DOCUMENT RESUME

ED 265 360		CE 043 317
TITLE	Apprentice Machinist (AFSC 53130), Volumes 1-4, and Change Supplement (AFSC 42730).	
INSTITUTION	Air Univ., Gunter AFS, Ala. Extension Course Inst.	
PUB DATE	81	
NOTE	491p.	
PUB TYPE	Guides - Classroom Use - Materials (For Learner) (051)	
EDRS PRICE	MF02/PC20 Plus Postage.	
DESCRIPTORS	*Apprenticeships; Behavioral Objectives; Correspondence Study; Distance Education; *Equipment Utilization; Learning Activities; *Machine Tools; *Machinists; Metal Working; Military Personnel; Military Training; Postsecondary Education; Shop Curriculum; *Trade and Industrial Education	
IDENTIFIERS	Air Force	

ABSTRACT

This four-volume student learning package is designed for use by Air Force personnel enrolled in a self-study extension course for apprentice machinists. The package consists of four volumes of instructional text and four workbooks. Covered in the individual volumes are machine shop fundamentals, lathe work, shaper and contour machine work, and milling and grinding machine work. Each volume in the set contains a series of lessons and a bibliography. Each workbook includes a study reference guide, chapter review exercises and their answers, and a volume review exercise. A change supplement, containing revised pages for volume 1 and pen and ink changes for volumes 1-4, is also provided. (MN)

ED265360

APPRENTICE MACHINIST

(AFSC 53130)

EXTENSION COURSE INSTITUTE

AIR UNIVERSITY

CE 043317

ECI COURSE MATERIALS SHIPPING LIST

COURSE NO	COURSE TITLE	EFFECTIVE DATE OF SHIPPING LIST
53130	APPRENTICE MACHINIST (AFSC 42730)	17 Nov 81

INSTRUCTIONS: The following materials are needed to complete this course. Check this list immediately upon receiving your course package, and if any materials are missing or incorrect (numbers don't match) notify ECI immediately. Use the ECI Form 17 for this purpose, and be sure to include your identification number, address, course and volume number, and VRE form designation (if a VRE is [illegible]). Send all correspondence separately from your answer sheet.

ITEM	TYPE	DESIGNATION OR TITLE	INVENTORY CONTROL NUMBER	VRE ANSWER SHEET IDENTIFICATION
1	ENV	ECI RETURN ENVELOPES (4 EACH)		
2	SHEET	YOUR KEY TO CAREER DEVELOPMENT	M-140	
3	VOL	VOL 1, Machine Shop Fundamentals	53130 01 0771	
4	VWB	VOLUME WORKBOOK (VOL 1)	53130 01 21	53130 01 21
5	VOL	VOL 2, Lathe Work	53130 02 0971	
6	VWB	VOLUME WORKBOOK (VOL 2)	53130 02 21	53130 02 21
7	VOL	VOL 3, Shaper and Contour Machine Work	53130 03 1071	
8	VWB	VOLUME WORKBOOK (VOL 3)	53130 03 21	53130 03 21
9	VOL	VOL 4, Milling and Grinding Machine Work	53130 04 1171	
10	VWB	VOLUME WORKBOOK (VOL 4)	53130 04 21	53130 04 21
11	SUPP	CHANGE SUPPLEMENT (VOLs 1,2,3 & 4)	53130 00 S01 7709	

NOTES: DIRECT ANY QUESTIONS OR COMMENTS RELATING TO ACCURACY OR CURRENCY OF TEXTUAL MATERIALS TO AUTOVON 862-2385.

YOU ARE NOT REQUIRED TO POST ANY CHANGES LISTED ON THIS SHIPPING LIST WHICH CORRECT TYPOGRAPHICAL ERRORS, UNLESS SUCH ERRORS CHANGE OR OTHERWISE AFFECT THE MEANING OF THE MATERIAL.

(SEE REVERSE SIDE FOR ADDITIONAL INSTRUCTIONS)

LIST OF CHANGES

COURSE NO. 53130	CAREER FIELDS, POLICIES, PROCEDURES AND EQUIPMENT CHANGE. ALSO ERRORS OCCASIONALLY GET INTO PRINT. THE FOLLOWING ITEMS UPDATE AND CORRECT YOUR COURSE MATERIALS. PLEASE MAKE THE INDICATED CHANGES.
EFFECTIVE DATE OF SHIPPING LIST 17 Nov 81	

1. CHANGES FOR THE VOLUME WORKBOOK: VOLUME 1

a. Page 2, Chapter Review Exercises, question 2: Change "metalworking" to "fabrication." Question 3: Change "53130" to "42730."

b. Page 18, Chapter Review Exercises, answer 3: Change to read "42 Aircraft Systems Maintenance career field; XX7 Fabrication career field subdivision; XXX3 skill level (apprentice); XXXX0 machinist career ladder."

c. Page 28, question 7: In the stem of the question, change "AFSC 53130" to "AFSC 42730" and change "AFSC 53150" to "AFSC 42750."

d. The following questions are no longer scored and need not be answered: 1, 2, 3, 4, 5 and 32.

2. CHANGES FOR THE VOLUME WORKBOOK: VOLUME 2

a. Page 23, question 17: In the stem of the question, delete "hexagonal."

b. Pages 31 and 32 of the workbook were printed in reverse. Page 31 should be 32 and page 32 should be 31. The questions are numbered correctly.

c. Question 86 is no longer scored and need not be answered.

3. CHANGE FOR THE VOLUME WORKBOOK: VOLUME 3

The following questions are no longer scored and need not be answered: 14 and 43.

NOTE: Change the currency date on all volumes to "March 1981."

53130 01 0771

CDC 53130

APPRENTICE MACHINIST

(AFSC 53130)

Volume 1

Machine Shop Fundamentals

Extension Course Institute

Air University

Preface

THE OBJECTIVE of this course is to provide the knowledge that you need in order to progress to the apprentice machinist skill level. The emphasis will be on broadening your technical knowledge and preparing you to perform the duties of an apprentice machinist.

Note the chapter titles on the contents page for Volume 1. Chapter 1 covers general machine shop information; Chapter 2, metals and heat treatment; Chapter 3, preparatory shop work; and Chapter 4, introductory machine and bench work. Now, leaf quickly through the pages of each chapter and note the numbered headings; this will help you to understand the scope of this volume; you will note that it covers nearly all the basic machinist knowledges except those that pertain to the more complex machine tools and machining operations.

Code numbers appearing on the figures and charts are for preparing agency identification only.

If you have questions on the accuracy or currency of the subject matter of this text, or recommendations for its improvement, send them to Tech Tng Cen (TSOC), Chanute AFB IL 61868.

If you have questions on course enrollment or administration, or on any of ECI's instructional aids (Your Key to Career Development, Study Reference Guides, Chapter Review Exercises, Volume Review Exercise, and Course Examination), consult your education officer, training officer, or NCO, as appropriate. If he can't answer your questions, send them to ECI, Gunter AFB, AL 36114, preferably on ECI Form 17, Student Request for Assistance.

This volume is valued at 24 hours (8 points).

Material in this volume is technically accurate, adequate, and current as of March 1971.

Contents

		Page
	Preface	*iii*
Chapter		
1	General Machine Shop Information	1
2	Metals and Heat Treatment	19
3	Preparatory Shop Wopk	30
4	Introductory Machine and Bench Work	67
	Bibliography	91

General Machine Shop Information

ONE OF the great needs of the Air Force today is for well-trained technicians. Opportunities are almost unlimited for workmen who are skillful with their hands and who are trained to think about their work, to diagnose troubles, and to suggest improvements. And, of course, no one can hope for success in any line of work unless he is willing to study and improve his skills.

2. Most of the skilled machinists and machine shop technicians in the Air Force are the products of Air Force apprentice training programs. This is your opportunity to lay the groundwork to become a highly qualified craftsman and to get started on a fine career. The rewards for the skilled machinist can be very gratifying, both in advancement and in pride of workmanship.

3. This volume is designed to give you the background information you need to know to get started in machine shop work. It covers general machine shop information, metals and heat treatment, preparatory shop work, and introductory machine and bench work.

1. The Metalworking Career Field

1-1. Everyone in the Air Force, regardless of his rank or grade, has a job to do. You are probably wondering what you will be doing throughout the remainder of your enlistment. It is only natural that you are concerned about what the future holds for you in your career field. The Air Force has a great need for skilled airmen in hundreds of different jobs. Your job assignment depends upon Air Force needs and on your ability to learn and do certain kinds of work.

