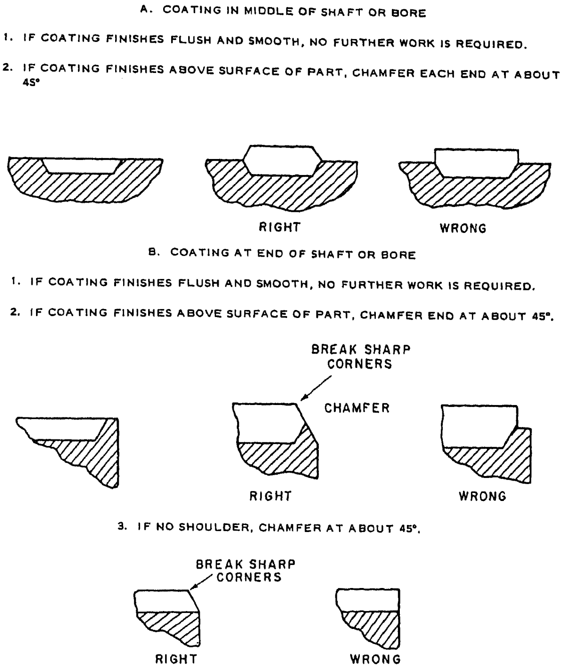
CAUTION: Clean the metallized piece thoroughly before putting it back in service. Any loose particles of sprayed metal might cause trouble.

Figure 14-9.—Finishing holes and other openings.

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, and so forth. This process is not recommended to replace bath plating. However, there are some advantages which make contact electroplating superior to bath plating in some situations:

- 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.



The ends of the coating must be finished off so that there is no load on any edge of the sprayed coating when the part is put back in service.

Figure 14-10.—Finishing the ends of coating.

- 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.

INTRODUCTORY INFORMATION

The following paragraphis provide an overview of the electroplating process before we begin more detailed discussions.

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 units in this range are portable, weighing less than 150 lbs, yet can provide the required power for most shipboard and shop 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.

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 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 certified by procedure tests. *However, plating and preparatory solutions of different manufacturers must not be used for the same plating job.*

For plating operations, solution is either poured into shallow glass or plastic 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 ensure maximum tool to workpiece contact 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 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 training course, or other approved training course. To qualify the operator must show proficiency in the contact plating process which includes the following:
   - a. Preparation of a 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 MIL-STD-2197(SH).

Completion of an approved training course and certification will not always assure that the operator is skilled enough to do all jobs that he or she may encounter. 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 house capability, vendor services should be used to assist with the actual plating and to provide 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 a face shield, rubber gloves and a rubber apron or laboratory clothing at all times when electroplating.

- NEVER let your skin come in contact with the solutions. If you do contact a solution, wash your skin thoroughly with soap and water.

- When 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.

- Ensure that warning signs are 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 operation procedures.

- Wear resperators of the proper type during all plating operations.

Terminology

Contact electroplating is highly technical and introduces many terms of which you probably have little knowledge. The next few pages contain definitions which you will need as you study the process of contact electroplating. Read them carefully and then refer to them as you progress through the remainder of the chapter.

ACTIVATE: Removing passive film which is normally present or which forms quickly on 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 changes 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. 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 the 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 corrodes, rather than 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 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 additional layers will not adhere well to the burned layer 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 for chemical or mechanical removal of the burned layer. Plating then can be resumed after the surface is properly prepared.

CARBURIZED: Case hardened by impregnating carbon in the surface of a part and then heat treating 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.)

CATHODIC CORROSION PROTECTION: Corrosion protection offered by a deposit more reactive than the base material. The deposit must be pore-free, to prevent the base material from corroding 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, for decorative reasons, or as a base for paints.

COHERENT: Holds firmly together as one piece; has high resistance to breaking apart in pieces.

CONSTANT FACTOR: The factor (see *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 Rc 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 it in a special solution.

ETCH: To electrochemically remove material from a surface. Conducted with an 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: The formation of oxides in an area undergoing fretting. The oxides cause additional wear to the mating surfaces.

GALLING: The damaging of one or both metallic surfaces by the removal of particles during sliding friction.

GASSING: Development of hydrogen gas bubbles on the workpiece, either by activating or plating, or by chemical attack of the activator on chromium.

GRAIN STRUCTURE: The physical arrangement (appearance) of the grains of a metal. Grain size varies from invisible to the naked eye to perhaps 1/8 inch in diameter.

HARDCOAT: An oxide coating formed on aluminum by making the aluminum 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 Rc are common hardness tests.

HYDROGEN EMBRITTLEMENT: A condition in which a material is easier to break than usual because of its absorption of hydrogen. Occurs only with certain materials such as steel over 40 Rc, 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

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 testing thin or small areas such as an electrodeposit for hardness. Knoop hardness values are converted to more familiar Brinell or Rc hardness values by using conversion charts.

LITER: A 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 or greater.

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 nitrogen containing material. Nitrogen defuses 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.

PASSIVATE: The formation of a thin, invisible oxide film on certain metals which impairs adhesion of an electroplate.

pH: A measurement value on a scale of 0 to 14 of the acidity or alkalinity of a solution. 0 indicates 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 a plating solution other than the desired solution. Preplates are used to improve adhesion.

PREWET: Applying plating solution to the surface before applying current. The operation improves the adhesion of deposits from certain solutions by ensuring that plating begins on a surface covered all over with full strength solution.

Rc: Rockwell C hardness.

REACTIVE METALS: Metals that 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: Cathodic corrosion protection.

SCALE: Surface oxidation on a metal caused by heating in air or in an oxidizing atmosphere.

SEIZING: When two surfaces have fused together due to friction.

SMEARED METAL: Deformed metal 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: A factor that is not constant but which 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.

WATER BREAKS: The breaking of a water film into beads. Beading indicates contaminates on the surface.

Applications

The contact plating process is a rapidly expanding field. When used for depositing a corrosion resistant coating, electroplating has shown sufficient success to permit almost machinery is limited only by the knowledge and skills of the operator in areas where plating is allowed. Requirements for contact plating are specified in Table 14-1 which defines the area of permissible use of contact plating. For simplification, applications are classified as follows:

- Class I: Plating used for decorative or corrosion prevention functions only.
- Class II: Plating on parts that remain in static contact with other plated or unplated parts.
- Class III: Plating on parts that make rubbing contact with other plated or unplated parts, excluding those in Class IV.
- Class IV: Plating on rubbing contact parts in elements of turbine/reduction gearing, turbo or diesel electric power generating units, and main propulsion shafting.
- Class V: Plating on parts under the cognizance of the Nuclear Power Division.

- 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 of steam cuts and erosion damage.

Diesel engine cylinder blocks: Restoration of mating surfaces damaged by fretting.

Wave guide plumbing: Plating of 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, saltwater and freshwater service.

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.

- Close Tolerance Mating Parts

Pump impellers: Repair of worn bores and keyways to restore design size and fit on a 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 of a turbine nozzle control valve seat's hard facing by plating 0.003-0.005 inch thickness of cobalt over copper and nickel substrates. The thickness is 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, due to poor bonding. For this reason, you should not contact plate chromium on an existing bath chromium deposit on 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.

- Brush electroplating of lead and lead alloys is restricted. Use it only to repair plating on battery terminals and busing components where its use has been previously authorized.

- Deposition of chromium: Contact plating solutions can produce deposits with mechanical properties which will satisfy the requirements for most plating work. Therefore, brush on plating coatings can normally be used for repairs 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 chromium deposited by bath electroplating, the thickness of the buildup is limited, and the process is tedious and slow. As an alternate, you can use other metals such as cobalt or nickel. These will provide wear resistance and hardness properties which are suitable for most applications where chromium would normally be used. For areas that require 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 instructions covering the use of their products. You should follow these instructions closely 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.

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 5708 Schaaf Road Independence, Ohio 44131 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.

Quality Control

Quality control is composed of several factors: documentation, process control, general (all plating) inspection, and liquid penetrant inspection of plating for rubbing contact service.

DOCUMENTATION.— The quality control department ensures that each plating job 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

PROCESS CONTROL.—All parts to be plated should be handled according to written Process Control Procedures approved by the individual activity. Plating work should be set up to ensure a smooth flow of 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 the ship, the date, and the job order number when applicable.
2. Description of the part to be plated by proper name and piece number on the blueprint.
3. A sketch of the area requiring plating.
4. Identification of the base metal.
5. Final required thickness of the deposit.
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 INSPECTION PROCEDURE (ALL PLATING).—Prior to declaring the plating job complete, ensure that the finish satisfies the following inspection requirements:

- 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: Perform an adhesion test 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. Continue taping so that approximately 1 1/2 inches of the base metal on each side of the plated area is also taped. Tamp the tape down to ensure that it sticks thoroughly.
4. Grip the loose end of the tape and rip rapidly upward (at a right angle to the plating), removing the tape with a single jerk.
5. Inspect the tape. If any plating is stuck to the tape, reject the plating job.

Platings for Rubbing Contact Service

In addition to the general inspection, plating for rubbing contact service must meet the liquid penetrant inspection.

- Liquid Penetrant Inspection: Use Group I liquid penetrant in according to the requirements of 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, you may use water washable penetrant material (Group III or IV of MIL-STD-271) for liquid penetrant inspection.

POWER PACK COMPONENTS

The equipment must contain the safety features required by MIL-STD 454. Operations that could create personnel hazards or result in damage to the equipment or work must 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 labelled in figure 14-11, using a DALIC machine as an example.

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 you a second-to-second 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. 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 × time) of current passed through the d.c.

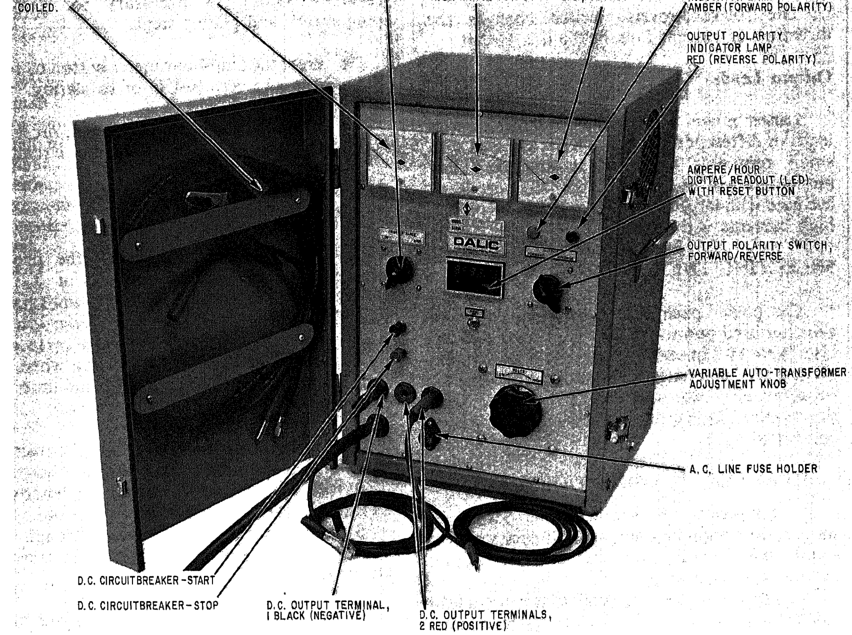


Figure 14-11.—DALIC power pack.

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, and so on 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 used with small terminals for small tools where low amperages will be drawn. Larger size wire leads are used with large terminals for large tools where high currents will be drawn.

SELECTING THE 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.

OPERATING THE POWER PACK

Prior to Plating

Perform 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. Turn the "volts" control to the extreme "low" position.
3. 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 the Plating Operation

- Press the "start" button to energize the 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 the plating is completed, press the "stop" button to deenergize the d.c. circuit.

SELECTING AND PREPARING PLATING TOOLS

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 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 you plate higher thicknesses on larger areas.

The preparatory steps (cleaning, deoxidizing, etching, etc.) are relatively short steps, compared to those of 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 should, if possible, cover the full length of the area to be plated to assure uniform preparation.

You can get sufficient solution on the tool by dipping for solution. In most cases, a standard plating tool will meet the above requirements and you will not need to make special preparatory tools.

Proper Plating Tools

The plating step generally represents the major part of a complete plating operation. Therefore, the selection of the proper plating tool is more critical than the selection of 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

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

28.X

also important to have the proper tool when uniformity of deposit thickness is necessary.

Optimum Contact Area for the Plating Tool

A tool that gives the optimum contact area on the area to be plated lets you plate a good deposit as fast as possible. The optimum contact area depends on the power pack to be used, the solution to be used, and the size and 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.

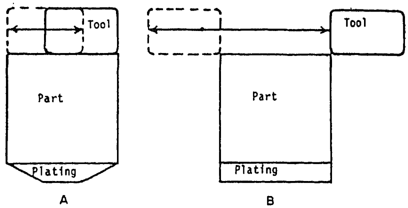
If, for example, Code 2080 solution is to be used with a 60-amp power pack, the maximum contact area required is 10 square inches.

Seven formulas that are useful with the Dalic plating process are discussed at the end of this chapter, beginning on page 14-59. You can use formula 3 to determine the optimum contact area mathematically. The optimum contact area is required on very large areas. On very small areas the contact area is the maximum contact area that you can obtain; 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, the optimum contact area for a flat surface is full contact up to an area the size given in table 14-7. 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, the 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 the 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 will 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 of the part. This motion, also, may not be practical if there is a shoulder at one side. The same situation



28.450X

Figure 14-12.—Plating—covering the full length.

applies to I.D.'s and flat surfaces. Summarizing, always try to have the tool cover the full length of the O.D. or I.D. or the full length or width of a flat surface.

Solution-Feed 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 the quality and reliability of the 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.

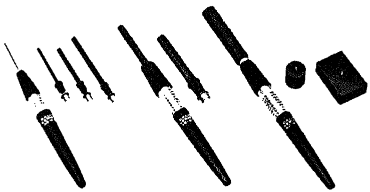
Use the following procedure to determine if it is worthwhile to use a solution-fed tool.

1. Use Formula 1 (page 14-59) to determine amp-hours required for one part and then multiply by the number of parts.
2. Determine the type of tool to be used and also its contact area. Then use formula 4 (page 14-60) to determine the total plating time if the solution is pumped through the tool.

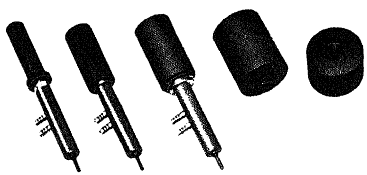
Since dipping for solution usually doubles plating time, the value you determine in step 2 above also represents the extra time you will spend dipping for solution. This possible savings in time can help you determine if it is worthwhile to set up to pump the solution.

Standard Tools

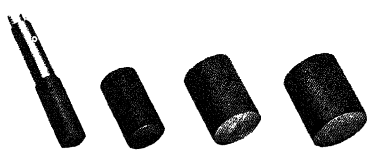
Standard tools (figures 14-13 and 14-14) are available for preparing and plating a wide variety



AC - SERIES



WC - SERIES



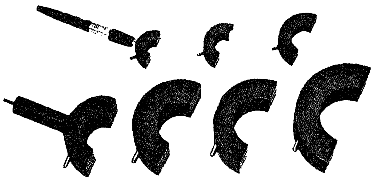
TOOL Cat. No.	COMPONENTS Handle	Anode	Adapter
AIR-COOLED (for small areas and I.D.'s)			
Solution Dip			
AC - 0	AC 1-3	AC-0* .09 ø x 2"	. . .
AC - 1	AC 1-3	AC-1 .180 ø x 2.25"	. . .
AC - 2	AC 1-3	AC-2 .25" ø x 2.25"	.
AC - 3	AC 1-3	AC-3 .31" ø x 2.25"	. . .
AC - 4	AC 4-7	AC-4 .375" ø x 3"	. . .
AC - 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 2.1[illegible]"	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

28.45

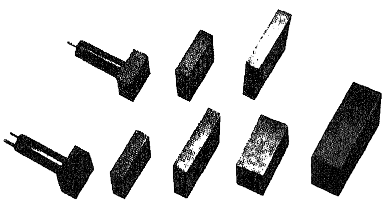
28.451X

Figure 14-13.—Standard plating tools.



SCC - SERIES / SCG - SERIES

TOOL Cat. No.	Handle	COMPONENTS Anode	Adapte
SOLUTION FED (for O.D.'s)			
SCC-10	AC 4-7	SCC-10 1" I.D. x 1" wide	A
SCC-15	AC 4-7	SCC-15 1.5" I.D. x 1" wide	A
SCC-20	AC 4-7	SCC-20 2" I.D. x 1" wide	A
SCC-25	AC 4-7	SCC-25 2.5" I.D. x 1" wide	A
SCG-25	G	SCG-25 2.5" I.D. x 2" wide	F
SCG-30	G	SCG-30 3" I.D. x 2" wide	F
SCG-35	G	SCG-35 3.5" I.D. x 2" wide	F
SCG-40	G	SCG-40 4" I.D. x 2" wide	F



FG - SERIES / FF - SERIES

FLAT & MULTI-PURPOSE TOOLS
Solution Dip

FG - 1	G	FG-1 2.5" x 2.5" x 1"	FC
FG - 2	G	FG 2 3.5" x 3.5" x 1"	FC
FG - 3	G	FG-3 4.5" x 4.5" x 1"	FC

SOLUTION FED

FF - 1	F	FF-1 2.5" x 2.5" x 1"	FC
FF - 2	F	FF 2 3.5" x 3.5" x 1"	FC
FF - 3	F	FF-3 4.5" x 4.5" x 1"	FC
FF - 4	F	FF-4 4" x 3" x 2"	FC
FF - 5	F	FF-5 6" x 4" x 2"	FC

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.

28.452

Figure 14-14.—Standard plating tools.

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 the area to be plated.
2. Cover the full length.

Plating Tools:

1. Provide the optimum contact area.
2. Cover the full length.
3. Allow for pumping the 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 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 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 for 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 14-60).

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 you 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 you 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, and so on based on design plating amperage. When dimensions of anodes are based on the optimum contact area, the plating amperage should be the amperage rating of the power pack. When the anode dimensions are based on maximum practical contact area, compute the expected plating amperage using formula 4 (page 14-60).

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 of 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 spaced 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 inch in diameter.

● Make the main distribution hole in the anode next to the inlet fitting at least 1/4 inch in diameter when you use a small submersible pump and 1/2 inch in diameter when you use a large submersible pump. This helps ensure that all outlet holes are reasonably well fed.

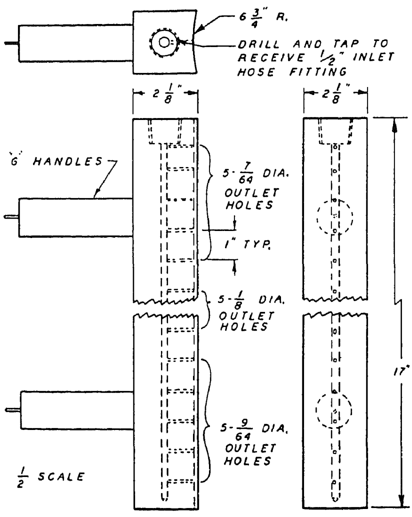
The following examples will help you understand how to make the special tools you may need in contact electroplating.

EXAMPLE #1

Plate a 16-inch length of a 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.

The 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

$$CS = \frac{34}{16} = 2\ 1/8 \text{ inches}$$



28.453X

Figure 14-15.—Design of a special tool.

Allowing for a cover thickness of 1/4 inch, put a 6 3/4-inch radius in the 16 inch × 2 1/8-inch face. To help keep the rather long tool squarely on the part, use two G-handles. Make the solution outlet holes slightly larger as their distance from the solution inlet port increases. (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 Rf-30 will give a small contact area, draw only approximately 30 amps, and result in a plating time of 2 hours. A better tool would be a pie wedge-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.

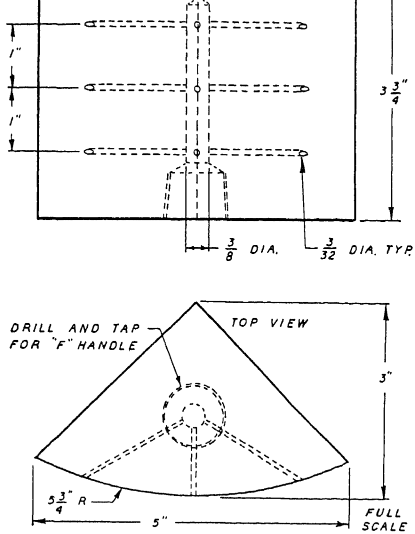
In view of the difficulty in moving the tool, make the tool 3 3/4 inches long to ensure full contact along the length. The bore being 3 inches long, the contact length remains 3 inches. (See fig. 14-16.) The optimum contact area is 15 sq in. The contact width then is

$$CS = \frac{15}{3} = 5 \text{ inches}$$

EXAMPLE #3:

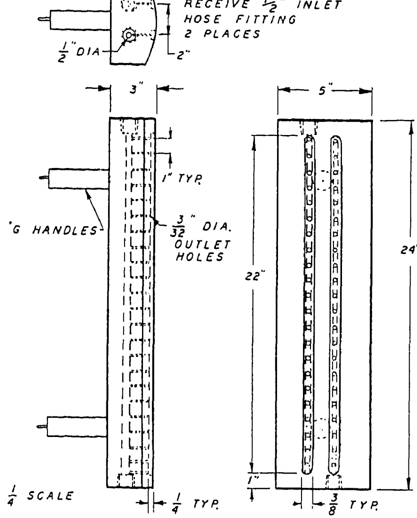
Ten bearings must be plated on a 20 inch long, 26 inch I.D. with .002 inch of babbitt, Code 4009, per side. The part will be rotated in a barrel rotator, leaving the I.D. accessible from both ends. Use a 100-amp power pack.

The optimum contact area is 100 sq in. Since the contact length is 20 inches, the contact width is 100/20 or 5 inches. 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.)



28.454X

Figure 14-16.—Design of a special tool.



28.455X

Figure 14-17.—Design of a special tool.

Plating Tool Anode Materials

A grade of graphite with maximum resistance to breakage and anodic corrosion is 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.

Use sandpaper or other similar abrasive materials 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 you may not always be able to clean a tool thoroughly, your best action is to identify it for and use it 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, and so forth to occur.

2. It mechanically scrubs the surface being plated which permits sound deposits to be applied rapidly.

3. It holds and uniformly distributes the solution where it is needed.

Several covering materials are used with the plating process. 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 the solution uniformly and has satisfactory wear resistance.

SPECIAL COVERS: Used for special effects such as described below.

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 a final cover for wear resistance.
Dacron Batting	Initial	Used very little because of its high cost compared 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 a moderate degree as a final cover. Very low cost and high purity and absorbency. Has less wear resistance than Dacron tubegauze. Used as a final cover for preparatory tools and for rhodium plating.
Dacron Tubegauze	Final	Widely used as a final cover especially for plating tools where its superior wear resistance compared 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 the workpiece under the tool.
Dacron Felt	Combination	Used frequently for plating tools 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, such as 0.005 to 0.015 inch, is required in a certain deposit. Keeps the deposite 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 rather than the material 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.

Gray Scotch-Brite (Continued)	Special Purpose (Continued)	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, and so forth) 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 the plating equipment 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 the outside surface of the anode. The felt is conductive enough to carry plating current, but not conductive enough to damage the part of 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, lower possible plating times. Cost is high and absorbency and purity are excellent.
Orion	Special Purpose Final Cover	Used for low thickness deposits (9.001 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 deposite and adhesion.
Pellon	Special Purpose Combination Cover	Very thin wear resistant cover useful for plating small I.D.'s, grooves, and so forth where conventional covers cannot be used. Absorbency is poor.

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.

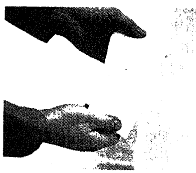
PREPARATION OF ANODES FOR THE ELECTROPLATING PROCESS

The following paragraphs contain step-by-step procedures for you to use in preparing various types of anodes for use in plating.

SCC AND SCG SERIES ANODES

To prepare SCC and SCG anodes for plating outside diameters, take the following steps:

PREPARE THE 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). The thickness of the cotton used may vary, according to the application. Experience has shown that a 3/16″ thickness works well for the average application.


1

28.457X

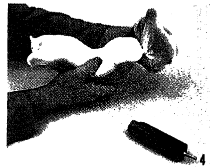
MOLD THE COTTON TO THE ANODE—Mold the cotton to the concave side of the anode (picture 2).


2

28.457X

FASTEN THE TUBEGAUZE—Cut a suitable size of tubegauze (at least twice the length of the anode) and slip half of the tubegauze over the anode and its cover (picture 3). Twist the remaining half of the tubegauze (picture 4) and slip it back over the anode. You then have two layers of tubegauze cover, you should secure the ends 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).

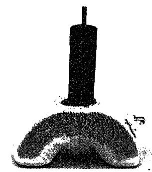

3


4


5


6


7

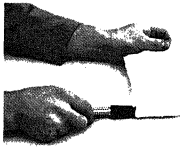

8

28.457X

AC, WC, AND RF SERIES ANODES-GENERAL PURPOSE

Take the following steps to prepare AC, WC, and RF series anodes for plating inside diameters and flat surfaces.

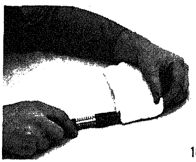
PREPARE THE COTTON BATTING—Cut a piece of long-fiber cotton batting about one inch wider than the length of the Dalic anode and six to eight times longer than the diameter. Split the cotton to about a 3/32″ thickness so that the final cover thickness after rolling will be 3/16″. Lay the cotton on a table and wet the anode with water



9

28.548X

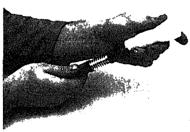
FOLD THE ENDS EVENLY—Fold the protruding end of the cotton evenly over the tip of the anode (picture 10).



10

28.548X

WRAP THE COTTON 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).



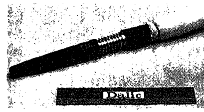
11

28.548X

SECURE THE COTTON WRAP WITH TUBEGAUZE—The application of tubegauze provides maximum wear resistance and prevents cutting through on sharp edges. Apply the

no bulges or thin spots (picture 13).

12



13

28.548X

FG AND FF SERIES ANODES-GENERAL PURPOSE

Take the following steps to prepare FG and FF series anodes for plating flat and other surfaces.

FOLD THE COTTON AROUND THE ANODE—Cut the long-fiber cotton pad for the FG and FF anodes to provide a 1/2″ 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).

14

15

28.548X

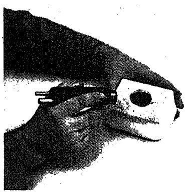
INSERT THE ANODE INTO THE TUBEGAUZE—Holding the wrapped anode by the bottom to keep the cotton smooth, insert it into a piece of tubegauze of appropriate size (picture 16). Secure the ends of the tubegauze tightly by twisting them and binding them with rubber bands or tubegauze ties (picture 17).

16

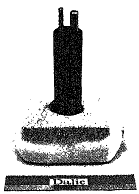
17

28.548X

CUT A HOLE FOR THE 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).



18



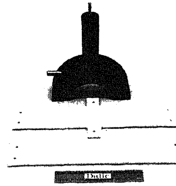
19

28.548X

SCC AND SCG ANODES-SPECIAL PURPOSE

Take the following steps to prepare SCC and SCG anodes with Scotchbrite, Dacron felt, and similar materials.

PREPARE THE 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).

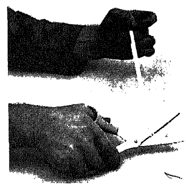


20

28.549X

MAKE TIES—Cut the tubegauze ties (#56 Dacron is best) as shown (Pictures 21, 22).

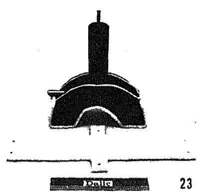
21



22

28.549X

TIE THE COVER TO THE TOOL—Secure the cover to the 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.

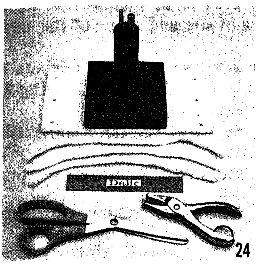


23

28.549X

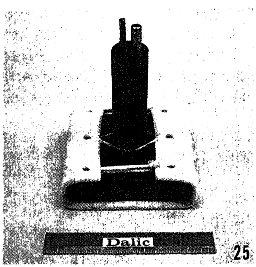
and other special anodes with Scotchbrite, Dacron felt, and similar materials for plating flat and other surfaces.

PREPARE THE SCOTCHBRITE AND THE 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 of the tool. Punch the necessary holes in the Scotchbrite and make the tubegauze ties (Picture 24).



28.460X

TIE THE COVER TO THE TOOL—Secure the cover to the tool with the ties (Picture 25).



28.460X

unlimited. Each manufacturer can provide you 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. You may restore these solutions to full effectiveness by heating them to approximately 140°F and stirring them until all salted out material is redissolved.

Return used plating solution 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.

As a solution is used and collected for reuse it tends to become diluted by water used to rinse the parts that are plated. A minor dilution will not cause a plating problem. However, when dilution reaches 25% the solution should be discarded.

MASKING

Masking serves several purposes in the electroplating process: It prevents plating from being applied on areas where it is not wanted. It provides a definite area to be plated, which permits more accurate thickness control. It reduces waste of metal from the plating solution. It reduces the possibility of contaminating the solution.

Masking tapes are generally used to mask off areas immediately adjacent to the area being plated. The materials tapes may be made of vinyl, polyester, aluminum, and copper tape. Do not use absorbent tapes, such as painter's masking tape, since they can cause small amounts of one solution to contaminate another solution.

Although you should mask all parts carefully, you must mask more carefully when you plan to use a corrosive solution on a reactive base material or when your plating process will develop considerable heat.

Careful masking includes:

1. Careful cleaning of the surface before you apply the tape.
2. Pressing down the 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 for which vinyl tape cannot be used.

Use aluminum tape on demanding masking jobs such as when you use corrosive solutions, when you plate with solutions that develop heat, and when you mask 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. You may then apply vinyl tape over the aluminum tape 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, leaving a 1/8 to 1/4-inch band of aluminum tape 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 vinyl 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.

You may occasionally have to mask off large areas to prevent corrosive solutions from attacking the part and to prevent solution contamination. In these cases, apply tape to the immediately adjacent areas; mask areas farther away 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 of paint.

SETTING UP THE 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, and so on are available. The material is arranged in a step by step manner developed from past practical experience.

We assume that a basic installation is available including a power pack. You can, however, use steps 1 through 7 to select an appropriate installation including a power pack or to assure that an appropriate installation has been purchased.

Step #1 Obtain the necessary information on the job including:

a. The number of parts to be done.

b. The 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 or carburizing, the plating will be applied on the surface material and not on what is underneath.

c. The area to be plated; that is, have a concrete idea of size and shape of the area to be plated.

d. The purpose and requirements of the deposit; that is, why the coating is being applied and what it is expected to do.

e. A general idea of what is adjacent to the area to be plated.

f. The required thickness of the deposit.

Step #2 Selecting the plating solution to use.

This is, in most cases, an extremely important step. Proper selection assures that you will get the desired results 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 for touching 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 you in both instances. Review these tables carefully before you make a selection.

Step #3 Calculate the amp-hours using formula 1, page 14-59.

Step #4 Decide on the general approach to the plating job:

a. Whether you will rotate the part or move the tool by hand.

b. Whether you will pump or dip the solution.

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

28.X

Plating Solution	Code	Content G/L	Factor	Nor. Max.	Avg.	Nor. Max.	Avg.	Yield %	Cu. In. Lit.	Gal.	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	80	.006	1	1	.017	.017	33	4.5	1.19	245
Babbitt-Soft	4010	80	.006	1	1	.017	.017	33	4.5	1.19	245
Babbitt-Navy #2	4011	80	.006	1	1	.017	.017	33	4.5	1.19	231

28.X

Table 14-6.—Properties of Deposits

Deposit	Code	Hardness Knoop	Hardness DPH	Hardness BHN	Hardness 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

28.X

Table 14-7.—Commonly Used Data

Plating Solution	Code	Factor	Plating Voltages: Minimum Small Tools and Areas	Plating Voltages: Maximum Large Tools and Areas	Procedures on New Jobs*	Current Density A/In^2: Nor. Max.	Current Density A/In^2: Avg.	Solution Required per Cu. In.: Lit.	Solution Required per Cu. In.: Gal.	Maximum Recommended Use Amp Hrs. Per: Lit.	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	2.8	.75	24.7	93.4
Chromium	2030	.137	8	25	8+1−1	12	6	56.0	14.8	24.4	92.4
Chromium	2031	.120	4	10	See 6.3.7	4	2	15.7	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	134.0
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	.007	6	25	6+1−1	4	2	3.0	.79	23.4	88.6

*Refer to SIFCO Selective Plating - DALIC Process Manual.