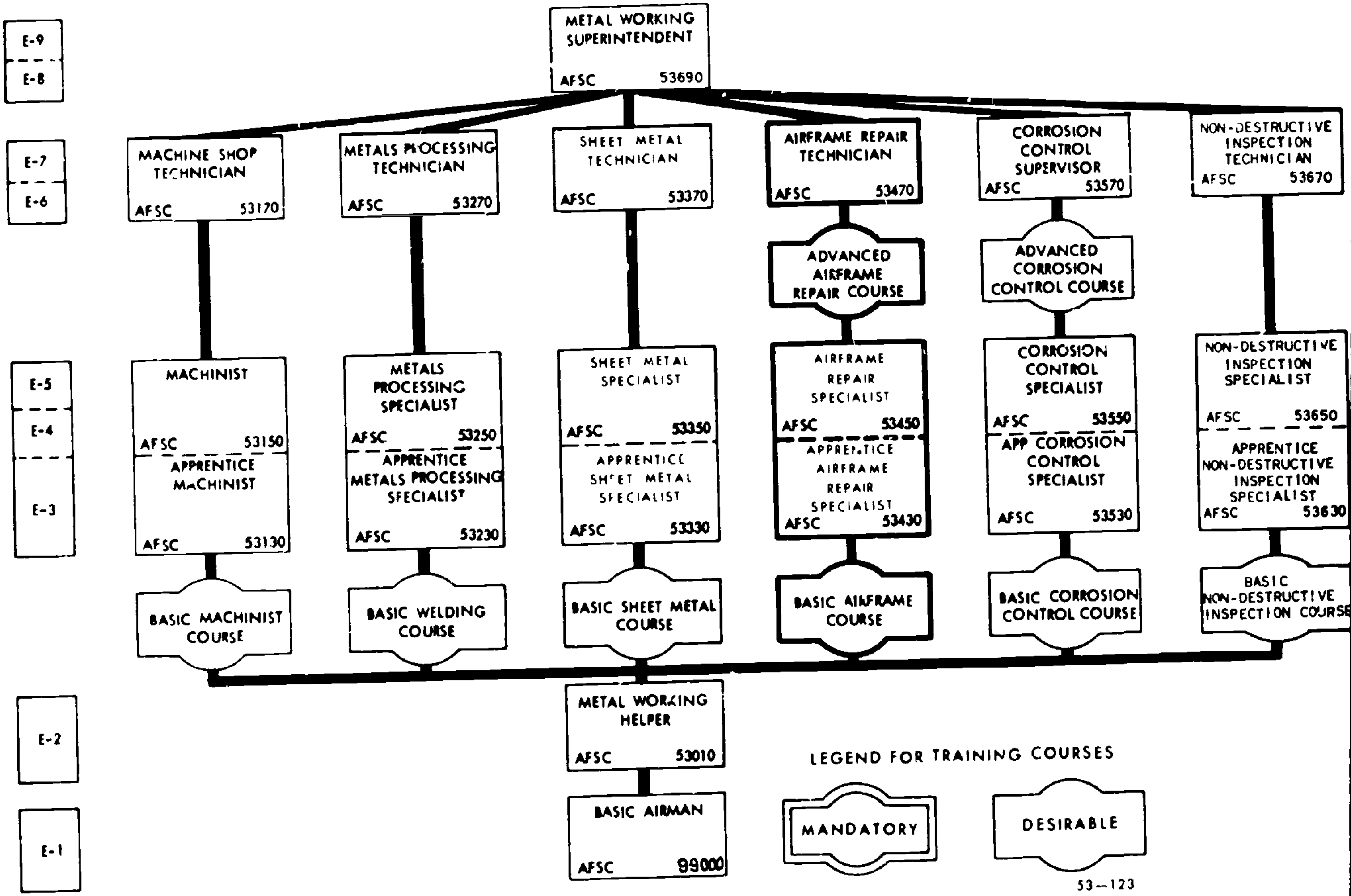
1-2. **Air Force Career Field System.** The Air Force has a system for grouping related jobs into common work areas or career fields. Each job grouping or career field requires the same general qualifications and the same sort of ability to learn and perform related jobs. The Airman Classification Structure Charts 39–1A and 39–1B are Air Force visual aid charts (AFVA) which show all career fields and their number designations These charts also show the skill levels and the equivalent grade spread for each job specialty. You can usually find one of these charts posted in the shop area where you work. As a result of your background and the job classification interviews and the tests which you took in basic training, you were assigned to the metalworking career field.

1-3. The metalworking career field includes fabricating, shaping, cutting, and joining metals and repairing metal parts; aircraft structural repair; and metal heat treating, welding, plating, and machining. The installation, modification, and formation of plastic articles are also a part of this career field. It also includes corrosion control for missile, aircraft, and support systems. Nondestructive inspection of metal parts, components, and pressurized systems are also included in this field. Career fields are divided into ladders, as shown in chart 1. The metalworking career field is divided into six ladders: Machinist, Metals Processing, Sheet Metal Repair, Airframe Repair, Corrosion Control, and Nondestructive Inspection. Each career ladder is divided into job specialties; for example, the machine shop ladder is divided into technician and machinist specialties. The apprentice machinist is a part of the machinist specialty.

1-4. **Air Force Specialty.** An Air Force Specialty (AFS) is identified by title and code. A five-digit code number is used to make up an Air Force Specialty Code (AFSC) which identifies an AFS. The first two digits identify the career field. The third digit, when other than 0, identifies the career field ladder. The fourth digit shows the skill level. The fifth digit, in combination with the other four, identifies the specific AFS. Since you are in training to become an apprentice machinist, let's show the breakdown of the apprentice machinist AFSC 53130. The breakdown of this AFSC is as follows:

53	Career field	Metalworking
1	Subdivision or ladder	Machinist
3	Skill level	Semiskilled
0	Specific AFS	General Machinist
53130	Complete AFSC of Apprentice Machinist	

CHART 1
METALWORKING CAREER FIELD



1-5. Study the metalworking career field chart shown in chart 1. Start at the bottom of the chart. You will note that, as a basic airman, you had the AFSC 99000. This was your AFSC during basic training because you had not yet been assigned to a career field. Upon completion of basic training, you were assigned to the metalworking career field as a helper and awarded the AFSC of 53010. This AFSC meant that you were qualified to enter into training in any of the ladders in the career field. You could have received your training by taking the basic machinist course at Chanute Air Force Base, Illinois. However, you were given a direct duty assignment of AFSC 53130 and you entered on-the-job training to become an apprentice machinist.

1-6. When you have satisfactorily completed your training you will be awarded the primary AFSC 53130, apprentice machinist. Your primary AFSC is the specialty in which you are most highly qualified. Your duty AFSC is the AFSC to which you are assigned. During training your duty AFSC is normally one skill level higher than your primary AFSC. For example, your primary AFSC is 53010, but since you are in training, your duty AFSC is 53130. By studying the chart you will note that there are five skill levels in each ladder of the metalworking career field. You will also note, in the far left column of the chart, the grade spread for each skill level. The top level, or metalworking superintendent AFSC 53690, can be an input from any of the ladders. You progress up the ladder from one skill level to the next, usually, through on-the-job training. We will discuss this in the following section.

1-7. **AFM 39–1, Airman Classification Manual.** Air Force Manual 39–1 contains the authorized Air Force Specialties and Air Force Specialty Codes. These AFSs and AFSCs are used in the classification of airmen positions and personnel. AFSs provide job standards for procurement, training, education, utilization, and development of airmen. AFSCs give us a systematic means for identifying training and position requirements.

1-8. Air Force Specialty descriptions are composed of the following parts:

a. Heading. The heading consists of the specialty code, specialty title, and effective date.

b. Summary. The summary is a concise statement of the content of the AFSC.

c. Duties and responsibilities. This part describes the scope of the job specialty in terms of duties and responsibilities.

d. Qualifications. This part gives the job qualification standards for adequate performance in the AFS. Standards are either mandatory or desirable. They are stated in five parts: knowledge, education, experience, training, and other—such as physical requirements, security clearance, certification, etc.

e. Specialty data. This part establishes the grade spread for the AFS. The grade spread is used for authorizing manning positions. This also designates jobs closely related to the AFS—for use in your initial selection or in your return to civilian life.

f. Specialty shredouts. This part designates authorized shredouts to be used with the AFS and the letter suffix identifier for use with the AFSC. It also gives the portion of the specialty to which it is related.

1-9. **Specialty Descriptions (AFSC 53130/50/70/690).** Air Force Specialty descriptions in AFM 39–1 describe the duties and responsibilities of the job specialty. You should be especially interested in the duties of your AFSC and in those of other specialties in your career ladder. We will not attempt to cover all the duties and responsibilities of each AFSC in your career ladder. You can read the complete descriptions in AFM 39–1. Therefore we will discuss only the major duties and responsibilities of each AFSC in your ladder.

1-10. *Machinist (AFSC 53130/50).* The specialty summary for a machinist states that he "operates metalworking machines in fabrication, rework, and repair of metal parts." You will note that this specialty description serves both AFSC 53130 and AFSC 53150, since both the 3- and 5-skill-level machinist perform essentially the same duties. The 5-skill-level machinist has more knowledge of his work and is more skillful than is the 3-level machinist. Also, the 5-level machinist has some duties in advanced machine shop work and supervision and training that the 3-level machinist does not have. The major duties and responsibilities of the machinist are as follows:

a. Manufactures and reworks machined parts.

b. Assembles and fits and machines parts.

c. Maintains hand and machine tools.

d. Supervises machine shop personnel. (At this time you are not concerned with this area of work. It will be discussed in CDC 53150.)

1-11. *Machine shop technician (AFSC 53170).* The specialty summary of the machine shop technician states that he "designs and machines precision tools, parts, and assemblies; inspects machine work; and supervises machine shop activities." The major duties and responsibilities of the machine shop technician are as follows:

a. Troubleshoots difficult metal machining, design, and production problems.

b. Inspects in-progress and completed machine work for quality of workmanship and serviceability.

c. Instructs in metals machining techniques and in maintenance of machinery and equipment.

1-12. *Metalworking superintendent (AFSC 53690).* The specialty summary of the metalworking superintendent states that he "superintends activities engaged in testing, fabricating, repairing, and machining metals and metal products; corrosion control in missile, aircraft, and support system equipment; and nondestructive inspection of aerospace material parts, components, and pressurized systems." The major duties and responsibilities of the metalworking superintendent are as follows:

a. Plans and organizes metalworking activities.

b. Directs metalworking activities.

c. Establishes and conducts on-the-job training for metalworking personnel.

d. Inspects and evaluates metalworking activities.

e. Performs technical metalworking activities.

1-13. **Coordinating with Other Shops.** A field maintenance organization is made up of people working in various career fields. The machine shop and the other metalworking shops are usually housed in the same building or central area. This arrangement permits close coordination of work with other maintenance activities. Since a large part of all machine shop work is in support of other maintenance activities, the machine shop must coordinate its work with other maintenance shops. As an example, an engine mechanic may have a broken stud which you are required to remove. The hydraulic or instrument mechanic may need a special tool or a part which you are required to make. Materials and supplies have to be obtained through the supply section. Close coordination and cooperation between maintenance activities are needed if you are to get the job done

2. On-the-Job Training

2-1. What is on-the job training (OJT), what is its purpose, and how does it work? Since the Air Force has to train thousands of airmen in various career fields and AFSs, it is neither practical nor economical to send all airmen to formal training schools for their 3-skill-level AFSC. In many specialties the Air Force can benefit from the work the students do while they are learning. When they attend a formal training school, some productive maintenance work is lost while they are in training. Both formal schooling and OJT have certain advantages, depending upon the type of job.

2-2. **Dual Channel Concept of OJT.** The dual channel concept of OJT consists of two parts: career development and job proficiency development. The first part consists of studying a career development course (CDC). By studying the CDC the trainee learns the information he needs if he is to do the various duties and tasks of his AFSC. In the second part he develops his skill by using equipment and by doing the jobs required in his AFSC. He does both parts at the same time He must satisfactorily complete both parts before he is to be upgraded to the AFSC for which he is training.