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	Procedures on New Jobs*	Current Density A/In^2: Nor. Max.	Current Density A/In^2: Avg.	Solution Required per Cu. In.: Lit.	Solution Required per Cu. In.: Gal.	Maximu Recommen Use Amp Hrs.: Lit.	Maximu Recommen Use Amp Hrs.: G
Zinc	2100	.011	8	20	8+1−2	6	3	2.3	.62	47.1	[illegible]
Zinc	2101	.011	9	20	9+1−2	14	7	3.1	.82	35.3	[illegible]
Zinc	2102	.011	9	20	9+1−2	14	7	2.9	.77	37.6	[illegible]
Zinc	2103	.011	7	17	7+1−2	14	7	2.9	.77	37.6	[illegible]
Gallium	3011	.015	4	12	-------	3	1.5	6.5	1.71	23.2	[illegible]
Gold	3020	.006	4	20	See 6.3.27	3	1.5	6.3	1.67	9.5	[illegible]
Gold	3021	.006	4	20	See 6.3.27	3	1.5	6.5	1.71	9.3	[illegible]
Gold	3022	.006	4	15	See 6.3.28	3	1.5	7.0	1.86	8.5	[illegible]
Gold	3023	.007	4	10	See 6.3.29	.5	.25	25.3	6.7	2.8	[illegible]
Indium	3030	.009	8	25	See 6.3.30	4	2	4.0	1.05	22.5	[illegible]
Palladium	3040	.017	4	14	4+1−1	6	3	13.1	3.47	12.9	[illegible]
Platinum	3052	.150	3	10	See 6.3.32	12	6	35.2	9.33	42.7	[illegible]
Rhenium	3060	.750				6	3	52.2	13.84	152.5	[illegible]
Rhodium	3072	.030	4	15	4+1−1	6	3	6.8	1.80	44.1	[illegible]
Rhodium	3074	.030	3	12	3+1−1	4	2	20.4	5.4	14.7	[illegible]
Silver	3080	.005	8	20	8+1−1	8	4	2.7	.72	18.4	[illegible]
Silver	3081	.005	4	10	4+.5−1	2	1	3.4	.91	14.5	[illegible]
Silver	3082	.005	5	16	5+1−1	5	2.5	3.4	.91	14.5	[illegible]
Silver	3083	.005	5	16	5+1−1	5	2.5	3.4	.91	14.5	[illegible]
Nickel-Cobalt	4002	.030	8	25	See 6.3.39	12	6	10.4	2.75	28.9	[illegible]
Tin-Indium	4003	.008	8	25	-------	4	2	3.3	.86	24.5	[illegible]
Tin-Lead-Nickel	4005	.006	5	15	5+1−1	3	1.5	3.8	1.01	15.6	[illegible]
Cobalt-Tungsten	4007	.020	10	25	See 6.3.41	12	6	14.5	3.83	13.7	[illegible]
Nickel-Tungsten	4008	.025	8	25	See 6.3.42	12	6	11.9	3.13	21.1	[illegible]
Babbitt-SAE 11	4009	.006	3	15	See 6.3.43	1	1	4.5	1.19	13.2	[illegible]
Babbitt-Soft	4010	.006	3	15	See 6.3.43	1	1	4.5	1.19	13.2	[illegible]
Babbitt-Navy #2	4011	.006	3	15	See 6.3.43	1	1	4.5	1.19	13.2	[illegible]

*Refer to SIFCO Selective Plating - DALIC Process Manual.

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	116	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.

28.X

Step #5 Decide on what type of plating tool you will use, whether a standard tool or a special tool. If you plan to use a special tool, determine its design. (See figures 14-15, 14-16, and 14-17.)

Step #6 Based on the plating tool you will use, determine the contact area if you did not determined it in Step #5.

Step #7 Based on the contact area, determine the plating current if you did not determine it in Step #5. Use formula 4, page 14-60.

Step #8 Determine the plating time using formula 5, page 14-60. If you plan to dip for the solution, double the plating time.

Step #9 Determine the amount of plating solution necessary, using formula 6, page 14-60. Multiply by a factor given

4. (See figure 14-18.)

Step #10 Determine the preparatory and preplate solutions required using table 14-9. Determine the type of tools to be used with these solutions using figures 14-13 and 14-14.

Step #11 Determine the covers to use on all preparatory and plating tools.

Step #12 Determine the masking required.

Two examples of the planning procedure used on actual jobs follow below. This information is you begin any plating operation.

EXAMPLE #1

Step #1 Information on the job.

a. No. of parts—1

b. Base Material—Steel

c. Area to be plated—1″ long × 3.500 + 0.000 bore in a turbine wheel.

d. Purpose of the deposit—To repair a worn I.D. Color match is important. Good hardness, adhesion and cohesion are required.

28.456X

Figure 14-18.—Plating Solutions.

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 alumimum, 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. It 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.

28.X

thickness of about 1 inch. Numberous turbine blades are at the O.D.

f. Thickness of deposit required—The diameter after truing up the I.D. by grinding must be 3.5015. A plating thickness of 0.001 inch will bring the bore to the middle of the desired tolerance.

Step #2 Select plating solution to be used. Cobalt 2043 meets all requirements.

Step #3 Amp-hr required.

$$A = 3.14\ DL = 3.14 \times 3.50 \times 1.00 = 11.0$$

$$\text{Amp hr} = F \times A \times T = 0.020 \times 11 \times 10 = 2.2$$

Step #4 General approach.

The small area, amp-hr, and thickness involved suggest that (1) a special tool is not required and (2) that the solution need not be pumped. This will be justified in the following steps. The part will be cleaned, etched, rinsed, and so on over a drain and then, being light enough, will be placed over a 14″ × 17″ collecting pan. A hole in the collecting pan will direct the solution back to the solution 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.

An RF-30 tool with a 1/4″ thick cover will just match the I.D.

Step #6 Plating tool contact area.

Although the tool with its cover just matches the I.D., pressure on the tool cover will compact it and lead to perhaps a 50% contact area, or 5.5 square inches.

Step #7 Plating amperage.

$$\text{Plating Amps} = CA \times ACD = 5.5 \times 7 = 38.5$$

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 the solution will be dipped for. The total plating time, therefore, is so short that turning the plate in a lathe or pumping the solution is not necessary and the tool will be moved by hand.

Step #9 Plating solution required.

$$\text{Liters} = Q(L) \times T(I) \times A = 5.4 \times 0.0010 \times 11 = 0.059$$

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, and 1023, and 2080.

b. Tools: AC-5. These, although relatively small, give a 1/2″ × 1″ contact area and should be satisfactory.

c. Quantity of solution required: Approximately 0.1 liter for each tool. 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 lst for the 15-minute plating time.

Step #12 Masking.

Use aluminum tape and contact paper to prevent the part from contaminating the solution.

EXAMPLE #2

Step #1 Information on the job.

a. Number of parts—1

b. Base material—Steel with loose metal spray from a previous repair.

c. Area to be plated—7″ long area on a 2.436 OD

$$\frac{+\,0.001}{-\,0.000}$$

d. Purpose and requirements of deposit—To repair a loose fit on the inner race of a roller bearing.

e. Although the part is a large recirculating fan about 5 feet long 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 the edges. After machining, the diameter was 2.285″. The 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 the buildup at the edges and to improve the surface, a total of approximately 0.100 of inch plating should be planned on.

Step #2 Select the 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.

$$A = 3.14\ DL = 3.14 \times 2.436 \times 7 = 53.5$$

$$\text{Amp-hr(Cu)} = F \times A \times T = 0.013 \times 53.5 \times 1000 = 696$$

$$\text{Amp-hr(Ni)} = F \times A \times T = 0.015 \times 53.5 \times 5 = 4.01$$

Step #4 General approach.

The part will be rotated in a lathe because a lathe is available. The solution will 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 × 2 3/8″ wide × 1 7/8″ high. It will have a 1 1/2″ radius (1/4″ allowance for the tool cover) placed in the 2 3/8″ × 7 1/2″ face. The solution will be fed through an F-handle to a 1/2″ hole in the anode, running in the 7″ direction (capped off at the 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″ × 1″ along the circumference with a soft pad. CA = 3.5 × 1 = 3.5 sq in.

Step #7 Plating current.

Copper—Not required (determined in Step #5).

Nickel—Plating amperage

$$\text{Plating Amps} = CA \times ACD = 3.5 \times 7 = 24.5$$

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 the solution is dipped for total nickel plating, the time will double, to 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) × T(I) × A = 1.29 × 0.100 × 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) × T(I) × A = 1.54 × .0005 × 53.5 = 0.041

Nickel 2085(liter) = Q(L) × T(I) × A = 5.8 × .0005 × 53.5 = 0.155

Since an F-3 tool will be used to apply the nickel, 0.155 liter will not be sufficient to wet the tool and the area to be plated. Approximately 1/2 liter is required.

a. To activate the base material—1010, 1022, 1023, and 2080. To activate the 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 liter (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 the 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, when you encounter a new base material, or when you plan to use a new plating solution. In practicing on a new base material, try shorter and longer operations until you are appearance. In practicing with a new solution, use very high and very low voltages until you are certain that you know what good deposit and bad deposits (burned or otherwise) look like. If possible, run a plating test on a 1″ × 1″ area using an AC-5 or similar size tool; you should be able to plate a good deposit at the volts and amps given in table 14-3 and in 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 the Part for Plating

1. Inspect the area to be plated for any signs of a foreign surface being present such as an electroplate, paint, scale, or anodized coating. Remove the coating by suitable means such as vapor or dry blast, sandpaper, wire brush, and so forth. 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 or perchlorethylene). 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 you plant to move plating tool 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 them on a range.
   c. Putting immersion heaters into the solution.

Base Material – Cast Iron	Area Plated – 3″ I.D. × 2″ Long	Deposit Applied – Nickel 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.

28.X

Figure 14-19.—Sample flow chart for setting up plating.

Solution	Code	on 1 in.^{2}	Volts	Amps	Lit.	Gal.	ft./min.	Optimum	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	49.0	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

28.X

Setting up the Equipment

Set up the power pack near the work so that 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 of 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 the setup so that everything you will use is handy.

General Setup

As the operator, you should be as comfortable as possible, particularly on lengthly plating jobs. You can then concentrate your full attention on the job, you will not be diverted by 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 the necessity for ventilation, gloves, special clothing, and so on.

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 to produce a finer finished product.

GENERAL PREPARATION INSTRUCTIONS

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, and passive films are sufficient to prevent an atomic attraction, thus preventing the adhesion of the electroplate. A preparation cycle is used, therefore, just prior to plating to remove, step by step, all of 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. You must carry out each operation properly to ensure maximum adhesion. You do this when:

1. You use the proper solutions in the proper sequence.
2. You use the solutions one after another are used in the proper direction, in other words forward or reverse.
3. You perform the operations one after another as rapidly as possible without allowing the surface to dry between operations.
4. You obtain the desired results in each operation.

In most operations, you can tell by the appearance of the surface 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 shown in the following pages on preparing specific base materials. When you use a small tool on a small area, use a low voltage in the range. When you use a large tool on a large area, use a high voltage in the range. 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

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

28.X

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. However, reverse current (anodic electrocleaning) must be used whenever hydrogen contamination and embrittlement of the base material must be avoided, such as in the cleaning 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 heat developed in the tool are helpful in cleaning stubborn areas. When you clean the area to be plated, also clean the surrounding area since oil and grease travel on the surface of water. Follow the cleaning with a thorough water rinse. If water "breaks" on the surface, the cleaning and deoxidizing time was too short and you should repeat the operation.

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, the area will develop a uniform, dull, grainy appearance, indicating that you should stop the etching operation. Normally, you will remove 0.000050 to 0.0002 inch of material. 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 the carbon film left on the surface after the etching of a carbon steel. These layers can interfere with maximum adhesion and should be 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, and so on 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. Therefore, you will need to perform an activating operation 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. Avoid contaminating the solution from any source since this operation 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 to help you determine whether or not you have performed the operation properly. The passive film is invisible and on most materials such as nickel and stainless steel you cannot detect a change when it is removed. Any change that *is* apparent may indicate contamination from the activating solution, the anode, or the plating tool. You must, therefore, carry out the operation on a timely basis, spending about 3 seconds on each part of the total area. With an activating tool covering all the area to be plated, spend 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

Follow the final preparatory operation as quickly as possible with the plating operation, whether it is a preplate or the final desired plating. This is of particular importance when your last procedure was an activating operation. DO NOT ALLOW THE PART TO DRY BETWEEN THE ACTIVATING AND PLATING OPERATIONS.

VERIFYING THE IDENTITY OF THE BASE MATERIAL

Obtaining good adhesion of a deposit begins with proper identification of the surface being plated. You will be frequently misinformed about the identity of the base material and whether or not a coating is present. This, of course, can lead to adhesion problems. However, by carefully watching the etching operation, you can frequently detect incorrect identifications or the presence of coatings. The following descriptions may help you make these determinations. (Also refer to table 14-3.)

Result of No. 2 or No. 4 Etching Reverse Operation

Material	Appearance of the Etched Surface	Color of the Solution in the 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

PREPLATING INSTRUCTIONS

It may be necessary, in some cases, to apply a preplate with an appropriate plating solution. Apply the preplate immediately after you prepare the surface. After you finish applying the preplate, immediately rinse the surface with water and plate 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 results from 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
3049	5	8

SUMMARY OF ELECTROPLATING

The ideal plating operation is carried out when (1) the quality of deposit is the best possible;

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	2080	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

(2) the deposit is applied in a minimum amount of time; and (3) the deposit has a uniform desired thickness happen simultaneously.

You should have taken a number of steps in the initial and final preparations to ensure that you could apply the best possible quality deposit. Some of them include use of clean anodes, uncontaminated solution, and proper cover material. You, at the time of plating, however, must still carry out the operation properly.

Guidelines for the Operator

The operator guidelines 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 quality deposit.

KEEP THE AREA BEING PLATED CLEAN.—Contamination of the area by oil, grease, dirt, and so on 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 the solution moving over a dirty surface; correct this 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 next 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 you dip 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. You cannot obtain a good bond without activation. To prevent passivation, avoid all unnecessary interruptions of the plating operation, minimize the length of time of unavoidable interruptions, and ensure that you cover all areas being plated periodically (at least every 10 seconds) during plating.

PREVENT THE SOLUTION FROM DEPLETING IN THE WORK AREA.—Depletion of the solution in the work area (where the cover meets the part) has various effects depending on the solution. With most plating solutions, there is a greater tendency for depletion to produce 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 into 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 a flat part then move it in a straight back and forth motion instead of in a rotary motion or move the tool in the direction of rotation of a rotating part, in some spots you momentarily have no relative movement. Burning of the deposit can result from this.

USE VISUAL CONTROL.—While you plate, you can see what the deposit looks like as it goes on. Its appearance gives you 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 make appropriate corrections, such as 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 the plating's appearance while plating, and be able to make appropriate corrections.

Evaluating Deposits

The qualities to look for in all plating deposits are good adhesion, proper thickness of the coating, and high density of deposit. In corrosion protection applications where you use non-sacrificial coatings, also be sure there are no pores or surface-to-base metal cracks.

Evaluating Adhesion

Some of the tests that you can use to see how well the deposite has adhered to the base metal are (1) the chisel, knife, and scratch tests or (2) the grind and saw test.

CHISEL, KNIFE, AND SCRATCH TESTS.—If the deposit is sufficiently thick to permit the use of a chisel, test the adhesion by forcing the chisel between the coating and the base 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 from the base material.

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 unplated base metal to the deposit. If adhesion is poor, the deposit will be torn from the base. You can use a hacksaw instead of the grinder, as long as you saw in a direction that tend to separate the coating from the base metal. Grinding and sawing tests are especially effective on hard or brittle deposits.

TROUBLESHOOTING

Poor Adhesion

Carefully inspect the plated area to determine at which stage in the plating process the separation occured. Examine the back side of the material coming off. Perform two etch tests using either Code 1022 or 1024 solution. Etch part of the area where the material came off and the base material of the part in an area where etching will not cause a problem. Compare the appearance of the two areas to determine where the separation occurred. If the two areas are identical, the failure occurred at the base material. If they are different, the failure occurred between two of the plated layers.

In some cases, such as when you plate on metal spray, tungsten carbide, electroless nickel, and so on, the separation is in the base material, and, therefore, the DALIC plating cannot be faulted.

1. COMMON CAUSES FOR THE DEPOSIT COMING OFF OF THE BASE MATERIAL
   a. The base material was not correctly identified.
      (1) Determine, for certain, the identity of base material.
      (2) Determine if the surface was etched as it should have been
   b. The surface has a foreign coating such as metal spray, chrome plate, and so on.
      (1) Determine if the plated area and other areas on the workpiece etch the same. Use Code 1022 or 1024 solution and reverse current.
   c. The preparatory procedure was not thoroughly, properly, and quickly carried out.
   d. Contaminated preparatory solutions were used.
   e. Contaminated preparatory or preplate tools were used.
   f. The surface was not pre-wetted before it was plated.
   h. The wrong plating solution was used.
   i. An improper preplate was used.

2. COMMON CAUSES FOR THE FINAL DEPOSIT COMING OFF OF THE PRE-PLATE
   a. The preplate was not followed quickly by the final plating solution.
   b. The surface was not pre-wetted according to manufacturer's instructions.
   c. The wrong plating solution was used.

3. COMMON CAUSES FOR THE FINAL DEPOSIT COMING OFF OF ITSELF
   a. The deposit was burned.
   b. The plating operation was interrupted for too long.
   c. The solution was contaminated.
   d. The anode was contaminated.
   e. The wrong anode cover was used.

stress-cracking, and stress-crack lifting. Common causes are as follows:

1. The wrong solution was used.
2. The plating solution was contaminated.
3. The plating tool was contaminated.
4. The wrong cover was used or the cover was too thick or too thin.
5. The plating method was wrong.

Low Thickness Deposit

The deposit did not achieve the desired thickness. Common causes are as follows:

1. The operator did not properly calculate the area.
2. The operator did not properly calculate the amp-hr.
3. Considerable plating went on the aluminum tape or adjacent areas.
4. The operator overetched the base material.
5. The operator plated wrong with a "variable factor" solution.
6. Certain solutions were overused.
7. The supply of solution to the tool was insufficient.
8. Plated in the cover. Wash and examine the cover to see if this actually occurred.

Nonuniform Thickness of the Deposit

1. The wrong tool was used.
2. Tool was not used correctly.
3. The solution was not distributed uniformly in the cover.
4. The tool cover thickness varied.

Took too Long to Finish the Job

1. The wrong solution was used.
2. The plating tool was too small.
3. The power pack was too small.
4. The operator did not plate as fast as possible with the existing tool or solution.
5. The operator did not properly preheat certain variable factor solutions.

MACHINING AND GRINDING

The following paragraphs discuss basic requirements for machining and grinding plated deposits.

easily machined and do not require specific recommendations.

Cobalt, iron, and nickel deposits or their alloys are difficult to machine. If possible, grind rather than machine these deposits. When it is absolutely necessary to machine, use good equipment and a good technique. 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, such as approximately 50 ft/min.

Grinding Nickel and Cobalt Deposits

The Norton Company, Worcester, Massachusetts, makes the following recommendations concerning the grinding of nickel or cobalt deposits:

1. Wet grinding recommended. Use plenty of coolant.
2. Wheel—C36K6V.
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 the 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

$$\text{Amp-Hr} = F \times A \times T$$

Use this formula to determine the ampere-hours that should pass during plating to provide the desired thickness of deposit on the area to be plated.

In this formula, F is the factor you obtain from the plating solution bottle or from table 14-5.

A = area of the surface to be plated in square inches.

T = thickness of the deposit desired measured in ten-thousandths of an inch.

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: You can determine the proper value for T to put in the above formula by writing the thickness desired in inches and then moving the decimal point four (4) places to the right.

Example:

You desire a thickness of 0.001 inch. Since 0.001 is the same as 0.0010, move the decimal point four places to the right to get 0.010. T, therefore, is 10.

Formula 2: Formula to determine the 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 you to compute the current density at which you are plating in a given operation. You can then make comparison with values given in table 14-3 to determine if you are plating at a low current density, a normal current density, or an excessive current density. You can use this information to make appropriate adjustments while plating.

Formula 3: Formula to determine the optimum plating tool contact area when you design special tools

$$OCA = \frac{MA}{ACD}$$

MA = maximum aperage output of the power pack to be used

ACD = average current density for the solution to be used

Use this formula when you design special tools to develop the right size tool, neither too large nor too small.

Formula 4: Formula to estimate the plating amperage to draw with a given solution and plating tool

$$PA = CA \times ACD$$

ACD = average current density for the solution

Use this formula for two purposes:

1. In conjunction with Formula 5 to estimate plating time.
2. By itself to determine if you are plating at the right amperage.

Formula 5: Formula to estimate the plating time

$$PT\ (Hrs) = \frac{Amp\text{-}Hrs}{PA}$$

PA = the value from Formula 4 for purpose 1 or the average current while plating for purpose 2

Use this formula is used for two purposes:

1. To estimate the plating time in setting up a job
2. To control the thickness when no ampere-hour meter is available

Formula 6: Determining amount of solution required

$$Liters = Q(L) \times T(I) \times A$$

Use this formula (1) in estimating jobs and (2) to ensure that you have the appropriate amounts of solution to use in a given job.

T(I) = thickness of the deposit desired in inches

A = area of the surface to be plated

Formula 7: Formula to check ampere-hour meter accuracy

$$\text{Amp-Hrs} = \text{Amps} \times \text{Hrs}$$

Use this formula periodically as a maintenance procedure to ensure that the amp-hr meter is accurate, or in cases where you suspect its accuracy.

Run the test by shorting the d.c. output leads and running the power pack for a set time (hr) at a

Amps = 20

$$\text{Hrs} = \frac{3}{60} \text{ or } 0.05$$

Placing these values in the above formula

$$\text{Amp-Hr} = 20 \times 0.05$$

$$\text{Amp-Hr} = 1.00$$

The computed value (1.00) should be close (within a small percentage) to that passed on the amp-hr meter when you short the d.c. output leads and run the test for 3 minutes at 20 amps.

THE REPAIR DEPARTMENT AND REPAIR WORK

As a Machinery Repairman you may be assigned to almost any type of 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 these ships, you will be part of the repair department and should know something about its functions, personnel, and shops. This chapter, will teach you about the repair department and will give you some examples of repair work you are likely to encounter.

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 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 specific 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 best known types of tenders are the destroyer tender (AD) and the submarine tender (AS). Submarine tenders are capable of tending both conventional submarines and fleet ballistic missile submarines; however, individual ships specialize in either conventional submarines or ballistic missle submarines. The organization of the repair department of an AS that tends conventional missile submarines differs somewhat from that of an AS that tends fleet bassistic missile 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 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).

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. The repair

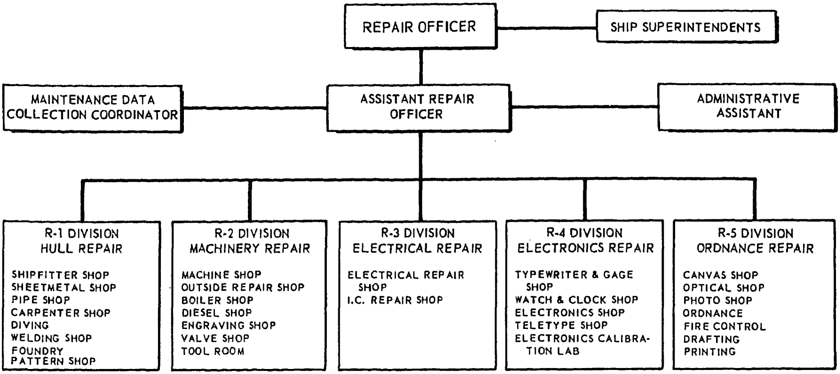


Figure 15-1.—Organization of the repair department in a typical repair ship (AR).

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 the following actions:

- Accomplishing repairs and alterations to the ship itself (tender or repair ship) which are beyond the capacity of the engineering department or other departments.
- Maintaining a well-organized and efficiently operated department.
- Issuing and enforcing repair department orders which govern department procedures.
- Enforcing orders of higher authority.
- Knowing the current workload and capacity of the ship's crew and facilities, and keeping the staff maintenance representative informed of their current status so that the maintenance representative may properly schedule and assign ships.
- Reviewing work requests received via the staff maintenance representative from the ships assigned for repair and for accepting or rejecting the individual jobs according to the capacity of the repair department.
- Reviewing and accepting any work lists or work requests which develop after an availability period has started.
- Operating the department within the allotment granted and initiating requests for further funds, if required.
- Ensuring the accuracy, correctness, and promptness of all correspondence, including messages, prepared for the commanding officer's signature.
- Reviewing 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 departmental conditions and to ensure adequate standards, the repair officer 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:

- Planning, preparing, and carrying out schedules for alterations and repair work assigned to the repair department.

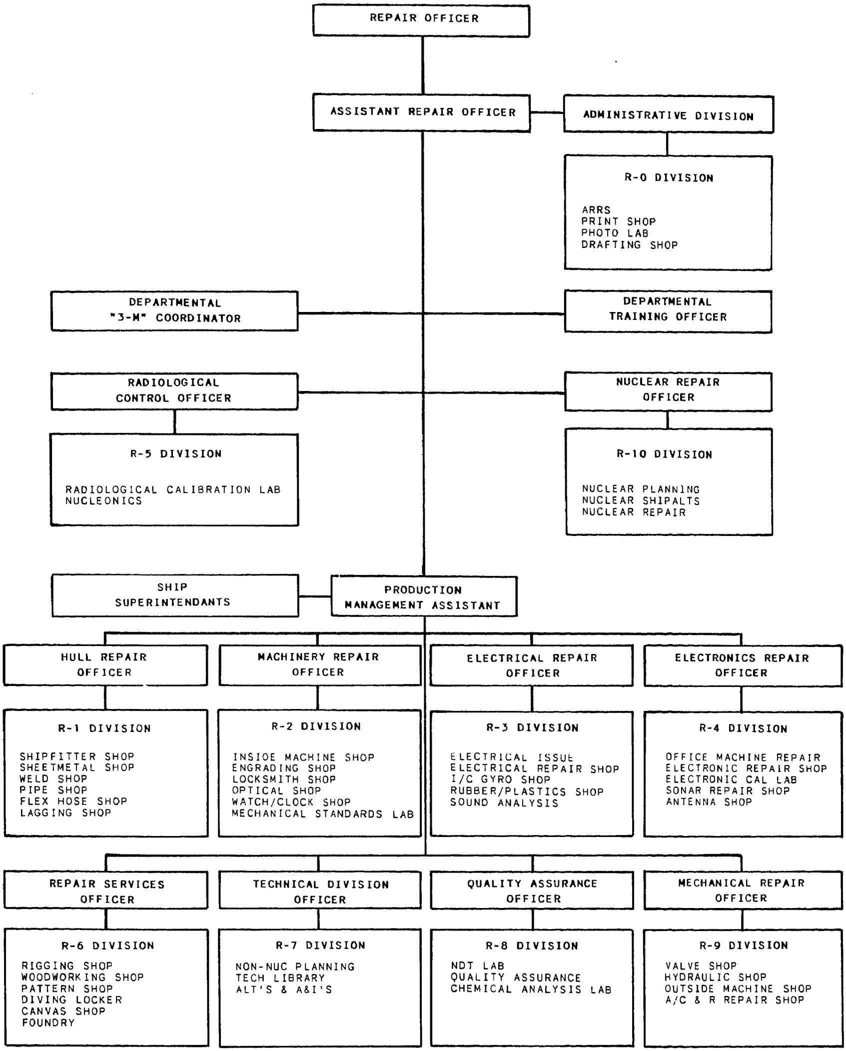


Figure 15-2.—Organization of the repair department in fleet ballistic missile submarine tender (AS).

- Establishing and operating the Planned Maintenance System of the 3-M System.
- Coordinating repair capabilities, work assignments, and available personnel to ensure maximum use of manpower.
- Supervising and inspecting repairs and service to ensure timely and satisfactory completion of work; providing controls for quality control.
- Preparing records, reports, forms, and orders in connection with repair functions and duties.
- Ensuring proper operation of all equipment and material assigned to the repair department.
- Ensuring strict compliance with safety precautions and security measures.
- Reporting to the commanding officer the progress of major repairs and alterations; keeping the executive officer informed; reporting promptly any inability to meet scheduled completion dates.

ASSISTANT REPAIR OFFICER

In the absence of the repair officer, the assistant repair officer assumes the responsibilities of the repair officer. The assistant repair officer is the personnel administrator for the repair department, and is responsible for the assignment of personnel, the administrative control of the repair office, and the departmental 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:

- Assigning personnel to divisions, schools, shore patrol, and beach guard.
- Having a basic knowledge of courses, schools, and rating programs necessary to further the education of personnel and their advancement in rate for their benefit and that of the ship and the Navy.
- Maintaining the office stores and accounts.
- Assisting 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.
- Reviewing all work requests as they are received.
- Assigning work and priority ratings to the division and its shops.
- Maintaining liaison with the supply department for materials on order or to be ordered for the work requested.
- Procuring the necessary blueprints, sketches, or samples for the shops.
- Scheduling the services of tugs, cranes, and technical services, as available, for successful completion of an availability.
- Surveying reports from each shop to ascertain the successful completion of all work during the allotted time.
- Analyzing man-hour shop reports to determine an even balance of work versus personnel assigned.
- Coordinating the actions of the repair office and the shops to keep the repair facilities fully productive.

In addition to the assistant repair officer, there are usually several other officers who assist the repair officer in performing repair department functions. These may include a production engineering assistant, a repair assistant, a radiological control officer, a department training officer, a production management assistant, 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, 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 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

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.

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.

REPAIR DEPARTMENT SHOPS

Each shop in the repair department is assigned to one of the divisions. As a Machinery 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 the 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 another for assistance.

MACHINE SHOP

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 submarine tender.

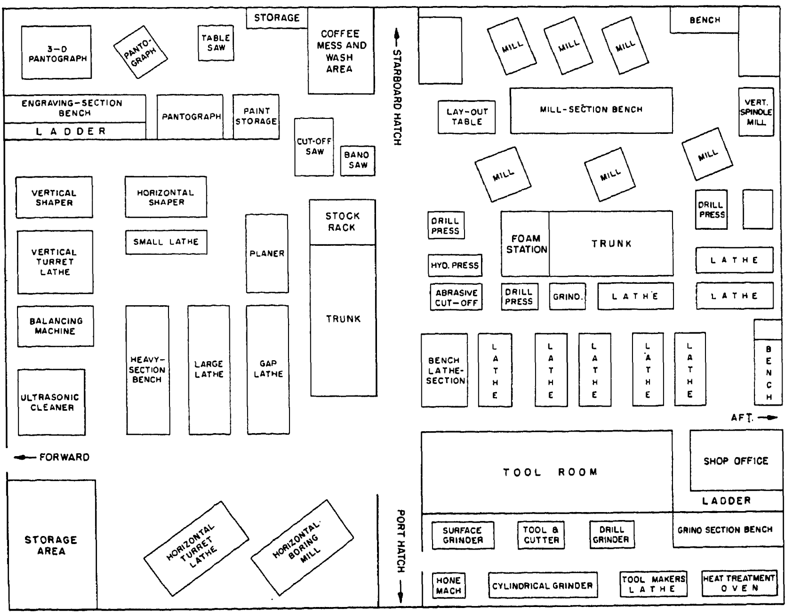


Figure 15-3.—Machine shop layout (submarine tender).

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 are a toolroom and a 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.

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 become familiar with 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 a job that requires coordination is the making of hatch dogs. The pattern shop makes the pattern, the molders cast them in the

28.314

Figure 15-4.—Machine shop lathe section.

28.315

Figure 15-5.—Machine shop milling section.

foundry, the machine shop machines them, and 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

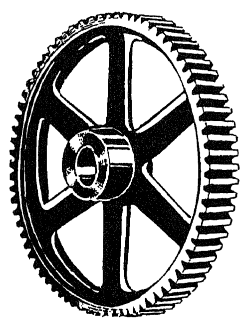


Figure 15-6.—Part made in a machine shop.

work because of alignment problems in the machining operation.

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. To do this, you will have to use specific formulas, which you can find in various machinist's handbooks and in *Mathematics, Volume 1*, NAVPERS 10069 (series).

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 you manufacture gears, you may need to calculate simple gear trains or gear trains using compound gearing. Information on this subject is contained in *Basic Machines*, NAVPERS 10624 (series).

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, in adition to how to index the blank, so that the teeth are equally spaced and have the correct profile.

Spur Gear Terminology

The following terms (see fig. 15-8) are used to describe gears and gear teeth (symbols in

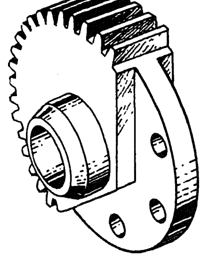


Figure 15-7.—Cutting specially shaped grooves.

parentheses 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; the 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.

ADDENDUM (ADD): The height of the part of the tooth that extends 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.

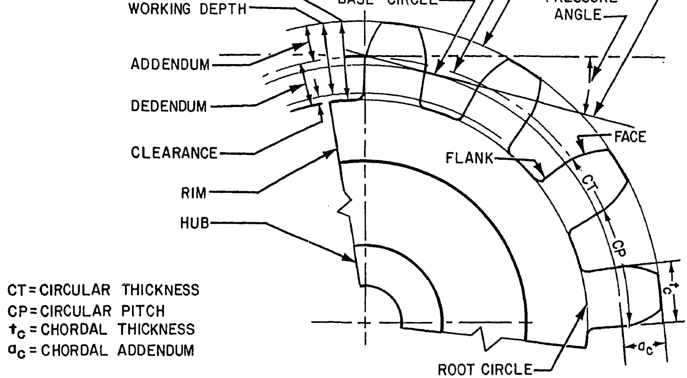


Figure 15-8.—Gear terminology.

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 the circle that bounds the bottom of the gear teeth.

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 the chordal thickness line at the pitch circle (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 number of teeth on the gear divided by the number of inches of pitch diameter.

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	(none)
Pitch Diameter	PD	D
Center to Center Distance	C-C	C
Addendum	ADD	a
Dedendum	DED	d
Working Depth	WKD	hk
Clearance	CL	c
Whole Depth	WD	ht
Root Circle	RC	(none)
Outside Diameter	OD	Do
Circular Thickness	CT	tc
Circular Pitch	CP	P
Diametral Pitch	DP	P
Number of Teeth	NT	N
Root Diameter	RD	DR
Chordal Thickness	t_c	(none)
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 it 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 you use 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:

$$PD = \frac{(ND)\ OD}{NT + 2}$$

$$PD = \frac{24 + 3.250}{24 = 2} = \frac{78}{26} = 3.000 \text{ inches}$$

2. Find the diametral pitch (DP) using the formula:

$$PD = \frac{NT}{PD}$$

$$PD = \frac{24}{3} = 8$$

3. Find the whole depth of tooth (WD) by using the formula:

$$WP = \frac{2.157}{DP}$$

$$WP = \frac{2.157}{8} = 0.2696 \text{ inch}$$

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. The number of cutter that you should use depends upon the number of teeth the gear will have. The following chart shows which cutter to use to cut various numbers of teeth on a gear.