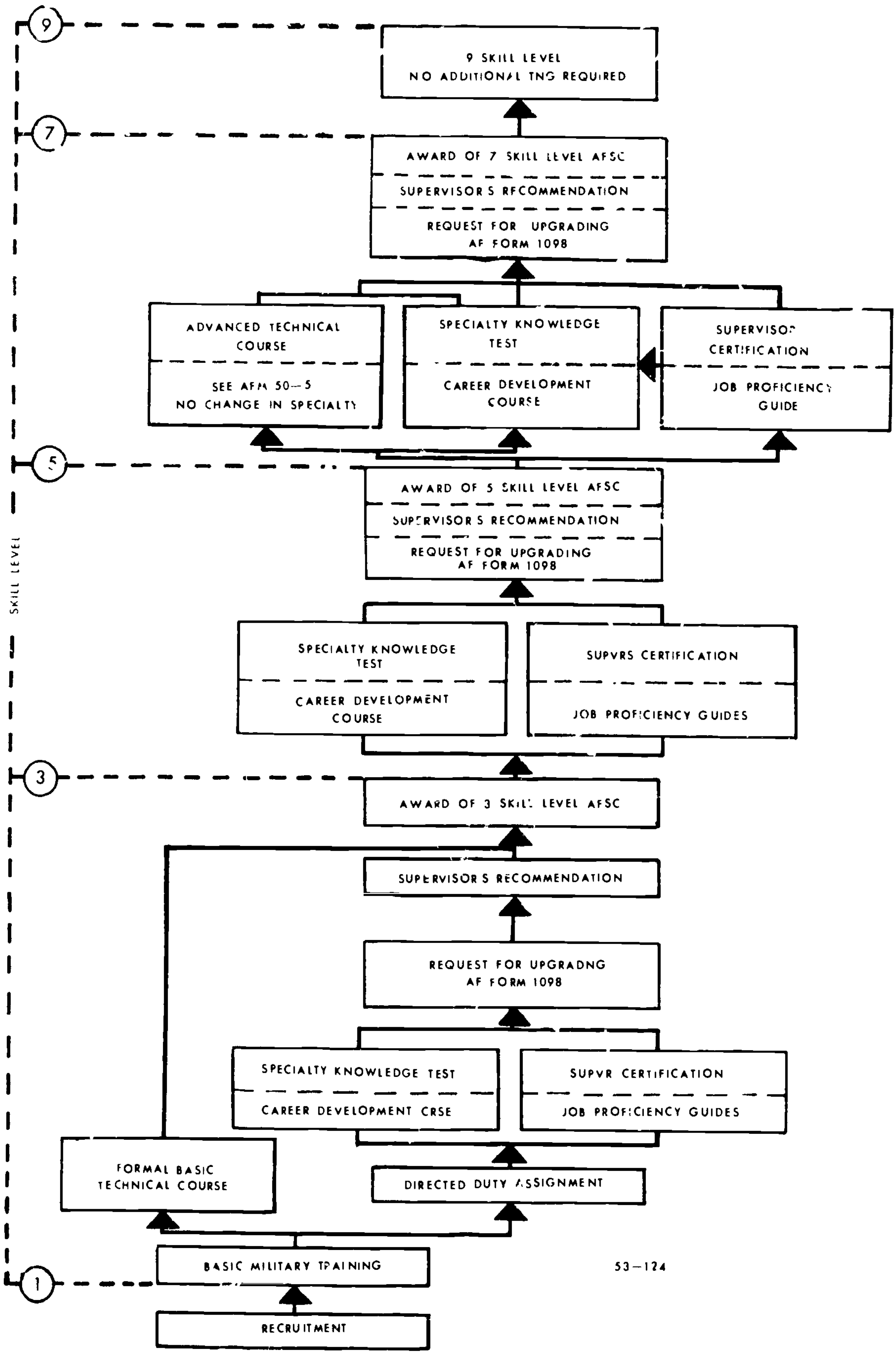
2-3. *Career development.* CDCs are correspondence courses based upon the specialty job description in AFM 39-1 and the related specialty training standard (STS). They include general Air Force subjects, specialty theory, and fundamentals and knowledge requirements for the airman's career progression in the AFSC of his assignment. The subject matter content of CDCs is prepared by Air Training Command. The CDCs are published and administered by the Extension Course Institute (ECI) under the direction of the Air University.

2-4. *Job proficiency development.* Job proficiency guides (JPGs) are a means by which airmen can attain proficiency by performing tasks of their specific assignment. This training uses the principle of "learning by doing," under the guidance of a qualified person. A JPG is used to develop each trainee's job proficiency and is required for training in his job. The JPG identifies specific tasks or duties to be performed and the degree of skill to be attained. The JPG indicates a supervisor's acknowledgment of a student's satisfactory achievement of required tasks and duties. It also contains necessary study reference materials. Specialty training standards (STS) are readily adjusted for use as JPGs. When STSs are used they should be so identified. A separate continuation sheet is prepared, if required. Supervisors must certify job proficiency as a prerequisite for upgrading actions.

2-5. **Use of CDCs for Upgrade Training.** Air Force Regulation 50-26 makes it mandatory for you to enroll in this CDC for upgrade training in your AFSC. Your supervisor will see that your training is conducted in accordance with regulations and that proper procedures are followed. It is up to you to make the most of your training. No matter how good the training program is, only you can do the learning and develop your skills. Only you can satisfactorily pass the course tests required for upgrading and only you can satisfactorily pass the skill knowledge test (SKT) required for promotion.

2-6. You will be continually enrolled in OJT throughout your career progression. As you satisfactorily complete your training in one AFSC of

CHART 2
CAREER PROGRESSION



your career ladder you will be eligible for enrollment in the next higher AFSC. Study the career progression chart shown in chart 2. This chart shows you how you progress for one skill level to th next in your career ladder Start at the bottom of the chart and read upward. The arrows indicate your career progression from the time you enlisted until you reach the top skill level of your career ladder.

2-7. **Consolidated Training Record (AF Form 623).** It is the responsibility of the shop supervisor to insure that an AF Form 623 is maintained for each airman in his shop. This form is a continuous record of all the career training you have received, including formal training, career development, and job proficiency training. The AF Form 623 is a four-page folder and is maintained in the shop. The JPG, which includes the STS, continuation sheet, and the management guide, is inserted in the folder. The proper maintenance of his AF Form 623 is also a concern of each airman in the shop, since it shows an up-to date. complete record of his training. The AF Form 623 accompanies your field record group if you are reassigned. Consult AFM 50–23, *Guide for Planning and Conducting OJT*, for detailed guidance on maintaining AF Form 623.

3. Technical Publications

3-1. What are technical publications? What are technical orders? Are they the same thing? We must start at the beginning in order to get a few points clear before we can progress further on this subject. When a person buys a new appliance, a new car, or even a small mechanical device, a little folder of operating and maintenance instructions is usually furnished by the manufacturer. These instructions are supplied so that the user may obtain the greatest possible satisfaction from his equipment. In the same way the Air Force is interested in seeing that its equipment functions properly and is maintained as efficiently as possible. Therefore the Air Force provides printed instructions for use with its equipment. These printed instructions (manuals, handbooks, sheets, charts, etc.), which provide technical instructions and information pertaining to the operation and maintenance of Air Force aircraft and equipment, are referred to as technical publications.

3-2. **Standard Publications.** Standard publications are authorized for issue by the Chief of Staff, USAF, or by Air Force commanders. We will explain briefly the types that you will most probably use in the shop.

a. Regulations announce policies, assign responsibilities, and direct actions. Regulations are permanent directives.

b. Manuals contain permanent and detailed instructions, procedures, and techniques telling personnel how to do their jobs. A manual may also be general and deal with principles.

c. Pamphlets contain information rather than directive material.

d. Supplements are auxiliary publications by which a commander insures efficient local compliance with directives issued by higher headquarters.

Consult AFR 5–5, *Publications, Policies, Responsibilities, and Standards,* for more detailed information about standard publications.

3-3. **Technical Orders.** Technical orders (TOs) are official Air Force publications which provide technical information, instructions, and safety procedures pertaining to the operation, maintenance, and modification of Air Force equipment and materials. A brief explanation of the various types will help you to understand the purpose, scope, and application of the technical order system.

a. Technical manual (TM). A TM contains instructions pertaining to the operation, maintenance, servicing, overhaul, installation, and inspection of specific items of equipment and materials.

b. Methods and procedures technical order (MPTO). An MPTO establishes policies and provides information and instructions on safe methods and procedures relating to such subjects as preventive maintenance, periodic inspection, and product improvement. There are two types of MPTOs: one deals with maintenance management or administration, the other with equipment in general.

c. Time compliance technical order (TCTO). A TCTO contains instructions for modifying equipment, for making special inspections, or for imposing temporary flight restrictions.

d. Index type technical order. The index type shows the status of all TOs, provides a means for selecting needed TOs, and groups TOs pertaining to specific items of equipment.

e. Abbreviated technical order. The abbreviated technical order is primarily a work simplification device, such as a checklist, inspection workcard, lubrication chart, or sequence chart.

3-4. *Files.* Finding the information in a technical order that you need depends upon your ability to locate the proper technical order file and to use technical order indexes. There are hundreds of technical orders and somewhere in them you can find almost any information relative to a machinist job. You will not find all of these technical orders in your shop file—this would not be practical. Your shop has a limited file. It contains those technical orders that are most frequently used by the people in your shop—they are the technical orders that explain how to do common maintenance tasks assigned to your shop. It is the responsibility of the shop foreman to procure the technical orders that are needed in the file. Suppose that your shop has a repair job for a transient aircraft jet engine component. You will probably not have a technical order in your file which explains how to do this job. You may be able to locate it in the jet engine repair shop. You may not be able to find it there, either, since this file is also a limited file. You may have to go to the field maintenance squadron file or even to the wing technical order file to find the technical order that you need.

3-5. *Indexes.* Locating the file which contains the technical order you need is one thing—determining which technical order you need is another. The secret of being able to determine which technical order you need is knowing how to use technical order indexes. You may find the information you need in a general, aircraft, specific aircraft, or an equipment technical order, or you may have to consult more than one type to find the desired information. Technical orders cover such a wide range of information that a single index would be impractical. Therefore, several indexes are required; there is even an index of indexes. Without a little advance knowledge, you can get lost in the indexes! Let's use some examples to determine which technical order you need. Assume that at various times you need to find information on the following subjects:

a. The Air Force technical order system.
b. Light-drive fits of aircraft bolts.
c. Metal surface activation procedure, B–52G.
d. Care and maintenance of cold chisels.

3-6. Your first task is to find the technical order which explains the Air Force technical order systems. This is a general technical order. Where do you start? You ca[illegible]ays start with the first numbered technical [illegible]ex—Technical Order 0–1–01, *Index of* [illegible]The –01 contains a list of indexes in numerical sequence, including 0–1–01 itself. Going down the list of indexes, you quickly come to Technical Order 0–1–02, *Basic Only—General Technical Orders.* This is the technical order that you need. The –02 contains a list of ge[illegible]ral technical orders in numerical sequence. Going down the list, you find Technical Order 00–5–1, *Air Force Technical Order System.* TO 00–5–1 is the technical order you need. Other examples of general technical orders listed in the –02 are the 00–20 series (maintenance management system), the 00–25 series (miscellaneous technical orders), and the 00–35 series (administrative technical orders).

3-7. Your second task is to find a technical order on light-drive fits of aircraft bolts. Obviously this is a general aircraft technical order and the technical order index for general technical orders will not help you, so you go back to the index of indexes—TO 0–1–01. Going down the list of indexes you come to Technical Order 0–1–1–1, *Basic Only—General Aircraft and Missile Technical Orders.* You are on the right track. Consult the table of contents of Technical Order 0–1–1–1. It is divided into four parts:

Part I, Published and Unpublished Technical Orders.
Part II, Aircraft to Engine Table.
Part III, Pilot's and Flight Crew Member's Information File.
Part IV, New Publications Listing.

Part I is the part you want. The listing in Part I, however, is divided into two sections:

Section 1, General Aircraft Technical Orders.
Section 2, Microfilm Indexes.

Section I of Part I is the section you want. Section I is broken down into technical order series:

1–1, General Aircraft Technical Orders.
1–1A, General Engineering Manuals.
1–1B, Weight and Balance Technical Orders.
1–1C, Airflight Refueling Technical Orders.

3-8. You can narrow your search to two series: 1–1 and 1–1A. There is really no clue that the 1–1 series is not the one you want. When you go down the list of technical order titles, though, you do not find one that is appropriate. So you go down the list of the 1–1A series titles and you find Technical Order 1–1A–8, *Engineering Handbook Series for Aircraft Repair—Aircraft Structural Hardware.* Consulting the table of contents for this technical order you find that Section III covers aircraft bolts. You have come to the end of your search; Section III, Technical Order 1–1A–8, explains how to obtain a light-drive fit on bolts.

3-9. Your third task is to find a technical order which will give you the procedure for activating metal surfaces on the B–52G. This will be a specific aircraft technical order. Again, the place to start is the *Index of Indexes,* Technical Order 0–1–01. Since you know that the B–52G is a bomber aircraft, locate the title, *Basic Only—Bomber Aircraft Technical Orders,* in the –01. The technical order number is 0–1–1–2. Consulting this index, you find that the technical order for structural repair instructions (B52G & B52H) is 1B–52G–3. Consulting the table of contents, you find that this technical order is divided into nine sections. The section titles enable you to limit your search to not more than two sections: I, General, and II, Typical Repairs. The information you need can be found in Section I.

3-10. Your fourth task is to find a technical order on the care and maintenance of cold chisels. This will be an equipment technical order. Again, you start with Technical Order 0–1–01. Go down the list until you find 0–1–32, *Standard and Special Tools*. Now, consult this index and you find Technical Order 32–1–101, *Maintenance and Care of Handtools*. It is a simple matter to find the section of the technical order which explains the care and maintenance of cold chisels.

4. Supply Discipline

4-1. The Air Force does not own property. The equipment issued to the machine shop and the toolbox issued to you are not owned by the Air Force. They are public property, controlled by the Air Force. The subject of supply is concerned with property accountability and responsibility and with the USAF Supply System.

4-2. **Property Accountability and Responsibility.** Accountability in the Air Force is the obligation imposed by regulation on a person to keep an accurate record of property or funds. Responsibility is concerned primarily with custody, care, and safekeeping. For example, the machine shop may have a numbered stock records account listing the tools and equipment used in the shop. The shop foreman may be required to sign for this account. He would then be obligated to maintain a record of this account. In this instance he is accountable for tools and equipment; he is also responsible, though not solely responsible, for care and safekeeping. Every user of a tool or item of equipment must pay for or make good the loss, destruction, or damage caused to that item by his neglect in its use. Suppose that some person, other than the shop foreman, signed the account. The foreman would then not be accountable for the tools and equipment, but, as the supervisor of the shop, he would share in the responsibility, along with his people, for the use, care, custody, or safeguarding of the property. According to AFR 67–10, *Responsibility for Public Property in Possession of the Air Force*, "pecuniary liability may be shared in any particular case by persons having command, supervisory, and custodial responsibility."

4-3. **Federal Supply Classification System.** The USAF is a complex organization, and it uses a complex supply system. The mission of a machine shop is to do maintenance. To carry out its mission, it must be supplied with a great variety of equipment: lathes, milling machines, hammers, and paper clips, to name a few representative items. Some of these items are nonexpendable, some are expendable. Maintenance has become so complex in the Air Force that it is necessary to separate the management from the production

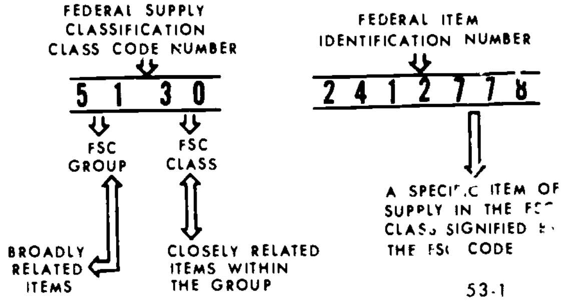


Figure 1 Federal classification.

functions. These two functions have also been separated in the USAF Supply System. The equipment management office (EMO) carries out the management functions, and base supply does the production functions. Both EMO and base supply are under the Chief of Supply. Another source that can assist you with supply problems is the Material Control Section of the maintenance organization. In an efficient machine shop most of your time should be devoted to maintenance and training, but knowing how to use stocklists and catalogs, however, is also important.

4-4. All branches of the Department of Defense use the Federal Supply Classification System. This system contains four, sometimes five, features:

- Supply classification—identification of supply items by group.
- Class identification—achieved by means of a four-digit code number.
- Item name—standardized nomenclature for an item.
- Item description—clear, concise description of an item.
- Item illustration—graphic representation of an item if necessary.

4-5. *Stock number.* To use a USAF stocklist you need to understand Federal stock numbers (FSN). At present, there are 76 Federal Supply Class (FSC) groups which are divided into approximately 564 classes. A four-digit code number is used to identify the group and class. For example, in FSC code 5130, the first two digits (51) identify the FSC group (handtools) and the last two digits (30) identify the FSC class (handtools, power driven). Figure 1 shows a Federal stock number for an item of supply. A stock number normally consists of 11 digits: a four-digit classification code number, plus a seven-digit Federal item identification number (FIIN). As explained in figure 1, the item identification number, 2412778, identifies a specific item in class 30 of group 51 of Federal supply items.

4-6. *Stocklists.* Stocklists are compiled and published on the basis of groups or classes in individual volumes which are arranged in numerical sequence in the stocklist file. Thus SL 5130 is the volume which contains information on items in code 5130. Normally stocklists contain an identification section, a stock control data section, a reference data section, and cross-reference data and indexes, if applicable.

4-7. The preface to each stocklist no longer contains general data that applies to all groups or classes. This data is found in Volume S–1; this volume should be used, therefore, in conjunction with individual stocklists.

4-8. In brief, the sections of a stocklist include:

a. Identification. The identification section is an alphabetical listing of all items in a numbered stocklist.

b. Stock Control Data. In the stock control data section, items in the numbered stocklist are listed numerically, as shown in figure 2 You will use this section frequently, in cooperation with supply personnel, to locate the information you need to fill out requisition forms and turn-in slips. We will explain the columns that you will use:

Column 1, Change. This column indicates a new or revised item in the stocklist. Refer to the preface for an explanation.

Column 2, Stock Number. This column lists all items in FSN sequence.

Column 3, Index Number. In this column the locator number for each item indicates where the item may be found in the item identification section. An index number beginning with a numeral indicates that the item description is found in Part 1 of the identification section, while an index number beginning with an alphabetical character indicates that the item description is found in Part 2.

Column 8, Status. An alphabetical code in this column designates status of items with regard to procurement and issue An explanation of the codes is found in USAF S–1.

Column 9, Expendability, repair, and cost codes. These codes determine the expendability of the items. The Air Force uses 10 codes; each is fully explained in USAF S–1.

STOCK CONTROL DATA SECTION
IN FEDERAL STOCK NUMBER SEQUENCE

CHG N C	STOCK NUMBER		INDEX NUMBER	VOL NO.	I / S	M / C	ISSUE AND FUND CONTROL	STATUS AF DDD	EXP	UNIT OF ISSUE	Q U P	STANDARD PRICE	SOURCE OR CATEGORY
1	4930	888-4961	0170-2000	B			IN	5	XF3	EA		92.00	SE
1	4930	888-4963	0175-3000	B				5	XF3	EA		70.00	SE
	4930	888-4964	CNS W7H 4930 886-9085										REF
	4930	888-4965	CNS W7H 4930 886-9087										REF
	4930	888-4968	CNS W7H 4930 886-9088										REF
1	4930	888-4970	0041-4000	B				5	XB3	EA		3.35	SE
1	4930	883-4972	0103-4000	B				5	XB3	EA		.27	SE
1	4930	888-5000	8000-4965	B				5	XB3	EA		7.20	SE
1	4930	888-5001	8000-37C7	B				5	XB3	EA		7.35	SE
1	4930	888-8580	8000-4509	B				5	XF3	EA		140.00	SE
1	4930	888-8581	0165-6000	B			IN	5	XF3	EA		52.00	SE
1	4930	888-8582	0165-5000	B				5	XF3	EA		40.00	SE
•	4930	888-8583	0165-4000	B				5	XB3	EA		17.00	SE
1	4930	888-8584	0170-6000	B				5	XB3	EA		19.00	SE
	4930	888-8585	0172-8000	B								3.50	SE
•	4930	888-8600	0020-2000	B								158.00	SE
1	4930	888-8601	0148-7000	B								18.00	SE
1	4930	892-2152											NPUB
	4930	892-4792	0085-4000	B	I			5 5	NB2	EA		137842.00	SE
1	4930	892-6261	8000-3341	B								149.00	SE
1	4930	892-6271	8000-4963	B								52.00	SE
1	4930	893-0582										.95	SE
1	4930	893-0583	8000-2745	B				5	XB3	EA		.50	SE
•	4930	893-0584	8000-4639	B				5	XB3	EA		155.00	SE
1	4930	893-0586							XB3	EA		.52	SE
1	4930	893-0587	8000-4655	B				5	XF3	EA		44.00	SE
1	4930	893-0588	0125-6000	B				5	XF3	EA		67.00	SE
1	4930	893-0589	8000-[illegible]005	B				5	XB3	EA		91.00	SE
	4930	893-0643	[illegible] 893-0643										REF

SEE PART 2 OF ITEM IDENTIFICATION SECTION

SEE PART 1 OF ITEM IDENTIFICATION SECTION

53 2

Figure 2. Stock control data section.