If, for example, you need a cutter for a gear that has 24 teeth, use a number 5 cutter since as a number 5 cutter will cut all gears containing from 21 to 25 teeth.

Range of teeth	Number of cutter
135 to a rack	1
55 to 134	2
35 to 54	3
26 to 34	4
21 to 25	5
17 to 20	6
14 to 16	7
12 to 13	8

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 you calculate the chordal addendum, you must determine the addendum (ADD) and circular thickness (C_t).

To determine the addendum, use the formula:

$$ADD = \frac{PD}{NT}$$

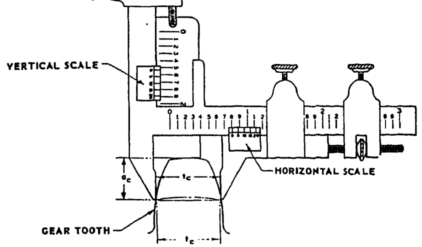


Figure 15-9.—Measuring gear teeth with a vernier caliper.

Using values from the preceding example,

$$\text{ADD} = \frac{3.000}{24} = 0.125 \text{ inch}$$

To determine the circular thickness, use the formula:

$$\text{CT} = \frac{1.5708}{\text{DP}}$$

Using the values from the example,

$$\text{CT} = \frac{1.5708}{8} = 0.1964 \text{ inch}$$

The formula used for finding the chordal addendum is

$$a_c = \text{ADD} + \frac{(\text{CT})^2}{4(\text{PD})}$$

$$= 0.125 + \frac{(0.1964)^2}{4 \times 3}$$

$$= 0.125 = \frac{(0.0386)}{12} = 0.128 \text{ inch}$$

The formula for finding the chordal tooth thickness is

$$t_c = \text{PD} \sin \left(\frac{90°}{\text{NT}}\right)$$

$$\left(24 \right)$$

$$= 3 \times \sin 3°45''$$

$$= 3 \times 0.0654$$

$$= 3 \times 0.1962 \text{ inch}$$

(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 whole depth (WD) is too shallow; if the reading is less, the whole depth (WD) 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, you can put these dimensions into one or more of the formulas in Appendix II and calculate the required dimensions.

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 excess material on the diameter and thickness of the blank for cleanup cuts.
2. Mount 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 bore required.
3. Drill and bore to the required size (within tolerance).
4. Remove the blank from the lathe and press it on a mandrel.
5. Set the mandrel up between the centers of the index head and the footstock on the milling machine. Dial in within tolerance.
7. Select a number 5 involute gear cutter (8-pitch) and mount and center it as described in chapter 11.

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 hand 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 you machine a rack, space the teeth by moving the work table an amount equal to the circular pitch of the gear for each tooth cut. Calculate the circular pitch by dividing 3.1416 by the diametral pitch:

$$CP = \frac{3.1416}{DP}$$

You do not need to make calculations for corrected addendum and chordal pitch 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.

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 circled numbers indicate a sequence of operations for machining the various surfaces of the shaft.

Select and cut a piece of round stock 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 10 11/16 inches. Most of the linear dimensions in figure 15-10 are given in the form of mixed numbers of 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.

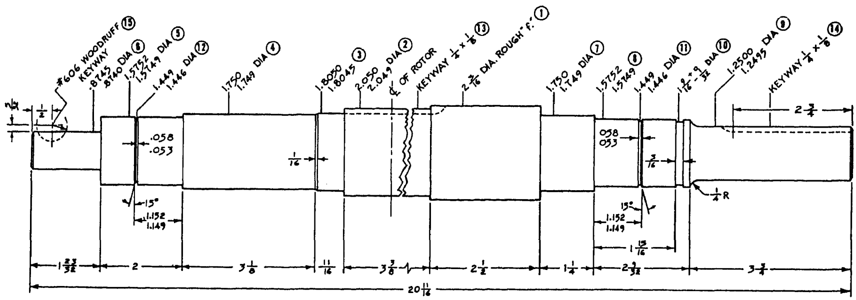


Figure 15-10.—Steps in making a shaft.

Plain turning required on surfaces 1 through 6 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 (15). To machine 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 11 3/32 inches and lay out the shoulder between the 2 3/16 inch diameter and the 2.050 inch finish diameter. Using the crossfeed handwheel with the micrometer collar set on zero, feed the tool in 0.068 inch (one-half of the difference between 2.050 and 2 3/16). Make a short length of cut at the end of the shaft and measure the diameter with a micrometer. Adjust the crossfeed handwheel as required to provide the $2.050 \begin{smallmatrix} +.000 \\ -.001 \end{smallmatrix}$ 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 illustration.

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, position the tool (by rule measurement) 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 rest 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.

to the required dimensions.

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 step:

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 a center.

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 within acceptable limits.

If little or no change in runout results from the first straightening attempt, spring the shaft further in the second operation to overcome the elasticity of the shaft so that it bends in the required direction. It is better to make several attempts to straighten the shaft a few thousandths of an inch at a time than to attempt to straighten the shaft in one or two tries with the possibility of bending the shaft too far in the opposite direction.

Damaged ends of shafts can be repaired by removing the bad section and replacing it with a new "stub" end. Check to see if the type commander allows stubbing of shafts.

Take the following steps to stub 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), of the same material as the shaft, 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 2 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 stud 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 the 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 (series). The information supplied here applies to globe, ball, and gate valves but the procedures discussed can usually be adapted for repairing any type of valve.

Globe Valve

Closely inspect the valve seat and disk for erosion, cuts on the seating area, and proper fit of the disk to its seat. Inspect all other parts of the valve for wear and alignment and, if you find them defective, repair or renew them. 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 with prussian blue 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 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 of the disk face surface and apply a thin, even coat of prussian blue on the contact face of the seat. 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 not 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 contact surfaces of the seat and disk. This 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,

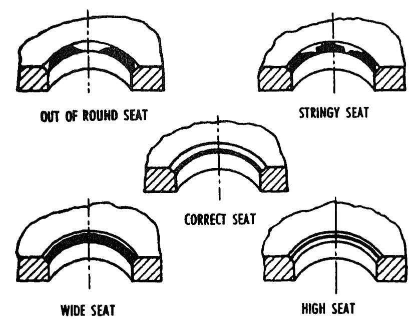


Figure 15-11.—Examples of spotted-in valve seats.

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 using such a tool, you can remove slightly larger irregularities from the seat than you can 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.

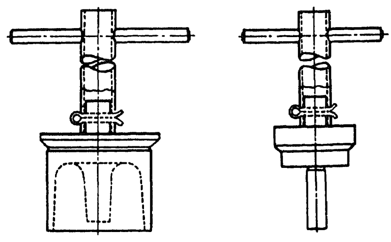


Figure 15-12.—Lapping tools.

5. Use only clean compound.
6. Replace the compound often.
7. Spread the compound evenly and lightly.
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. When you complete the lapping job, spot-in and grind-in the disk to the 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 corase grade: may be used also at the start of the 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.

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, replaced with a new seat.

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 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 remove the existing weld material by machining and then rebuild the seating surface with successive deposits of new weld material. After you have made a sufficient deposit of weld material, 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 the ring from the valve body. You may then either

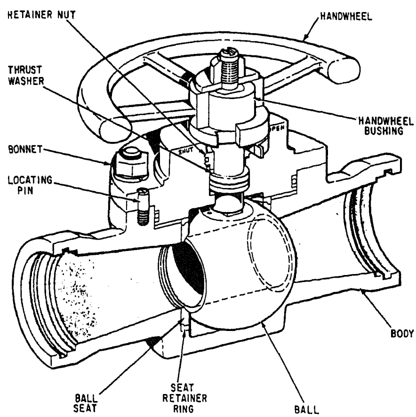


Figure 15-14.—Typical seawater ball valve.

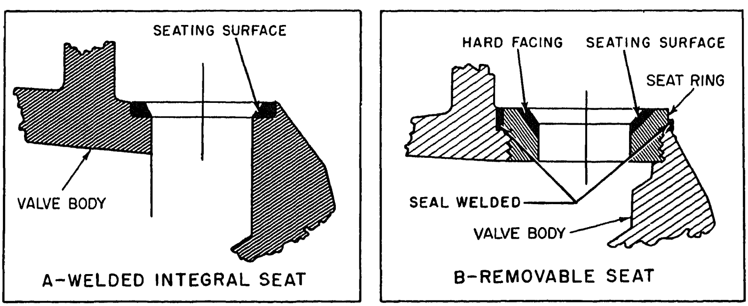


Figure 15-13.—Valve seat construction.

maintain the existing seat contact as previously described.

The actual machining operations for valve seats and disks are described in chapter 8. After you have completed the machining, spot-in, lightly grind-in, and respot the seat and the disk to ensure that the valve disk-seat contact is as it should be.

Ball Valve

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 operated by planetary gears. This type of gearing requires a relatively small handwheel and opening force to operate a fairly large valve. The gearing does, however, increase the time for opening and 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

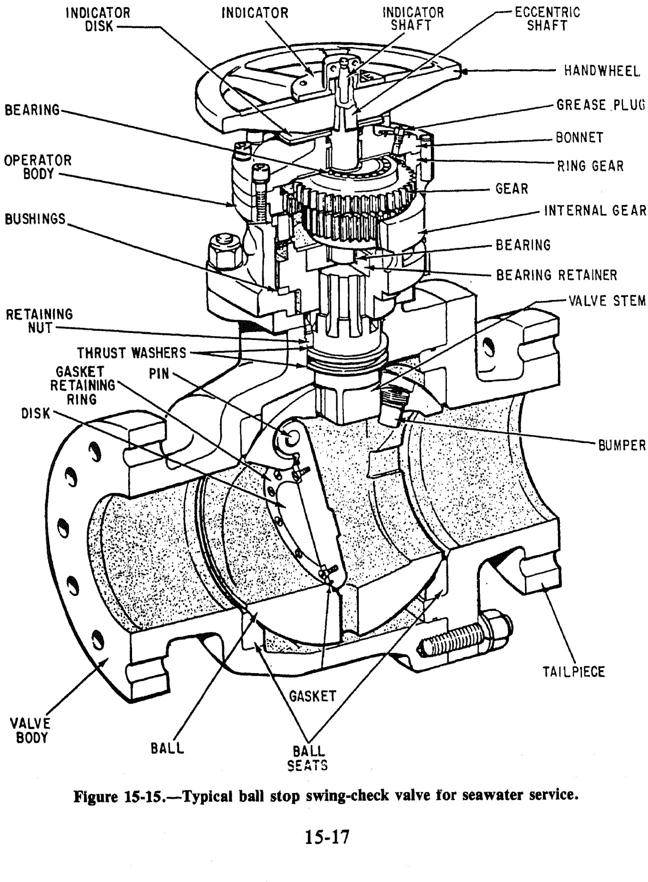


Figure 15-15.—Typical ball stop swing-check valve for seawater service.

with planetary gear operation. Ball valves are normally found in the following systems onboard 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 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 or eroded to the extent that its function is reduced and serviceability is affected. The repair of ball valve waterway lips in this instruction applies only to straight waterway valves whose stem connection does not enter the waterway. 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. The detailed repair steps are in Portsmouth Process Instruction Number 4820-921-339B of 24 June 1977, which cancels number 4820-921-339A.

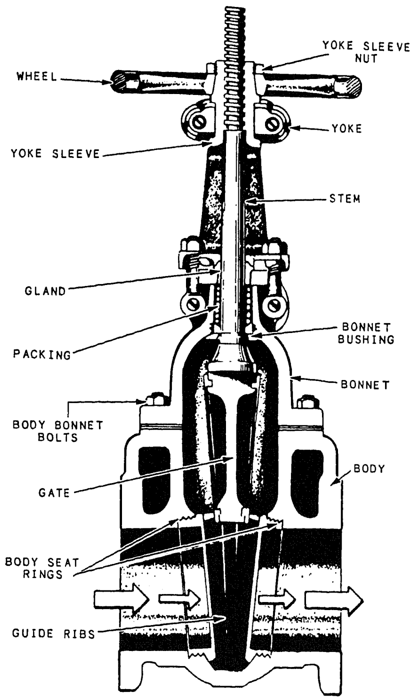


Figure 15-16.—Cutaway view of a gate stop valve (rising stem type).

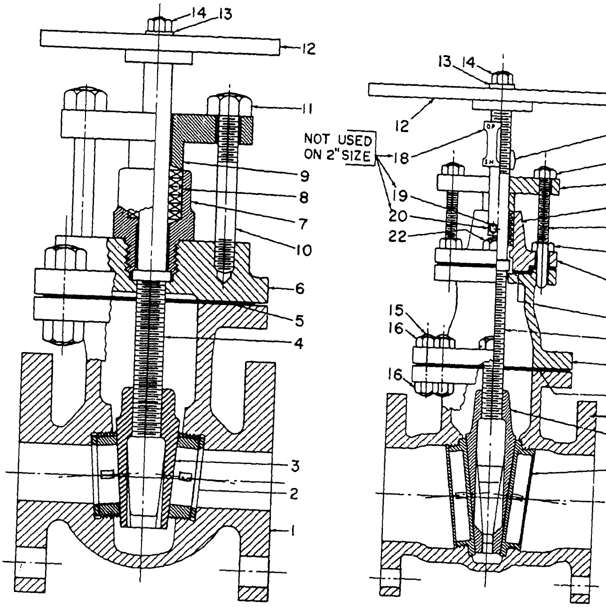
Gate Valve

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 its 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 of valve almost always has a pointer type indicator threaded onto the upper end of the stem to indicate the gate's 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



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	PACKING	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		

Figure 15-17.—Cross-sectional views of gate stop valves (nonrising stem type).

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. (Note: Shim has to be applied behind both seats to maintain the proper 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 special valves called constant-pressure governors. A constant-pressure 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 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 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

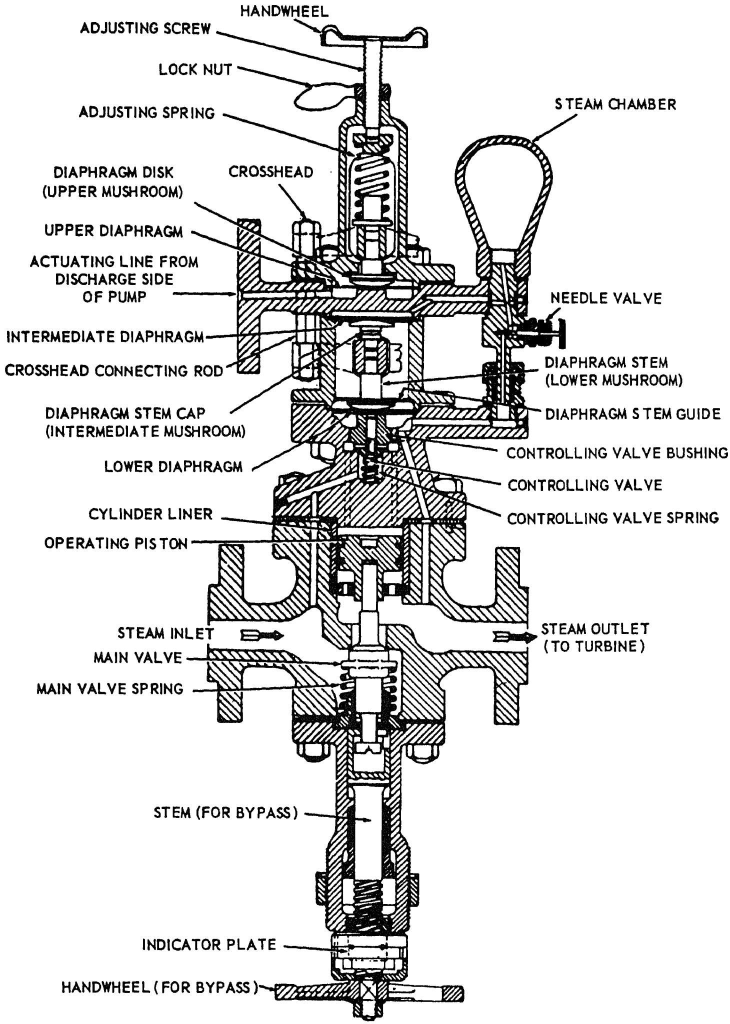


Figure 15-18.—Constant-pressure governor for main feed pump.

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 use of the handwheel, 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

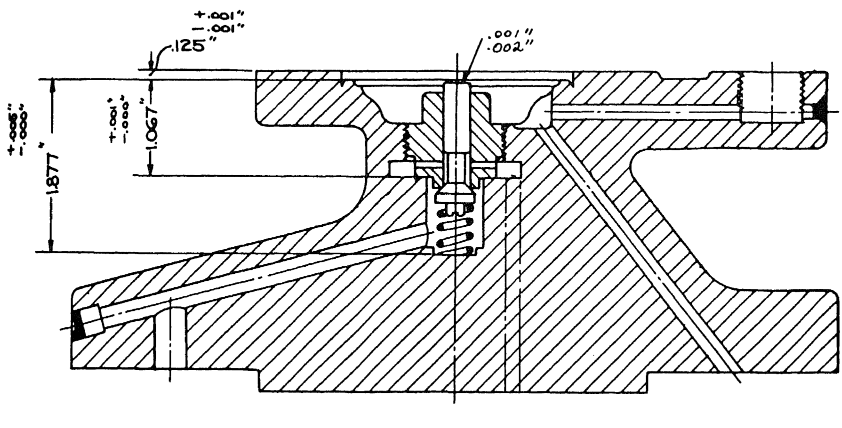


Figure 15-19.—Critical dimensions of the Leslie top cap.

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 they are available.

TOP CAP.—If the top flange of the top cap of the governor becomes damaged, you must be extremely careful when you machine 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 the 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, and 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 them.

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. 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.

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 way 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 cost 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. Consult local type commander and quality assurance procedures to find out which method is best suited for your situation. Check with the shop's leading petty officer before you undertake 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

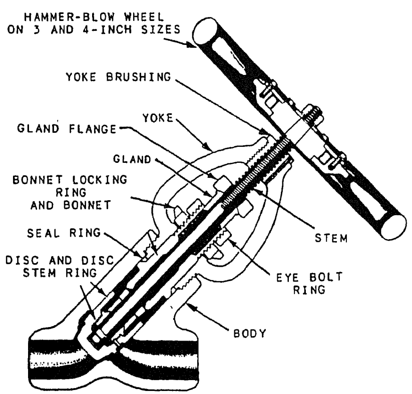


Figure 15-20.—1500-pound pressure seal bonnet globe valve.

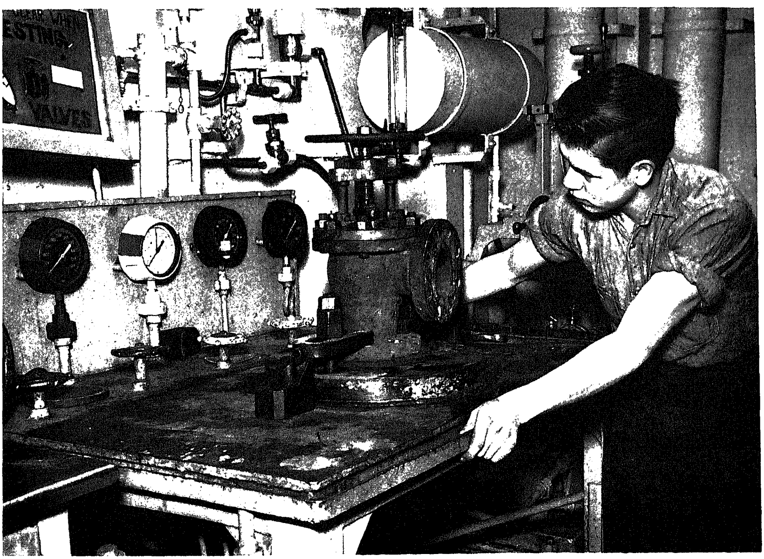
type, and you will be involved in machining the bonnet seal area to specifications provided by either the applicable blueprint or the Hull Technician doing the welding. This basic type valve is used in steam systems; it is also commonly found in the nuclear systems in submarines and submarine tenders. This type of valve is also referred to as canopy seal valve. In some instances you may be required to work closely with the radiological control division since these valves are used in nuclear systems that must be closely monitored for radiation levels and possible contamination of equipment and tools used during the repair procedure. Inn 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 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 "A" stamped on the crown. If you do not see such an identification on a stud, 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 torque wrench to measure for correct tightening tension



28.263

Figure 15-21.—Applying a hydrostatic test to a high-pressure steam valve.

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.

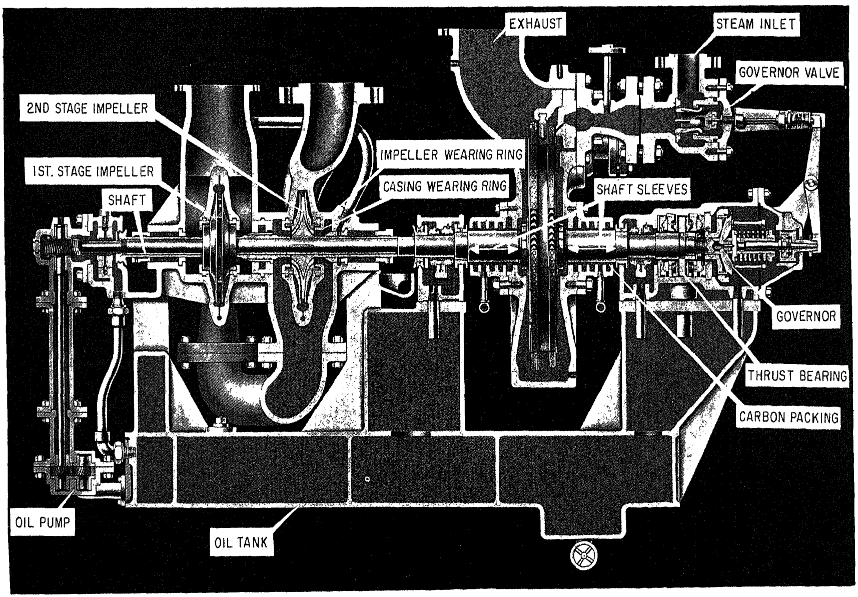
After you finish applying a test pressure to the lower part of the valve, turn the valve over, with the other flange down, and test the bonnet joint.

When you test 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 onboard ship is provided in *Fireman*, NAVEDTRA 10520 (series). 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.



38.109

Figure 15-22.—Two-stage main feed pump.

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 during operation. They must be renewed from time to time to maintain the 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 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 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 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. The centrifugal pump sleeve-shaft-shoulder joint is usually made up with a hard fiber wash to prevent liquid from leaking through the joint and out of the pump between sleeve and the 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 clearance between the casing rings and the 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

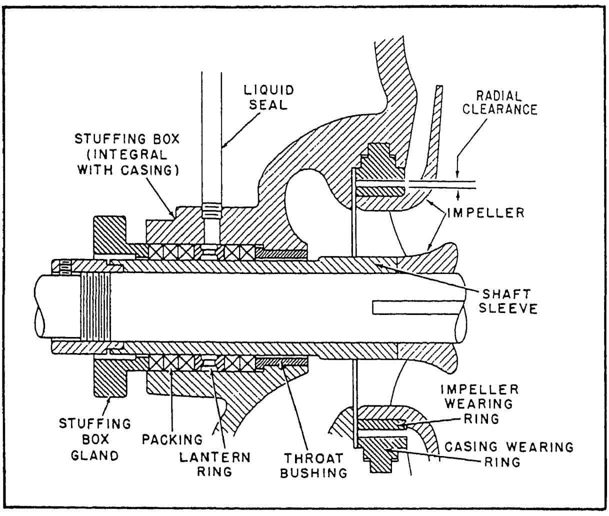


Figure 15-23.—

the outside diameter of the impeller wearing ring, and the inside 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 you proceed 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 rings. 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. (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 its axis with the lathe axis.

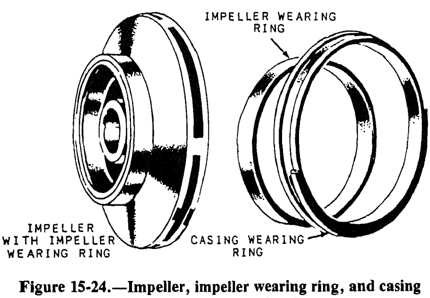


Figure 15-24.—Impeller, impeller wearing ring, and casing wearing ring for a centrifugal pump.

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 their 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 them 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 technical manuals list the desired clearance as either diametrical clearance or radial clearance. Diametrical clearance is the total amount of clearance required. Radial clearance is one-half of the clearance required and must 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 and 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 (series), 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 depend on 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). A rust prevention program should be a part of your daily cleanup routine. Keep all bare metal surfaces clean and bright, and apply a light coat of machine oil to protect them. Use an approved rust preventive compound to help keep decks, bare metal surfaces, and machinery parts from rusting.

It is sometimes said that a machine tool operator can be judged by the condition of his or her 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 you apply 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 you use 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 if necessary.

- Before you take an electric power-driven handtool from the toolroom, examine it carefully for mechanical and electrical defects and ensure that the electrical safety tag is current.

- When you secure 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.
  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 locking pliers, as shown in

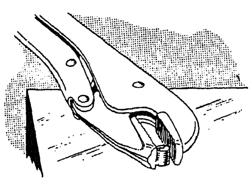


Figure 15-25.—Removing a broken stud with locking pliers.

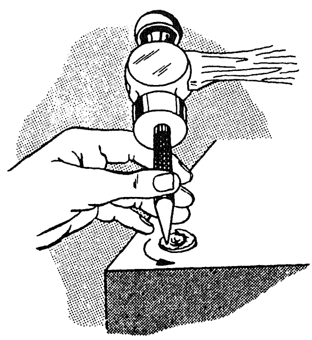


Figure 15-26.—Removing a broken bolt with a prick punch.

Table 15-1.—Chart for Screw and Bolt Extractors

Extractor		Used For—		
Size No.	Overall Length, Inches	Nominal Screw And Bolt Size, Inches	Nominal Pipe Size, Inches	Use Drill Size Dia., 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

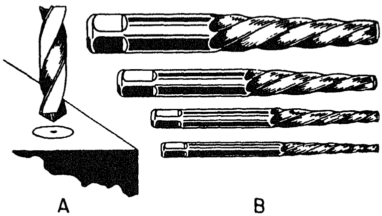


Figure 15-27.—Screw and bolt extractors for removing broken studs.

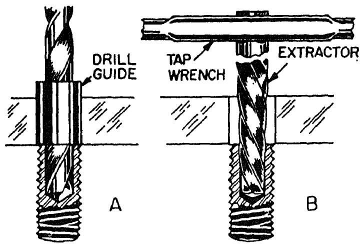


Figure 15-28.—Removing a stud broken off below the surface.

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 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 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.) 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.

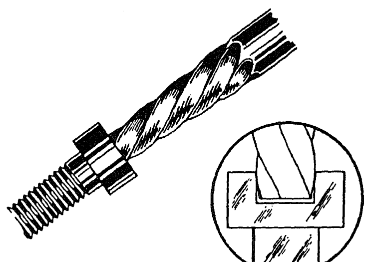


Figure 15-29.—Removing an Allen head capscrew with a bolt extractor.

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

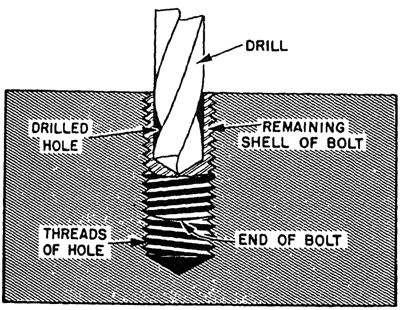


Figure 15-30.—Removing a broken bolt and retapping the

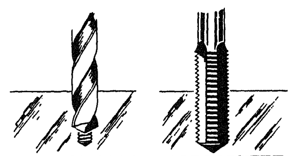
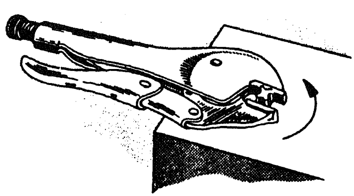


Figure 15-31.—Removing a broken bolt and retapping the hole to a larger size.



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 clean threads and continue tapping until you have cut away the shell and restored the original threads.

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 the original capscrew or stud with a larger size.

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 locking pliers. This operation is shown in figure 15-32. Carefully ease the tap out of the hole, adding penetrating oil as necessary.

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 loose 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 quickly heat the immediate area around the hole with an oxyacetylene torch. This quick heating expands the adjacent metal of the work, allowing you 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.

MAKING PISTON RINGS

To make a cast iron piston ring, select a billet of sufficient size to permit you to remove 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 you to remove 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.

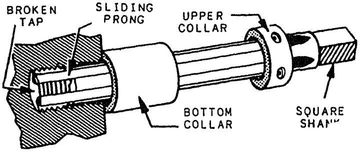


Figure 15-33.—Removing a broken tap with a tap extractor.

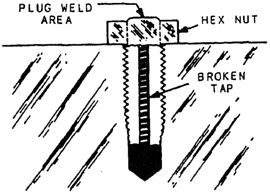


Figure 15-34.—Using a plug weld to remove a broken tap.

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 the required 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 it to the 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 you turn. 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 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, and so forth, 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 tension is maintained by twisting the wire as the spring is wound. A common example of such a spring is the ordinary screen door spring. When in a static condition (before being installed on a door), these springs will not begin to stretch as soon as the load is applied. The load must first overcome the initial tension already in the spring.

TABLES FOR SPRING WINDING

When springs are to be wound on 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 gauge. 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, for example, that you must wind a compression spring of No. 10 Brown and Sharpe gauge wire. From the table, you will note that this spring should have four and one-half coils per inch. Gear the lathe as you would to cut four and one-half threads per inch.

Table 15-3 gives data for winding piano wire tension springs. 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 would 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 the spring and the shaft. The wire sizes given in the table conform to the English music wire gauge.

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 lengthwise in the brass to hold the wire in place. 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 is given in the *Machinery's Handbook*.

Tension Spring — I Compression Spring — II												
Number of Wire Gage	Brown & Sharpe		Birmingham or Stub's		Washburn & Moen Mfg. Co.		Trenton Iron Co.		Prentiss		Old English Brass Manufacturers'	
	I	II	I	II	I	II	I	II	I	II	I	II
000000	......	.....	......	......	2	1	......	.....	.....	.....	...	.....
00000	......	.....	......	......	2 1/4	1 1/8	2	1	.....	.....	...	.....
0000	2	1	2	1	2 1/2	1 1/4	2 1/2	1 1/4	.....	.....	...	.....
000	2 1/4	1 1/8	2 1/4	1 1/8	2 3/4	1 3/8	2 3/4	1 3/8	2 3/4	1 3/8	...	.....
00	2 3/4	1 3/8	2 1/2	1 1/4	3	1 1/2	3	1 1/2	3	1 1/2	...	.....
0	3	1 1/2	2 7/8	1 7/16	3 1/4	1 5/8	3	1 1/2	3 1/4	1 5/8	...	.....
1	3 1/4	1 5/8	3 1/4	1 5/8	3 1/2	1 3/4	3 1/2	1 3/4	3 1/4	1 5/8	...	.....
2	3 1/2	1 3/4	3 1/2	1 3/4	3 1/2	1 3/4	3 1/2	1 3/4	3 1/2	1 3/4	...	.....
3	4	2	3 1/2	1 3/4	4	2	4	2	4	2	...	.....
4	4 1/2	2 1/4	4	2	4	2	4	2	4	2	...	.....
5	5 1/2	2 3/4	4 1/2	2 1/4	4 1/2	2 1/4	4 1/2	2 1/4	4 1/2	2 1/4	...	.....
6	6	3	4 1/2	2 1/4	5	2 1/2	5	2 1/2	5	2 1/2	...	.....
7	6 1/2	3 1/4	5 1/2	2 3/4	5 1/2	2 3/4	5 1/2	2 3/4	5 1/2	2 3/4	...	.....
8	7	3 1/2	6	3	6	3	6	3	6	3	...	.....
9	8	4	6 1/2	3 1/4	6 1/2	3 1/4	6 1/2	3 1/4	6 1/2	3 1/4	...	.....
10	9	4 1/2	7	3 1/2	7	3 1/2	7	3 1/2	7	3 1/2	...	.....
11	11	5 1/2	8	4	8	4	8	4	8	4	...	.....
12	12	6	9	4 1/2	9	4 1/2	9	4 1/2	9	4 1/2	...	.....
13	14	7	10	5	10	5	10	5	10	5	...	.....
14	14	7	12	6	12	6	12	6	12	6	12	6
15	16	8	13	6 1/2	13	6 1/2	14	7	13	6 1/2	13	6 1/2
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	11 1/2	23	11 1/2	24	12	23	11 1/2	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	0.0270
0.187	0.197	0.222	4	0.0125	0.187	0.212	0.269	12	0.0285
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	0.250	0.284	0.348	14	0.0320
0.312	0.330	0.365	7	0.0175	0.312	0.348	0.412	14	0.0320
0.375	0.395	0.430	7	0.0175	0.375	0.414	0.478	14	0.0320
0.125	0.138	0.176	8	0.0190	0.437	0.478	0.542	14	0.0320
0.187	0.202	0.240	8	0.0190	0.500	0.545	0.609	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	0.420	15	0.0350
0.187	0.209	0.253	9	0.0220	0.375	0.417	0.487	15	0.0350
0.250	0.271	0.315	9	0.0220	0.437	0.480	0.550	15	0.0350
0.312	0.335	0.379	9	0.0220	0.500	0.547	0.617	15	0.0350
0.375	0.400	0.444	9	0.0220	0.562	0.611	0.681	15	0.0350

Table 15-3.—Data for Winding Piano Wire Tension Springs—Continued

Diam. of Man-drel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Num-ber of Piano Wire	Diam. of Piano Wire, Inches	Diam. of Man-drel, Inches	Inside Diam. of Spring, Inches	Outside Diam. of Spring, Inches	Num-ber 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	0.490
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	0.562	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	0.0380	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	0.691	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	0.0470	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

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, and so on. Basic quality assurance guidelines are usually set by type commanders such as SERVLANT, SUBLANT, SERVPAC, and SUBPAC. Until it is coordinated under one system, you will have to follow local guidelines. In most ships and at shore installations there are 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 a current 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 results on a Quality Assurance Form. On this form is recorded the name of the ship, the part manufactured, the print number used, the 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 Mechanical Instrument Repair and Calibration Shops (MIRCS) require labels and tags to indicate the status of calibration or testing. In marking labels and tags, MIRCS personnel should write 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 figure 15-35 and 15-36.