Column 10, Unit of Issue. An entry in this column indicates the unit of issue: each (ea), pound (lb), carton (ct), gross (gr), etc. S–1 gives a complete list of two-position abbreviations.

Column 12, Source of Category. Codes in this column show the source of supply or the category of the item. In requisitioning an item, the correct source or category code, together with the correct issue and fund control code, must be used.

c. Reference Data. This section is found in some stocklists. It contains drawings or illustrations of the items. Illustrations may be typical, rather than exact, representations; they are sufficiently accurate to enable you to identify the item.

d. Index. Two indexes normally appear in stocklists: manufacturers' name and address index and manufacturers' Federal code index. The first is an alphabetical listing of manufacturers' names cross-referenced to the manufacturers' Federal code numbers; the second is a listing of Federal code numbers cross-referenced to the manufacturers' names.

4-9. *Supply classification indexes.* The number of items covered by the Federal Supply Classification System sources is so vast that it is necessary to have indexes to assist the user.

a. Cataloging Handbook H2–1. This handbook gives the classification structure (groups and classes) of the FSC arranged by four-digit codes.

b. Numeric Index (Cataloging Handbook H2–2). This index is arranged numerically by groups and classes and contains within each class an alphabetical listing of entries included in the class.

c. Alphabetical Index (Cataloging Handbook H2–3). This index is an alphabetical listing of the entries in the numeric index. Each entry is referenced to a single four-digit FSC code number. You use this book and class if you have some idea of the correct name for an item.

d. Cross-reference indexes. These indexes assist in locating items of supply in stocklists. S–00–1–1 and S–00–1–2, both titled *USAF Cross-Reference Index,* cross reference manufacturers' identifying part numbers to manufacturers' codes and Federal stock numbers. The first lists them in manufacturers' part number sequence, and the second lists them in Federal stock number sequence.

e. USAF S–2A–1 index. This index lists USAF stocklists, Department of Defense Federal catalogs and related cataloging publications.

4-10. **Department of Defense (DOD) Federal Supply Catalogs.** DOD Federal supply catalogs will eventually replace USAF stocklists when a standard supply system for all military services becomes a reality. DOD Federal supply catalogs will be used whenever possible to requisition supplies. DOD Federal supply catalogs are indexed in S–24–1 by FSC classes. An example is FSC 5120, *Handtools, Federal Catalog C5120–1L–AF and C5120–ML–AF.*

a. The identification list (IL) consists of an alphabetical index, Federal stock number index, manufacturers' index, information table, and illustrations. It is used when a Federal stock number is known, when descriptive data or dimensions are required, or when the name of the item is known and the Federal stock number is required.

b. The management data list (MDL) is arranged in stock number sequence and contains such information as the item status, source category, and price.

4-11. How would you use DOD Federal supply catalogs to find information about an item? We will explain three different methods of research.

4-12. *Research by name.* Suppose you need to find the stock number for a 16-ounce tinner's hand riveting hammer. First, the group class must be determined by using the alphabetical index (H2–3) and looking under the listing "Hammers, Hand." You will find "Hammers, Hand" in class 5120. This will give you the first four digits of the stock number. Next, use the alphabetical listing in *DOD Federal Supply Catalog C5120–1L–AF* to find "Hammers, Hand Riveting." Here you will find the nomenclature and identifying numbers to complete the stock number. In this case, the stock number is 5120–224–4113 and the complete nomenclature is Hammer, Hand: Tinner's Riveting, Style 6, Section A, Ref DWG Group 84, FED GGG–H–86, Type IX, Class 1. You will also notice in the item description that style number 6, section A, and reference group 84 are mentioned. To be sure you have the right hammer you should look in the front of the catalog and find section A and the illustration of group 84 and style 6. To find the unit of issue, source category, and cost, look in the ML section. Stock numbers are listed numerically; and under the stock number 5120–224–4113 you will find the item is local purchase (LP), unit of issue (ea), cost $1.80, and the source category General Services Administration (GSA).

4-13. *Research by stock number.* Suppose you have stock number 5120–224–4113 and wish to find the item description. By looking in the index section of *DOD Federal Supply Catalog C5120–1L–AF,* you will find the stock numbers listed numerically. In this case the index number for stock number 5120–224–4113 is 100040–600. Using this index number, which is a locator number for quick reference, turn to the item description section of the catalog which is arranged by numerical listing of the index number. Index number 100040–600 will be followed by a description of the "Hammer, Hand Riveting."

4-14. *Research by part number.* Suppose you need the stock number and item description for a piece uipment in the shop and the manufacturer's identifying part number is known. Using *Cross-Reference Index* S–00–1–1, which lists part numbers numerically, locate the one you need. On the same line with this part number you will find the corresponding stock number. By using this stock number in the IL (or SL) index, you can find the corresponding index number. With this index number it is easy to find the item description, since the index numbers are listed numerically. When you research by part number, be sure the item description is for the right part or piece of equipment.

4-15. **Supply Condition Tags.** Supply condition tags are used to identify the condition of equipment, machinery, and parts. The 50-series tags have been replaced by new forms. The AF Form 50b (Serviceable) has been replaced by the yellow DD Form 1574; the AF Form 50e (Unserviceable, Condemned) tag is replaced by the red DD Form 1577; and the AF Form 50d (Unserviceable, Repairable) tag is replaced by the green DD Form 1577–2. These forms are to be attached to the supply/equipment item at all times in order to identify and indicate the condition of the item, except when it is installed or in use and the tag cannot be used because of operation hazards.

5. Communications Security (TRANSEC)

5-1. The degree to which you will be involved with security will depend on the mission of the base to which you are assigned, the type of equipment for which the base is responsible, and the location of the base. You may be involved with security quite often in your work or you may seldom be involved with it. Regardless of the degree of his involvement, every person in the Air Force, military or civilian, must be security conscious. The person who works with classified information only occasionally can be a potential source of danger, simply because he may be less aware of the importance of safeguarding security information.

5-2. **Classified and Unclassified Information.** The basic regulation which covers security is AFR 205–1, *Safeguarding Classified Information.* Classified information is official information which must be safeguarded in the interest of national defense. Unclassified information is official information which does not require safeguarding but may be subject to control for other reasons. Some unclassified information requires control in the public interest and is thus labeled "For Official Use Only." Defense information is classified according to its importance. The highest classification is given the greatest amount of protection. The three categories of classified information in the order of their importance are as follows:

TOP SECRET (TS).
SECRET (S).
CONFIDENTIAL (C).

Quoting from Section B, Chapter 1 of AFR 205–1:

> These categories are as follows:
>
> (1) The TOP SECRET classification applies only to that information or material the defense aspect of which is *paramount*, and the unauthorized disclosure of which could result in *exceptionally grave damage* to the nation.
>
> (2) The SECRET classification applies only to defense information or material the unauthorized disclosure of which could result in *serious damage* to the nation.
>
> (3) The CONFIDENTIAL classification applies only to defense information or material the unauthorized disclosure of which could be *prejudicial* to the defense interest of the nation.

5-3. **Safeguarding Classified Information.** Physical measures are taken to safeguard classified material and equipment. The purpose of any security program is to keep unauthorized people from gaining access to classified material and equipment. The Air Force may use armed guards, sentry dogs, or fences to keep unauthorized people out of restricted areas. A combination of all the physical safeguards might have to be used in a highly restricted area. A nuclear weapons storage area is, for instance, an area where all the safeguards might be used.

5-4. Banks use a safe or a vault for the storage of cash, bonds, and other valuable documents. Keeping classified documents locked in a safe when they are not in use is part of the Air Force's classified material control program. Even though the Air Force requires safes for the storage of classified material, it must also take proper precautions in assigning persons to handle classified materials. If you were assigned to a storage area for classified materials, you would need a security clearance equal to the highest level of the classification of the material you would handle. If the storage area contained SECRET documents, you would need a security clearance of at least SECRET.

5-5. A bank requires valuable documents to be registered and receipts to be signed whenever these documents are removed from a safe. In the Air Force, all TOP SECRET and SECRET documents must be registered and receipts must be signed whenever they are removed from a safe. Besides having official duties that require a SECRET clearance, you must meet two other requirements before you can be issued this document. You must sign a receipt and you must have a clearance of at least SECRET. A CONFIDENTIAL document may be issued without a signed receipt; however, a CONFIDENTIAL document must never be issued to an individual unless that person has a security clearance of at least CONFIDENTIAL.

5-6. **Transmission of Classified Information.** For information to be useful in the conduct of official business, the information must pass from one person or place to another. Mail, telephone, radio and messenger are modes of communication. Other modes of communication are the telegraph, television and signals. We will discuss only transmission by mail, telephone, radio, and messenger. Each method has certain advantages and disadvantages. Let's learn the uses, advantages, and disadvantages of each of these methods.

5-7. *Mail.* Within our country, mail is handled only by the United States Postal Service. Therefore it is given some protection. Registered mail requires special handling by the postal service. When the mailman brings you a registered letter, you must sign a receipt. An advantage of using mail is that the information is given some protection. It is inexpensive, and registered mail must be "receipted for." A disadvantage is that mail service is slow. Mail could get into the wrong hands, and only SECRET and below information can be sent by mail.

5-8. *Telephone.* The telephone is a very convenient mode of communication. Two advantages of using the telephone as a mode of communication are convenience and speed. A disadvantage of using the telephone is that it is a very insecure mode of communication, since telephone conversations are easy to monitor (listen in on). The telephone should not be used to send classified or sensitive information.