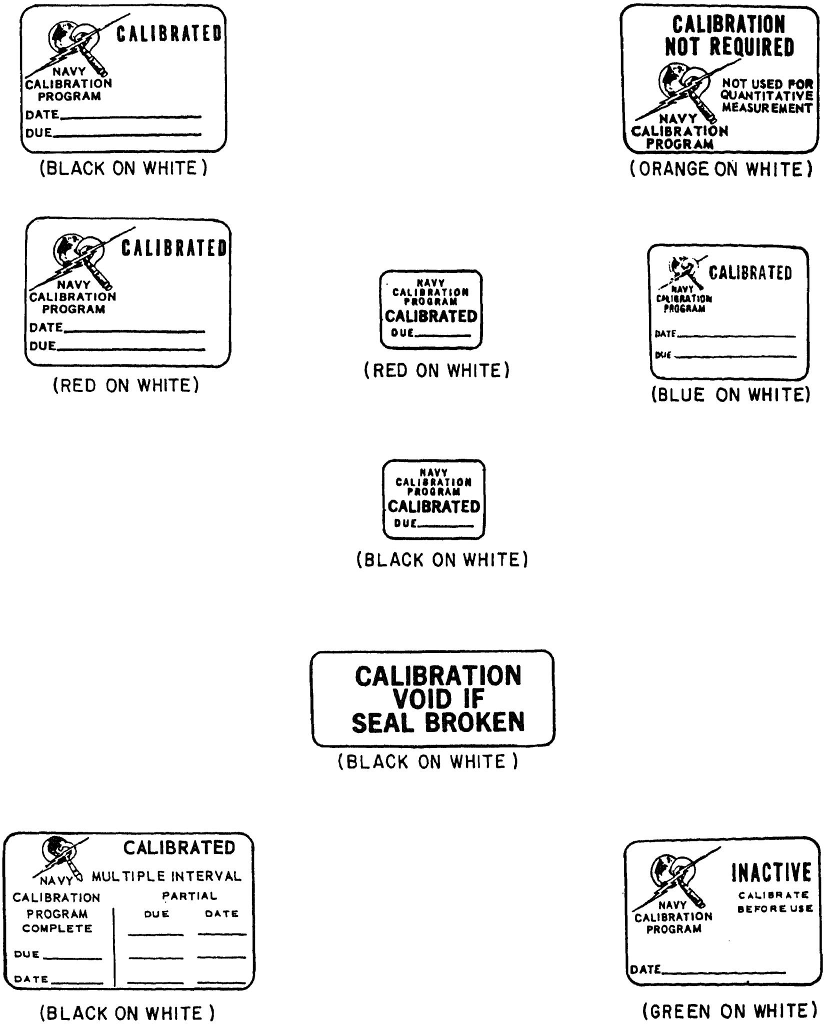
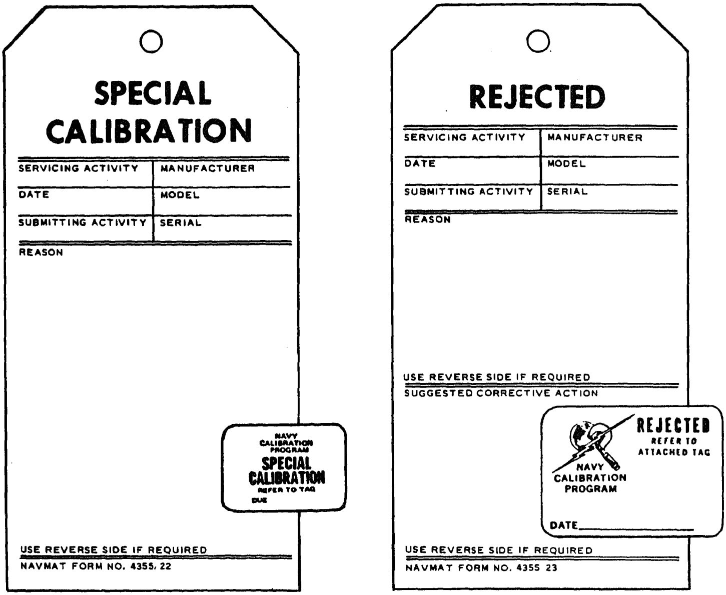


Figure 15-35.—Calibration labels.



(BLACK ON YELLOW) (BLACK ON RED)

Figure 15-36.—Labels and tags.

Calibrated

The CALIBRATED label is placed on each standard or piece of test and measuring equipment that has been checked against a standard of higher accuracy. Each check is made using approved Navy calibration procedures and checklists and is adjusted to meet (1) a predetermined specification or (2) a specified value of magnitude. 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 loss of accuracy are readily apparent during their normal use. 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 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.

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 it bears 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 it bears the inactive label.

APPENDIX I

TABULAR INFORMATION OF BENEFIT TO MACHINERY REPAIRMAN

Table AI-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	9/16	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 AI-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 AI-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 AI-4.—Formulas for Dimension, Area, and Volume

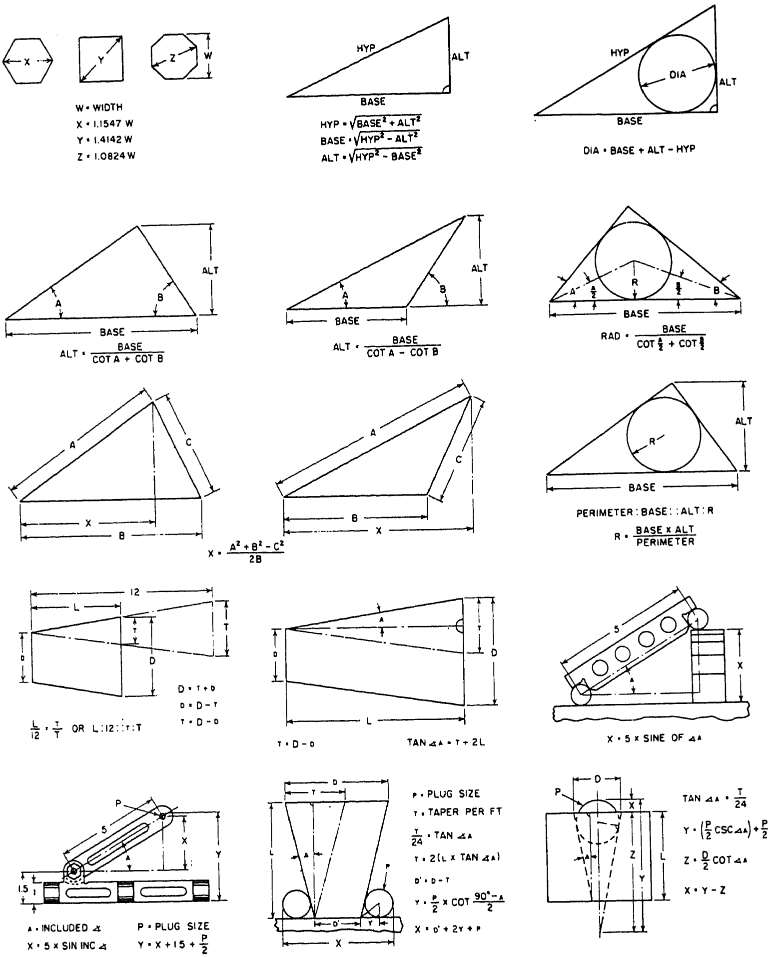
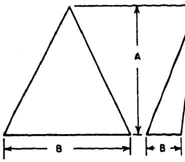


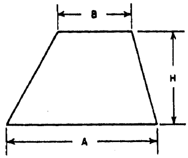
Table AI-4.—Formulas for Dimension, Area, and Volume—Continued

TRIANGLE



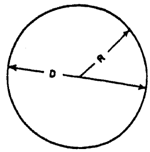
$AREA = \frac{1}{2} B \times A$

TRAPEZOID



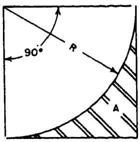
$AREA = \frac{1}{2}(A+B)H$

CIRCLE



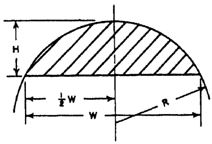
$AREA = 3.1416\ R^2$

FILLET



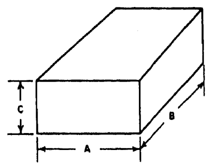
$A = R^2 - \frac{3.1416 R^2}{4}$

SEGMENT



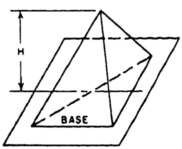
$R = \frac{(\frac{1}{2}W)^2 + H^2}{2H}$

RECTANGULAR PRISM



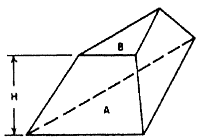
$VOLUME = ABC$

TRIANGULAR PYRAMID



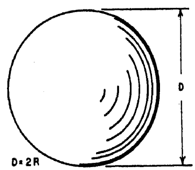
$VOLUME = \frac{AREA\ OF\ BASE \times H}{3}$

FRUSTUM OF PYRAMID



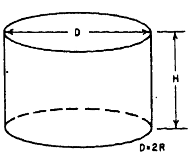
$VOLUME = \frac{H(A+B+\sqrt{AB})}{3}$

SPHERE



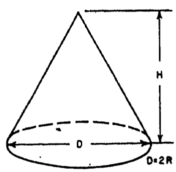
$VOLUME = \frac{4 \times 3.1416\ R^3}{3}$

CYLINDER



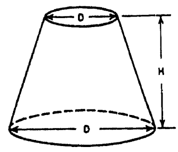
$VOLUME = 3.1416\ R^2 \times H$

CONE



$VOLUME = \frac{3.1416 R^2 \times H}{3}$

FRUSTUM OF CONE



$VOL = 0.2618\ H(D^2 + d^2 + Dd)$

Table AI-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 AI-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	fractional drills	decimal equiv.
80	...	.0135
79	...	.0145
	1/64	.0156
78	...	.0160
77	...	.0180
76	...	.0200
75	...	.0210
74	...	.0225
73	...	.0240
72	...	.0250
71	...	.0260
70	...	.0280
69	...	.0292
68	...	.0310
	1/32	.0312
67	...	.0320
66	...	.0330
65	...	.0350
64	...	.0360
63	...	.0370
62	...	.0380
61	...	.0390
60	...	.0400
59	...	.0410
58	...	.0420
57	...	.0430
56	...	.0465
	3/64	.0469
55	...	.0520
54	...	.0550
53	...	.0595
	1/16	.0625
52	...	.0635
51	...	.0670
50	...	.0700
49	...	.0730
48	...	.0760
	5/64	.0781
47	...	.0785
46	...	.0810
45	...	.0820
44	...	.0860
43	...	.0890

no. & letter drills	fractional drills	decimal equiv.
42	...	.0935
	3/32	.0937
41	...	.0960
40	...	.0980
39	...	.0995
38	...	.1015
37	...	.1040
36	...	.1065
	7/64	.1094
35	...	.1100
34	...	.1110
33	...	.1130
32	...	.1160
31	...	.1200
	1/8	.1250
30	...	.1285
29	...	.1360
28	...	.1405
	9/64	.1406
27	...	.1440
26	...	.1470
25	...	.1495
24	...	.1520
23	...	.1540
	5/32	.1562
22	...	.1570
21	...	.1590
20	...	.1610
19	...	.1660
18	...	.1695
	11/64	.1719
17	...	.1720
16	...	.1770
15	...	.1800
14	...	.1820
13	...	.1850
	3/16	.1875
12	...	.1890
11	...	.1910
10	...	.1935
9	...	.1960
8	...	.1990
7	...	.2010

no. & letter drills	fractional drills	decimal equiv.
	13/64	.2031
6	...	.2040
5	...	.2055
4	...	.2090
3	...	.2130
	7/32	.2187
2	...	.2210
1	...	.2280
A	...	.2340
	15/64	.2344
B	...	.2380
C	...	.2420
D	...	.2460
E	1/4	.2500
F	...	.2570
G	...	.2610
	17/64	.2656
H	...	.2660
I	...	.2720
J	...	.2770
K	...	.2810
	9/32	.2812
L	...	.2900
M	...	.2950
	19/64	.2969
N	...	.3020
	5/16	.3125
O	...	.3160
P	...	.3230
	21/64	.3281
Q	...	.3320
R	...	.3390
	11/32	.3437
S	...	.3480
T	...	.3580
	23/64	.3594
U	...	.3680
	3/8	.3750
V	...	.3770
W	...	.3860
	25/64	.3906
X	...	.3970
Y	...	.4040

no. & letter drills	fractional drills	decimal equiv.
	13/32	.4062
Z	...	.4130
	27/64	.4219
	7/16	.4375
	29/64	.4531
	15/32	.4687
	31/64	.4844
	1/2	.5000
	33/64	.5156
	17/32	.5312
	35/64	.5469
	9/16	.5625
	37/64	.5781
	19/32	.5937
	39/64	.6094
	5/8	.6250
	41/64	.6406
	21/32	.6562
	43/64	.6719
	11/16	.6875
	45/64	.7031
	23/32	.7187
	47/64	.7344
	3/4	.7500
	49/64	.7656
	25/32	.7812
	51/64	.7969
	13/16	.8125
	53/64	.8281
	27/32	.8437
	55/64	.8594
	7/8	.8750
	57/64	.8906
	29/32	.9062
	59/64	.9219
	15/16	.9375
	61/64	.9531
	31/32	.9687
	63/64	.9844
	1	1.0000

Table AI-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 AI-8.—Screw Thread and Tap Drill Sizes (American National)

screw size	threads per inch: NC coarse thread	threads per inch: NF fine thread	dimensions, inches: major diameter	dimensions, inches: pitch diameter	dimensions, inches: single depth of thread	dimensions, inches: minor diameter	tap drill 75% full thread: tap drill	tap drill 75% full thread: 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 AI-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
	¾	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

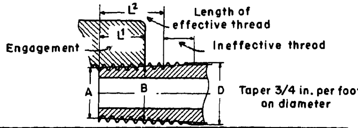
Table AI-9.—Full Thread Produced in Tapped Holes (Percentage)—Continued

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	70
	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

tap	tap drill	decimal tap drill	usual hole size	thread percentage
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 AI-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



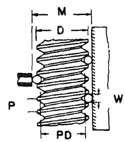
Size Inches	Threads per Inch	Pitch Diameter		Length		Pipe O.D. D Inches	Depth of Thread Inches	Tap Drills for Pipe Threads	
		A Inches	B Inches	L2 Inches	L1 Inches			Minor Diameter Small End of Pipe	Size Drill
1/8	27	.36351	.37476	.2639	.180	.405	.02963	.3339	R
1/4	18	.47739	.48989	.4018	.200	.540	.04444	.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	11 1/2	1.21363	1.23863	.6828	.400	1.315	.06957	1.1441	1 5/32
1 1/4	11 1/2	1.55713	1.58338	.7068	.420	1.660	.06957	1.4876	1 1/2
1 1/2	11 1/2	1.79609	1.82234	.7235	.420	1.900	.06957	1.7265	1 47/64
2	11 1/2	2.26902	2.29627	.7565	.436	2.375	.06957	2.1995	2 7/32
2 1/2	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	3 1/4
3 1/2	8	3.83750	3.88881	1.2500	.821	4.000	.10000	3.7375	3 3/4
4	8	4.33438	4.38712	1.3000	.844	4.500	.10000	4.2344	4 1/4

Table AI-11.—3-Wire Method—American National Std.

$$M = D - (1.5155 \times P) + (3 \times W)$$

$$PD = M + \frac{.86603}{\text{No. of thds. per inch}} (3 \times W)$$

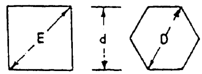
$$\text{To Check Angle } \frac{M_1 - M_2}{W_1 - W_2}$$



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
4 1/2	.222222	.128300	.200000	.124444
5	.200000	.115470	.180000	.112000
5 1/2	.181818	.104969	.163636	.101818
6	.166666	.096224	.149999	.093333
7	.142857	.082478	.128571	.080000
7 1/2	.133333	.076979	.120000	.074666
8	.125000	.072168	.112500	.070000
9	.111111	.064149	.100000	.062222
10	.100000	.057735	.090000	.056000
11	.090999	.052486	.081818	.050909
11 1/2	.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	.031111
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

Table AI-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 AI-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 its 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

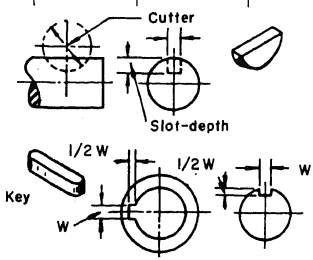
Circumference of a circle X .159155 = radius

Square root of the area of a circle X .56419 = radius

Square root of the area of a circle X 1.12838 = diameter

Table AI-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				

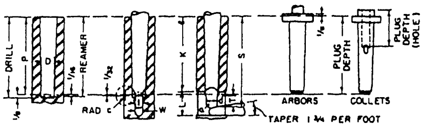


*The depth of a Woodruff keyway is measured from the edge of the slot.