5-9. *Radio.* The radio is a good example of wireless communication. In radio communications an antenna is used to send radio waves through the air. When SECRET codes and chiphers are used to encrypt (convert a plain-text message into unintelligible form) classified information, radio is a secure mode of communication. At each Air Force base there are usually people who are trained to encrypt classified radio messages. Radio, like the telephone, has the advantages of convenience and speed. It is possible to send classified information safely by radio. The true meaning of a radio message can be hidden by using SECRET codes and ciphers to encrypt the message. Disadvantages of using the radio for communication are that persons have no way of knowing who is listening to their conversation, radio messages are easily monitored, and there could be times when they would be unable to hear each other on the radio, and the equipment could break down. Radio communication is, therefore, sometimes undependable.

5-10. *Messenger.* Information sent by messenger is hand-carried from the sending office to the receiving office. The advantages of using messenger are that the material is secure, receipts are required, and material usually is not subject to inspection. A disadvantage of using a messenger is that it is slow and expensive compared to other modes of communication.

6. Shop and Flight Line Safety

6-1. The number one cause of accidents is people! People who neglect, ignore, or deliberately violate safety rules that have been formed through other people's misfortunes over the years. Most all accidents are preventable if everyone would use common sense and be aware of potential hazards. Safety is a full time job. It is something that will affect you 24 hours of every day both on and off duty.

6-2. **Good Housekeeping.** The first rule of good housekeeping is personal cleanliness. Much loss of time and pain can be avoided if you keep yourself and your work area clean and orderly. Floors should always be kept free of obstructions or liquid spillage. Obstructions can be raw stock, chips from a machining operation, or tools. Liquid spills should be wiped up as soon as possible. It seems to hurt twice as much to slip on someone else's oil spill!

6-3. Some units that you disassemble have small parts that can be easily lost, broken, or mixed with other parts. To avoid loss of time while you hunt for another part, keep your bench top in neat and orderly condition. A cluttered bench makes effective work almost impossible and can cause accidents. Dispose of wornout parts promptly.

6-4. Every shop has a designated place for toolboxes when they are not in use. Keep them in place and keep the lids closed. It does not require much time or effort to close the box after you get a tool. You may prevent someone from getting hurt. Most shops have a tool board for special tools; keep these tools on it.

6-5. Good ventilation is one of the main factors conducive to good work, good health, and safety. Your work output will drop off if you are uncomfortably hot or cold or if there is a lack of fresh air. If the air is dusty in the shop or if fumes are present, tell your supervisor so that the will be aware of the conditions and can correct them.

6-6. Good lighting is another requirement for doing good work. Definite standards have been set up by the Air Force to provide correct lighting.

These standards should be followed since good lighting works hand in hand with good housekeeping to eliminate many safety hazards.

6-7. **Fire Prevention.** Fire prevention cannot be overemphasized, since the Air Force loses millions of dollars every year in accidental fires. Most of these fires are caused by carelessness in disposing of combustible materials, such as oily rags and smoking materials. This is the main reason that smoking is not permitted within 50 feet of parked aircraft hangars, shops, or any building where flammable liquids are stored or being used.

6-8. All combustible materials, including clean rags, should be placed in covered metal containers. Soiled rags should go into a self-closing lid type metal container, and should not be mixed with any other type of waste. This is done to prevent spontaneous combustion, which can occur at any time.

6-9. All Air Force machine shops use paint and oil almost daily. These items should always be stored separately in metal lockers, at least 50 feet from the nearest building. All containers should be closed, to prevent spillage and waste. No smoking within 50 feet of this type of storage is permitted.

6-10. Overloaded electrical outlets and defective circuit breakers are also fire hazards. Here are a few precautions that you should observe; you can add to the list from your own experience.

a. Observe the signs in the NO SMOKING areas.

b. Do not allow your clothing to become saturated with fuel or oil. If they do, change your clothing and wash yourself as soon as possible.

c. Do not store gasoline, kerosene, jet fuel, or any other flammable liquids in open containers.

d. Always make sure that the static lines are in place and that the aircraft is grounded properly before you work on it.

e. Do not put cigarettes or matches in a waste basket even if they appear to be out.

f. Do not open any oxygen valve near a flame or a lighted cigarette.

6-11. If fires do occur, no matter how many precautions are taken, you must be ready to fight them quickly and effectively. You should know the telephone number of the base fire department, the location of the fire extinguishers, and which type of extinguisher to use for the type of fire you are fighting. The telephone number of the base fire department is usually posted in large figures on posters in the shop, in the barracks, and on the flight line. As a rule, the base telephone directory has this number printed in large figures on the cover page or on one of the first pages of the book. If alarm boxes are installed on your base, learn where they are and how to use them. Fire extinguishers look alike. Don't use the wrong type of extinguisher as it can make the fire worse. Chart 3 shows the types of fires that the various types of extinguishers can be used on. Every shop has at least two types of fire extinguishers, and it is a good idea to become familiar with their correct use. Water type fire extinguishers should never be used for any other purpose than fighting fires.

6-12. **Protective Equipment.** As a machinist you will be exposed to flying metal chips, sharp edges of tools and work, both moving and stationary, and slippery surfaces caused by lubricants and coolants. The Air Force issues all necessary protective safety devices, such as goggles, gloves, aprons, and boots. But it is up to you to use these devices. You will also be exposed to sound from jet engines and turbines. These sounds can eventually make you deaf if you don't protect your ears. Because of your high exposure to cuts and scratches it is a good idea to keep your tetnus immunication up to date.

6-13. Eye protection must be worn when chips or particles are thrown into the air by a machine tool. This includes grinding, shaping, milling, drilling, and lathe turning. Remember, you can chew with glass teeth, but you can't see with a glass eye.

6-14. You will work around machines every day, both in the shop and out on the flight line. To prevent getting caught in the machines, don't wear the following items: loose fitting jackets, unbuttoned sleeves, ties, finger rings (including wedding rings), bracelets, necklaces (including dog tags), key chains, or wristwatches. These items can cause you to lose a finger, hand, or worse if they should get caught in a piece of machinery.

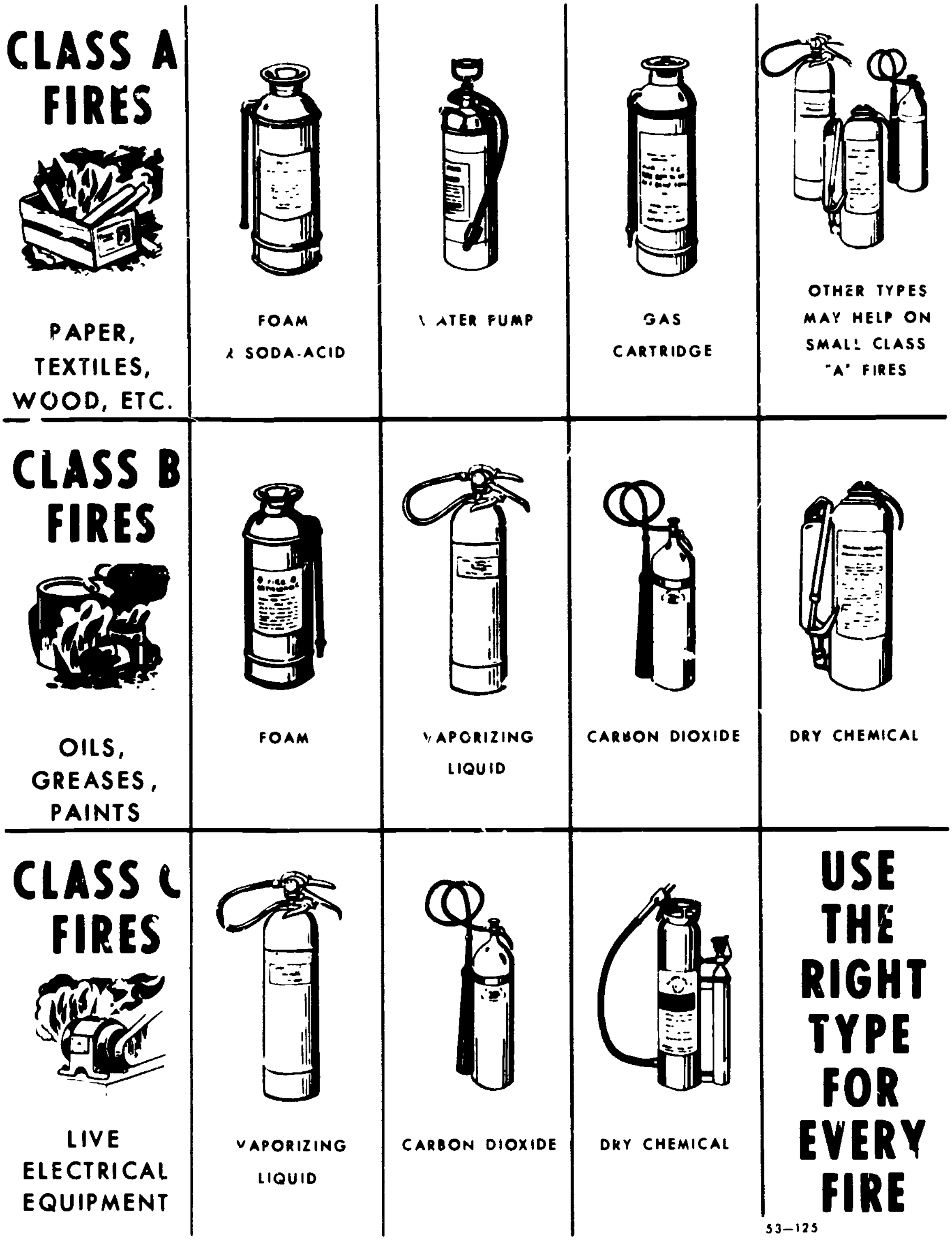
6-15. Guards are placed over moving parts of machines to prevent your fingers and clothing from getting caught and pulling you into the machine. A machine should not be run without the guards in place or in good condition. If you have ever ridden a bicycle without a chain guard while you were wearing long trousers, you know how easy it is to get caught.

6-16. Horse play and practical "jokes" have no place anywhere, let alone a machine shop. If you accidentally get shoved into a heavy, cast iron machine during a friendly scuffle, it doesn't hurt the machine at all, but can be very painful to you.