Table AI-15.—Tapers Per Foot and Corresponding Angles

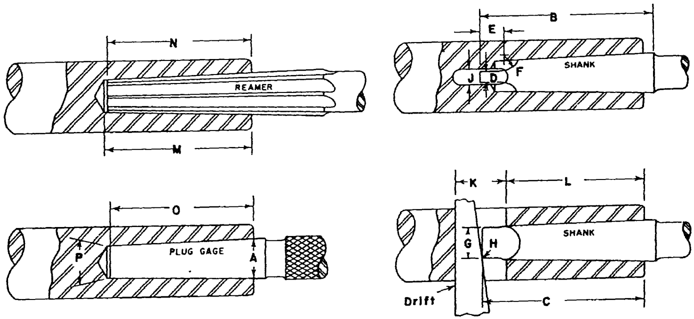
taper per foot	included angle			angle with center line			taper per foot	included angle			angle with center line			taper per foot	included angle			angle with center line		
	deg.	min.	sec.	deg.	min.	sec.		deg.	min.	sec.	deg.	min.	sec.		deg.	min.	sec.	deg.	min.	sec.
1/64	0	4	28	0	2	14	31/32	4	37	20	2	18	40	3 3/4	17	45	40	8	52	50
1/32	0	8	58	0	4	29	1	4	46	18	2	23	9	3 7/8	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	1 1/8	5	21	44	2	40	52	4 1/8	19	30	18	9	45	9
1/8	0	35	48	0	17	54	1 3/16	5	39	54	2	49	57	4 1/4	20	5	2	10	2	31
5/32	0	44	44	0	22	22	1 1/4	5	57	48	2	58	54	4 3/8	20	39	44	10	19	52
3/16	0	53	44	0	26	52	1 5/16	6	15	38	3	7	49	4 1/2	21	14	2	10	37	1
7/32	1	2	34	0	31	17	1 3/8	6	33	26	3	16	43	4 5/8	21	48	54	10	54	27
1/4	1	11	36	0	35	48	1 7/16	6	51	20	3	25	40	4 3/4	22	23	22	11	11	41
9/32	1	20	30	0	40	15	1 1/2	7	9	10	3	34	35	4 7/8	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	1 5/8	7	44	48	3	52	24	5 1/8	24	6	28	12	3	14
3/8	1	47	24	0	53	42	1 11/16	8	2	38	4	1	19	5 1/4	24	40	42	12	20	21
13/32	1	56	24	0	58	12	1 3/4	8	20	26	4	10	13	5 3/8	25	14	48	12	37	24
7/16	2	5	18	1	2	39	1 13/16	8	38	16	4	19	8	5 1/2	25	48	48	12	54	24
15/32	2	14	16	1	7	8	1 7/8	8	56	2	4	28	1	5 5/8	26	22	52	13	11	26
1/2	2	23	10	1	11	35	1 15/16	9	13	50	4	36	55	5 3/4	26	56	46	13	28	23
17/32	2	32	4	1	16	2	2	9	31	36	4	45	48	5 7/8	27	30	34	13	45	17
9/16	2	41	4	1	20	32	2 1/8	10	7	10	5	3	35	6	28	4	2	14	2	1
19/32	2	50	2	1	25	1	2 1/4	10	42	42	5	21	21	6 1/8	28	37	58	14	18	59
5/8	2	59	2	1	29	31	2 3/8	11	18	10	5	39	5	6 1/4	29	11	34	14	35	47
21/32	3	7	56	1	33	58	2 1/2	11	53	36	5	56	48	6 3/8	29	45	18	14	52	39
11/16	3	16	54	1	38	27	2 5/8	12	29	2	6	14	31	6 1/2	30	18	26	15	9	13
23/32	3	25	50	1	42	55	2 3/4	13	4	24	6	32	12	6 5/8	30	51	48	15	25	54
3/4	3	34	44	1	47	22	2 7/8	13	39	42	6	49	51	6 3/4	31	25	2	15	42	31
25/32	3	43	44	1	51	52	3	14	15	0	7	7	30	6 7/8	31	58	10	15	59	5
13/16	3	52	38	1	56	19	3 1/8	14	50	14	7	25	7	7	32	31	12	16	15	36
27/32	4	1	36	2	0	48	3 1/4	15	25	24	7	42	42	7 1/8	33	4	8	16	32	4
7/8	4	10	32	2	5	16	3 3/8	16	0	34	8	0	17	7 1/4	33	36	40	16	48	20
29/32	4	19	34	2	9	47	3 1/2	16	35	40	8	17	50	7 3/8	34	9	50	17	4	55
15/16	4	28	24	2	14	12	3 5/8	17	10	40	8	35	20							

Table AI-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 3/16	3/8	0.135	3/16	0.170	1/8	3/16	0.030	0.003
2	0.50200	0.25000	1 3/16			1 11/64	1 1/2	1/2	0.166	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	8.040	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 13/64	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	0.50160	0.45000		1 3/4		1 11/16	2 3/16	3/4	0.260	3/8	0.420	1/4	5/16	0.060	0.003
					2	1 15/16	2 7/16	3/4	0.260	3/8	0.420	1/4	5/16	0.060	0.003
			2 1/8			2 1/16	2 9/16	3/4	0.260	3/8	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	5/16	0.060	0.005
7	0.50147	0.60000			2 1/2	2 13/32	3 1/32	15/16	0.322	15/32	0.560	5/16	3/8	0.070	0.005
			2 7/8			2 15/32	3 13/32	15/16	0.322	15/32	0.560	5/16	3/8	0.070	0.005
				3		2 29/32	3 17/32	15/16	0.322	15/32	0.560	5/16	3/8	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	3/8	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 17/32	6 13/32	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		6 19/32	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		6 15/16	7 15/16	1 1/2	0.510	3/4	1.460	1/2	1/2	0.150	0.005
					6 1/4										
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												

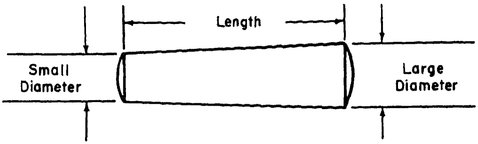
Table AI-17.—American Standard Tapers (Morse)



taper no.	diameter: plug at small end, P	diameter: gage line, A	shank: whole length, B	shank: depth, C	drilled hole depth, M	reamed hole depth, N	stan. plug depth, O	tang: thick-ness, D	tang: length, E	tang: radius, F	tang: dia., G	tang: radius, H	tang slot: width, J	tang slot: 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 AI-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	¼
LENGTH	DIAMETER OF SMALL END OF PIN AND DRILL SIZE																		LENGTH
¼	0.0573	0.0728	......	......	......	......	......	......	......	......	......	......	......	......	......	......	......	......	¼
	54	50																	
⅜	0.0547	0.0702	0.0862	......	......	......	......	......	......	......	......	......	......	......	......	......	......	......	⅜
	56	51	45																
½	0.0521	0.0676	0.0838	0.0986	0.1146	0.1306	0.1458	0.1616	......	......	......	......	......	......	......	......	......	......	½
	56	52	46	41	34	30	9/64	5/32											
⅝	0.0495	0.0650	0.0810	0.0960	0.1120	0.1280	0.1430	0.1590	......	......	......	......	......	......	......	......	......	......	⅝
	58	52	5/64	3/32	7/64	1/8	9/64	23											
¾	0.0469	0.0624	0.0784	0.0934	0.1084	0.1254	0.1404	0.1564	0.1774	0.2034	0.2344	......	......	......	......	......	......	......	¾
	58	53	48	43	36	31	29	24	11/64	8	1								
⅞	......	0.0598	0.0758	0.0908	0.1068	0.1228	0.1378	0.1538	0.1748	0.2008	0.2318	......	......	......	......	......	......	......	⅞
	......	54	49	43	37	31	29	25	18	9	1								
1	......	0.0572	0.0732	0.0882	0.1042	0.1202	0.1352	0.1512	0.1722	0.1982	0.2292	0.2682	0.3202	......	......	......	......	......	1
		54	50	44	38	32	30	28	19	10	2	G	O						
1⅛	......	......	......	0.0856	0.1016	0.1176	0.1326	0.1486	0.1696	0.1956	0.2266	0.2656	0.3176	......	......	......	......	......	1⅛
				45	39	33	30	27	18	11	2	G	5/16						
1¼	......	......	......	0.0830	0.0990	0.1150	0.1300	0.1460	0.1670	0.1930	0.2240	0.2630	0.3150	0.3830	......	......	......	......	1¼
				48	41	33	1/8	9/64	20	3/16	7/32	F	N	3/8					
1⅜	......	......	......	......	0.0964	0.1124	0.1274	0.1434	0.1644	0.1904	0.2214	0.2604	0.3124	0.3804	......	......	......	......	1⅜
					3/32	7/64	1/8	9/64	20	3/16	3	F	N	3/8					
1½	......	......	......	......	0.0938	0.1098	0.1248	0.1408	0.1618	0.1878	0.2188	0.2578	0.3098	0.3778	0.4608	......	......	......	1½
					43	36	31	28	5/32	14	3	¼	N	U	29/64				
1¾	......	......	......	......	......	0.1045	0.1195	0.1355	0.1565	0.1825	0.2135	0.2525	0.3045	0.3725	0.4555	......	......	......	1¾
						36	32	30	24	16	4	D	19/64	U	7/16				
2	......	......	......	......	......	0.0993	0.1143	0.1303	0.1513	0.1773	0.2083	0.2473	0.2993	0.3673	0.4503	0.5494	......	......	2
						41	34	1/8	26	11/64	13/64	C	M	23/64	7/16	17/32	......	......	
2¼	......	......	......	......	......	......	......	0.1251	0.1461	0.1721	0.2031	0.2421	0.2941	0.3621	0.4451	0.5442	......	......	2¼
								31	27	19	8	B	L	T	7/16	17/32			
2½	......	......	......	......	......	......	......	0.1196	0.1406	0.1666	0.1976	0.2366	0.2886	0.3566	0.4396	0.5390	0.6540	......	2½
								32	29	20	10	15/64	9/32	S	27/64	17/32	41/64		
2¾	......	......	......	......	......	......	......	......	0.1357	0.1617	0.1927	0.2317	0.2837	0.3517	0.4347	0.5338	0.6488	......	2¾
									30	5/32	3/16	1	J	11/32	27/64	33/64	41/64		

Table AI-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	0.3485 R	0.4295 27/64	0.5285 33/64	0.6435 5/8	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 5/8	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 5/8	0.7871 25/32	3½
3¾	......	......	......	......	......	......	......	......	......	......	......	......	0.2629 F	0.3309 21/64	0.4139 13/32	0.5129 ½	0.6278 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 D	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	......	......	......	......	......	......	......	......	......	......	......	......	......	......	......	......	0.5758 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 AI-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.

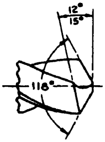


Fig.I-19-1

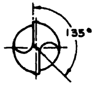


Fig.I-19-2

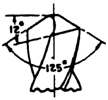


Fig.I-19-3

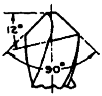


Fig.I-19-4

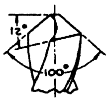


Fig.I-19-5

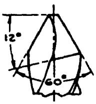


Fig.I-19-6

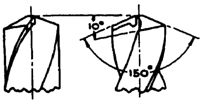


Fig.I-19-7

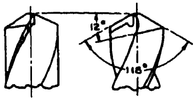


Fig.I-19-8

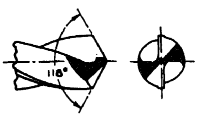


Fig.I-19-9

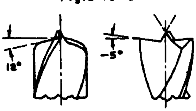


Fig.I-19-10

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″ diameter make angle of bit point to suit work

Table AI-20.—Allowances for Fit

(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

Table AI-21.—Plain and Differential Indexing

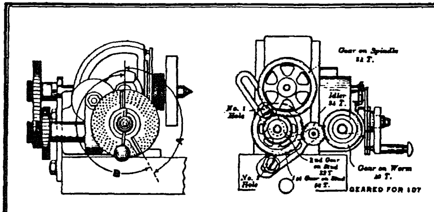


TABLE 2 to 50

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

TABLE 51 to 92.

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".

Table AI-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	80,000	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 AI-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 AI-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	Cast Iron	Stainless Steels Ingot Iron Wrought Iron	Tool Steels High-speed Steels	Aluminum and Alloys Brasses and Bronzes Magnesium and Alloys Zinc	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.	$OD = (NT + 2)\ 0.3183\ CP$
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) and 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.	$DP = \frac{NT + 2}{OD}$
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.	$NT = PD\ DP$

Having	To Get	Rule	Formula
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. AIII-1)

   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 1-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.

   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.

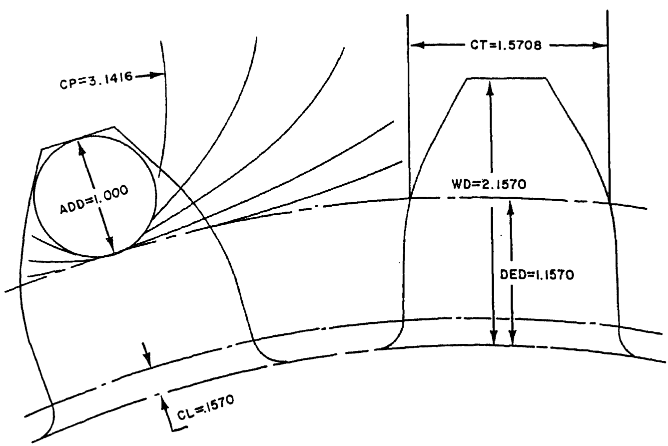


Figure AIII-1.—Tooth elements on a #1 diametral pitch gear.

e. Dedendum (DED)—1.15708

(1) The sum of an addendum plus a clearance.

(2)

$$\begin{array}{rl} 1.000 & \text{- ADD} \\ +\ \underline{0.1570} & \text{- CL} \\ 1.1570 & \text{- DED} \end{array}$$

f. Working Depth (WKD)—2.000

(1) The sum of two addendums.

(2)

$$\begin{array}{rl} 1.000 & \text{- ADD} \\ +\ \underline{1.000} & \text{- ADD} \\ 2.000 & \text{- WKD} \end{array}$$

g. Whole Depth (WD)—2.15708

(1) The sum of an addendum and a dedendum.

(2)

$$\begin{array}{rl} 1.0000 & \text{- ADD} \\ +\ \underline{1.1570} & \text{- DED} \\ 2.1570 & \text{- WD} \end{array}$$

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.

$$t_c = PD \sin \frac{90^0}{N}$$

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 × 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 × NT = PD

(b) ADD × (NT + 2) = OD

(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 AIII-2 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	

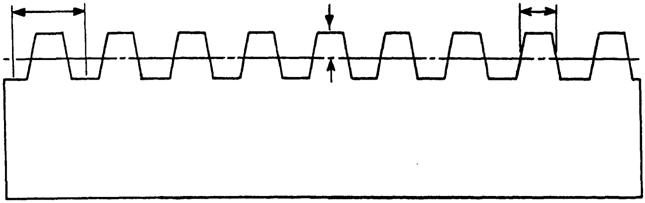


Figure AIII-2.—Linear pitch.

(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. $CL = \frac{0.15708}{DP}$

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 $PD = ADD \times NT$

11. $OD = ADD \times (NT + 2)$

APPENDIX IV

GLOSSARY

When you enter a new occupation, you must learn the vocabulary of the trade so that you understand your fellow workers 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.

ABRASIVE.—A hard, tough substance which has many sharp edges.

AISI.—American Iron and Steel Institute.

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.

BENCH MOLDING.—The process of making small molds on a bench.

BEND ALLOWANCE.—An additional amount of metal used in a bend in metal fabrication.

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 gasses.

BOND.—Appropriate substance used to hold grains together in grinding wheels.

BORING BAR.—A tool used for boring, counterboring, reboring, facing, grooving, and so forth, where true alignment is of primary importance.

BRINELL.—A type of hardness test.

BRITTLENESS.—The 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 breaking.

EXTRACTOR.—Tool used in removal of broken taps.

FABRICATE.—To shape, assemble, and secure in place component parts in order to form a complete device.

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 the surface of a casting to allow for machine finishing.

FINISH MARKS.—Marks used to indicate the degree of smoothness of finish to be achieved on surfaces to be machined.

GRAIN.—The cutting particles of a grinding wheel.

HARDNESS.—The ability of a material to resist penetration.

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 room 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 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.—A straight line that meets another straight line at a 90° angle. Also a vertical line extending through the outline 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.—The 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 revolving in it a cylindrical, slightly tapered tool with cutting edges running along its sides.

RECHUCKING.—Reversing of a piece of work on 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 graduations.

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 whose working edge is 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 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 finish machining of parts.

ZINC.—An alloy used widely in die casting.

INDEX

C

D

E

F

G

H

I

N

O

P

S

T

U

V

W

✩ U.S. GOVERNMENT PRINTING OFFICE: 1990-731-066/20043