6-17. The power tools that you will use are either electrically or air driven. Each type has its own hazards.

6-18. **Electrical Hazards.** Many people think that high voltage is the main cause of death in electrical shock. However, statistics show that over 700 people die every year from an ordinary house voltage of 110 or 220 volts. Extension cords are a potential source of trouble. Usually they are not heavy enough to stand the abuse to which they are

Chart 3
Classes of Fire Extinguishers



subjected Use an extension cord only if it is necessary. If a cord must be used to operate a piece of electrical equipment, be sure that the cord is large enough to carry the load. Oil or fuel spilled on the cord will cause rapid deterioration of the rubber cover. Place the cord so that it will not be stepped on or run over. The danger from electricity increases greatly if you work in wet clothing or stand in a wet place. Serious burns can result from contact with high voltage in junction boxes or in loose gound wires. Before using any electrical tool, make sure that it is properly grounded. This will help prevent your getting a possible fatal shock The cord from tool to plug should be in good condition, with no broken or frayed wires.

6-19. Never attempt an electrical repair unless you are authorized to do it. If you are permitted to make the repair, make sure you know what you are doing before attempting a job on wiring to which power is applied. If possible, remove the power before repairing a circuit. If you are not authorized to repair a circuit, shut off the power and notify the electrical repair section. If a fire should result from an electrical breakdown, use a chlorobromomethane (CBM) or carbon dioxide extinguisher to fight it. If the fire is in a close place, always use the carbon dioxide type, since CBM extinguishers emit very dangerous fumes. Before using an air-driven tool, be sure the pressure in the lines is within limits (don't exceed 150 PSI). This will prevent damage to the tool from overspeeding. The air hoses should be in good condition, to prevent rupture, because a ruptured air hose can whip around and injure you.

6-20. **Tools.** If tools are not used properly, they can become a hazard. Statistics show that approximately 10 percent of preventable injuries were caused by defective or improperly used tools. We have included some pointers on the use of tools:

a. Have a place for everything and keep everything in its place. The condition of your toolbix is a good indication of your mechanical ability. You should be able to find any of the tools you use most often without searching for them.

b. Store tools safely. Sharp-edged tools should not be stored loosely in your toolbox or in your pocket.

c. Keep tools in good condition. Chisels or punches should not have mushroomed heads. A dull cutting edge is dangerous on a chisel because you have to hit harder to do the same work. Hammer handles must be secured properly to prevent the head from flying off.

d. Keep tools clean. Oil prevents rust, but it also causes tools to become very slippery and a toolbox to become messy. Wipe excess oil from the tools.

e. Use the correct tool. A wrench is not a hammer and a screwdriver is not a pry bar.

f. Use tools properly. Knives or other sharp tools should not be pulled toward the body. Screwdrivers should not be applied to objects that are held in the hand. Never push a wrench when you can pull on it. When you push a wrench to loosen a nut, it may break loose unexpectedly and bruise or skin your knuckles.

6-21. **Flight Line.** You do most of your work in the shop; however, it may be necessary for you to do some of your work on the flight line. Flight line safety, therefore, will affect your directly. We will discuss some of the hazards that are encountered on the flight line.

6-22. *Jet engines.* The temperature and velocity of the exhaust gases behind an operating jet engine are great enough to cause serious injury. The temperature of the exhaust gases 25 feet behind one of the smallest engines, the J–69, installed in the T–37 training aircraft, exceeds 350° F. The velocity of the exhaust gases is another hazard. As a general rule the minimum safe distance behind an operating engine is 200 feet; for the F–4 it is 250 feet. Blast fences help reduce the safe distance. A jet engine uses a large volume of air. All of this air is taken into the intake. The suction developed immediately in front of the engine is enough to pull caps, coats, or men into the engine. The minimum safe distance in front of the engine is 25 feet. Do not approach closer than 5 feet to the duct entrances from the side or rear. All objects must be removed from the area in front of the ducts before the engine is started. Most bases use antipersonnel screens on the front of the intake.

6-23. *Turbine wheels.* The area in line with the plane of rotation of the turbine wheel must be kept clear. If a wheel should suddenly disintegrate when the engine is running, the pieces can be thrown a long distance. Standing directly in line with a turbine wheel is like looking down the barrel of a loaded and cocked rifle. The same precautions apply to the turbine wheels of pneumatic or combustion (fuel-air) starters. They turn at a very high rate of speed and are extremely dangerous. The turbine wheel danger area of the later model jet engines and turboprop engines are much larger, since they use multistage turbine wheels.

6-24. *Propellers.* Turboprop engines are used on some of the later model cargo aircraft, such as the C–130 The propeller is dangerous if proper precautions are not observed. Many persons have been seriously hurt or killed by propellers.

6-25. *Noise.* The high-frequency sound created by the modern jet engine is a hazard to physical and mental health. Prolonged exposure to noise can cause you to become completely deaf and can build up nervous tension to the breaking point. The noise level of a 5,000-pound thrust turbojet engine can cause pain. A larger turbojet with afterburner can cause not only pain but physiological symptoms. A good example of very high noise level is the F-4 aircraft which uses the 15,000-pound thrust J-79 engine. The area at a tangent behind the aircraft is extremely dangerous. Where the two sides of the cone meet, the danger area extends 1,600 feet in a traight line from the tail of the aircraft.

CAUTION: Always wear earplugs or protective muffs when you are in the area of an operating engine. These protective devices are somewhat uncomfortable to wear, but a hearing aid is uncomfortable, too. Noise can be considered the most hazardous of all, because you don't have to be close to the aircraft to be affected by it.

6-26. *Other hazards.* Other hazards can be encountered on the flight line. Most hazards can be avoided by simply paying attention to what you are doing. Remember, most accidents are caused by carelessness. Here are a few other flight line precautions:

a. Keep work stands clean and in good repair.

b. Install guard rails, especially on high stands.

c. Install the safety pins on all hydraulically operated work stands before you use the stands.

d. Keep tools in your box. A loose tool on a stand can cause a serious fall.

e. Do not place toolboxes in a position where they can fall and hurt someone.

f. Be careful working around the trailing edges of the wing and control surfaces. These edges are sharp. The leading edge of the wing on some aircraft, such as the T-38, is just as sharp.

g. Do not work in the flap or spead brake area until you are sure that these controls cannot be operated.

h. Be sure the aircraft static grounds are installed and in good condition.

i. Do not wear jewelry or a wristwatch while you are working. They can catch on sharp surfaces and seriously injure you.

6-27. **Radioactivity.** Radioactivity is not normally a health hazard in a machine shop. You may never be required to work near radioactive materials, such as certain parts on newer jet engines or radioactive contaminated materials; however, radiation is such a serious health hazard that elaborate precautions are taken in the Air Force to guard against even accidental exposure. Since you cannot see radioactivity, several hours may elapse after exposure before you feel any effects; therefore you must be able to instantly recognize radiation warnings.

6-28. *Types of radiation.* There are three types of radiation: alpha, beta, and gamma. A single particle of radioactive material may emit all three types of radiation. Alpha and beta radiation have very little penetrating power. Alpha radiation penetrates only a few inches of air and beta radiation penetrates only a few feet of air. External contact is not harmful. Alpha radiation is harmful only if you breathe, eat, or drink contaminated particles, or if they come in contact with a broken skin surface. Exposure to beta radiation is more hazardous than alpha radiation because of its greater penetrating power. You can protect against beta radiation by wearing a respirator and protective clothing. Gamma radiation penetrates many hundreds of feet of air; there is no barrier that can completely stop its penetration. External contact is extremely hazardous and can result in cell damage either externally or internally. The intensity of gamma radiation is expressed as roentgen hours (r/hr). The primary purpose of the various AFTO Form 9 warning signs is to guard against gamma radiation.

6-29. *Radiation signs.* Figure 3 shows a radiation warning placard. All radiation signs display the distinctive three-bladed magenta colored insignia against a yellow background with black block type. The warning signs are designed to attract immediate attention. Each sign is designed for a specific purpose, and the exact size for most of them is specified by technical order. The AFTO 9 series forms are listed below:

a. AFTO Form 9, Radiation Area Warning Placard (8½ x 11 inches). These placards are posted in conspicuous places. They indicate that the radiation intensity in the area exceeds one milliroentgen hour (mr/hr) but is less than 100 mr/hr.

b. AFTO Form 9A, Radiation Warning Tag (3½ x 6¼ inches). This tag identifies radioactive parts, equipment, or material. It is attached to each item.

c. AFTO Form 9B, Radiation Warning Label. This is a flexible but durable gummed label. A sufficient number of labels are attached to insure that one is visible from any direction of approach.

d. AFTO Form 9C, Radioactive Material Warning Placard (8½ x 11 inches). This placard identifies an area in which radioactive materials are stored.

e. AFTO Form 9D, Ingestion Hazard Placard (8½ x 11 inches). This placard warns against eating, drinking, or smoking in the area. It is displayed as directed by the base medical service.

f. AFTO Form 9E, High Radiation Warning Placard (18 x 24 inches). This placard identifies

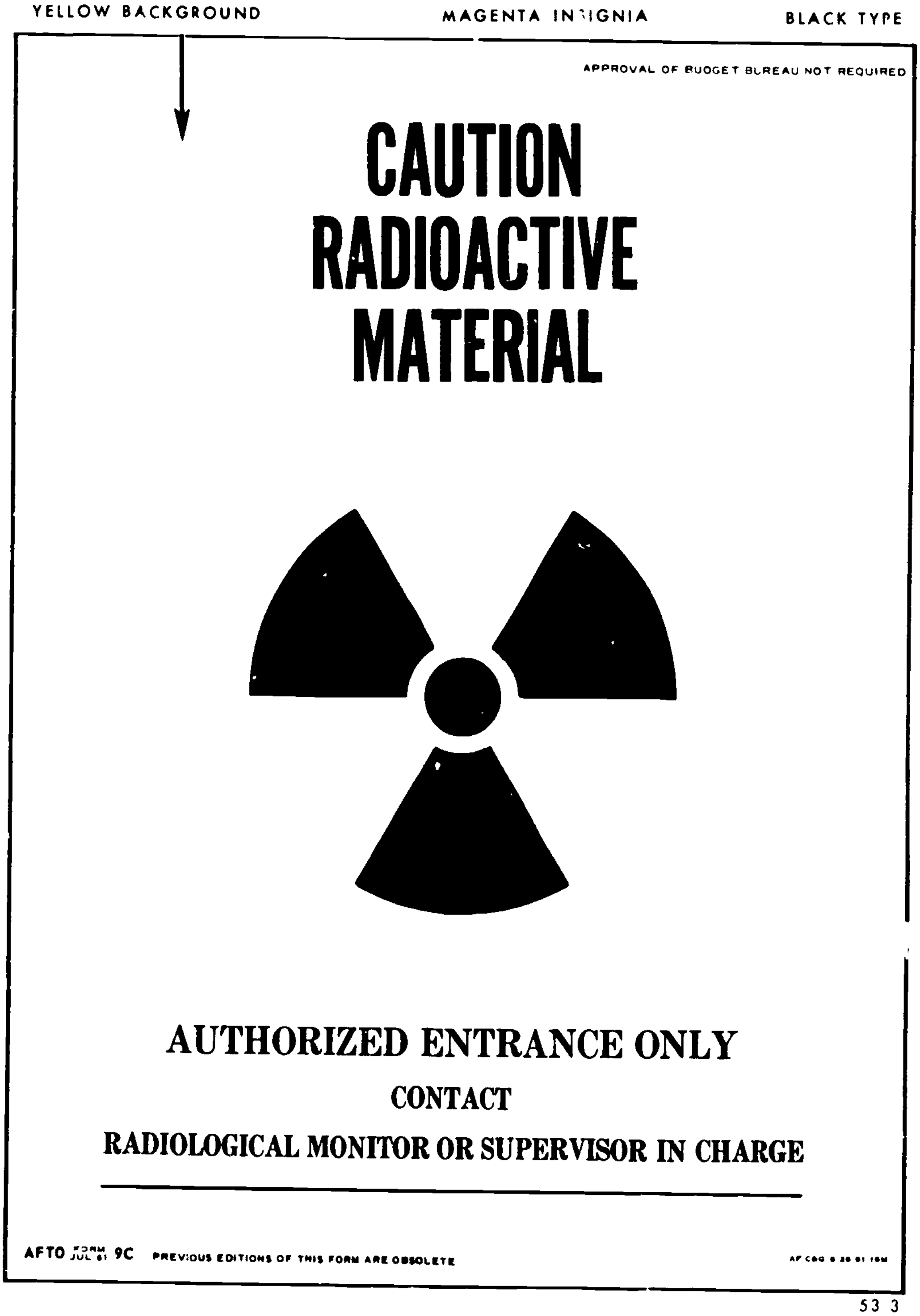


Figure 3. Radiation warning placard.

an area in which the radiation intensity exceeds 100 mr/hr.

g AFTO Form 9F. Airborne Radioactivity Warning Placard. The use of this placard is directed by the base medical service when radioactivity is present

6-30. Prior to working near radioactive or radioactive contaminated materials, machine shop personnel, and in particular the shop foreman, should become familiar with the 00–110 technical order series. This series provides useful information about safety precautions, decontamination procedures, and radioactive waste disposal. For more details about marking and identification of radioactive and radioactive contaminated materials, consult TO 00–110N–3, *Requisition, Handling, Storing, and Identification of Radioactive and Radioactively Contaminated Material.*

Metals and Heat Treatment

"WHAT METAL should I select to machine a new gear? Should it be heat treated? Will the finished gear be hard enough for its intended use?" These are only a few of the questions that you will probably have in mind when you begin work on a project. To answer these and other questions, you must have a knowledge of metals, heat treatment, and hardness testing. Without this knowledge you will not be productive in your job.

7. Classification of Metals

7-1. All metals may be classified as ferrous or nonferrous. A ferrous metal is any metal that has iron as its main element. A metal is still considered ferrous, even though it contains less than 50 percent iron, as long as it contains more iron than any other metal. A metal is considered nonferrous if it contains less iron than any other metal. Keep this in mind as we discuss the first grouping, ferrous metals.

7-2. **Ferrous Metals.** This group includes cast iron, steel, and the various steel alloys. The only difference between cast iron and steel is in the amount of carbon. Cast iron contains more than 2 percent carbon, while steel contains less than 2 percent. An alloy is a substance composed of two or more elements, one of which must be a metal. Therefore all steels are alloys of iron and carbon, but the term "alloy steel" normally refers to a steel that also contains one or more other elements. For example, if the main alloying element is tungsten, the steel is a "tungsten steel" or a "tungsten alloy." If there is no other alloying material, it is a "carbon steel."

7-3. Alloy steels have been developed to meet the needs of modern industry for tougher, stronger, and harder steels than can be obtained in simple carbon steels. Alloys such as nickel, chromium, tungsten, vanadium, manganese, and molybdenum all give distinct properties to the steel; but in all cases the principal qualities are the increase in hardness and toughness. The general characteristic of various steels are listed.

7-4. Some common steels and their uses include:

a. Carbon steels are usually classified as low, medium, or high carbon steels.

(1) Low carbon steels are generally used for parts which do not require much strength.

(2) Medium carbon steels are often used for parts not intended for aircraft use. They are stronger than low carbon steels.

(3) High carbon steels are used for springs and wear-resistant tools. High carbon steel can be made very hard by heat treatment.

b. Nickel steels have better impact resistance at subzero temperatures than carbon steels and are harder and stronger. They can be hardened by heat treatment.

c. Nickel-chromium steels are tough, strong, and hard and are used for aircraft and engine parts. They can be hardened by heat treatment. Their strength and impact resistance are not affected by subzero temperatures when they are properly heat treated.

d. Molybdenum steels are all hardenable by heat treatment. Plain molybdenum steel is used for aircraft structural parts. Molybdenum steels are often alloyed with chromium or nickel, or both. These alloys are used for aircraft engine and structural parts.

e. Chromium steels are used for parts requiring high strength, depth hardness, and corrosion-resistant properties.

f. Chromium-vanadium steels are used for high-strength parts. They are not affected by subzero temperatures when they are properly heat treated.

g. Tungsten steels are used for ultrahigh-speed cutting tools and are very heat resistant. The cutting edge does not break down when red hot, so high cutting speeds are possible.

h. Chromium-nickel-molybdenum steels are used for various aircraft structural parts. They are similar to the molybdenum steels. When properly heat treated, these steels are not affected by subzero temperatures.

i. Silicon-manganese steels are very tough and shock resistant. They are often used for springs but are brittle under impact at subzero temperatures.

j. Silicon-chromium steels are spring steels but are not recommended for subzero use.

k. "Tool steels" are steels suitable for the manufacture of cutting tools. High carbon, drill rod, tungsten, chromium-vanadium, and silicon-manganese steels all fall in this class.

l. "Stainless steel" is really a trade name but is commonly used to refer to any corrosion-resistant alloy. Corrosion-resistant steel can be a nickel-chromium steel or a chromium steel. These steels are used where a high degree of corrosion and heat resistance is required.

7-5. **Nonferrous Metals.** This group includes many metals that are used mainly for metal plating or an alloying element, such as tin, zinc, and silver. We are primarily concerned with the metals used for the manufacture of parts, such as aluminum, magnesium, titanium, nickel, copper, inconel, and their alloys.

a. Aluminum is very light in weight, corrosion resistant, and an excellent conductor of both heat and electricity. Because of its light weight, it is one of the most important metals used by the aircraft industry. It is generally used in the form of an alloy, with copper, manganese, silicon, magnesium, or zinc as the main alloying materials. The term "aluminum" used in reference to the various alloys as well as to commercially pure aluminum. Aluminum is used wherever weight reduction is a factor and extreme hardness and strength are not critical.

b. Magnesium is an extremely lightweight metal, weighing approximately two-thirds as much as aluminum. Because of its light weight, magnesium is also an important aircraft construction material; however, it will corrode readily in a salt-air atmosphere unless protected with paint.

NOTE: The machining of magnesium presents a fire hazard, because the fine chips produced by machining may ignite and burn intensely. Never use water soluble oil (water and oil solution) as a coolant or cutting lubricant for magnesium. Water intensifies a magnesium fire. *Dry* sand is the only safe extinguisher for burning magnesium.

c. Titanium is a lightweight metal that has a good weight-to-strength ratio. It is 44 percent lighter than stainless steel, yet it has about the same strength. It is also highly corrosion and temperature resistant.

NOTE: Fine chips of titanium may also ignite; however, they do not ignite as easily as magnesium. The chips should be kept dry, because when titanium oxidizes with water, highly explosive hydrogen gas is generated.

d. Nickel is a metal that is highly corrosion resistant. Copper, chromium, iron, aluminum, and zinc are used with it to form various alloys.

e. Copper is metal that is an excellent conductor of both electricity and heat. The most common of the copper-base alloys are brass and bronze. The chief alloying element in brass is zinc; in bronze, it is tin.

7-6. Information on machining all of these metals can be found in TO 1–1A–9, *Aerospace Metals.*

8. Identification and Selection of Metals

8-1. Metals may be identified by a numerical code or by a color code. When the specification number is not indicated, the machinist must select a suitable metal.

8-2. **Identification.** After 3 days of hard, tedious work on a complicated sheet metal punch part, Airman Menard took it to a metals processing shop to be heat treated. Much to his dismay, he was told that the material of which the part was made was not suitable for the heat treatment required. "I don't understand," he said, "I took it from the correct metal rack." Airman Menard had learned the hard way the value of being able to properly identify metal. Had he understood the numerical coding systems and the Air Force color code, his costly mistake could have been avoided.

8-3. *Numerical codes.* The terms "steel" and "aluminum" are general in meaning. There are many different types of steel and aluminum. They vary greatly in their chemical composition and mechanical properties. Every piece of metal is manufactured to meet certain specifications. Since it is not possible to mark all of this data on each piece of metal, it is represented by a specification number. The ideal situation is to have the specification number marked on each piece of metal in the metal rack. When this is not possible, each piece should be either tagged or color coded.

8-4. Unfortunately there is no single, unified numerical code for metals. Each manufacturer has his own code or specification number system; this is confusing, since there is no uniformity between manufacturers. In an attempt to correct this situation, several agencies of the metals industry and the Federal Government have developed specification number systems. As a result you may find any one of seven different specification code numbers stamped on a piece of metal or written on an identification tag. Five of the seven codes cover both ferrous and nonferrous metals. The Aluminum Association code is restricted to aluminum. The American Iron and Steel Institute code is restricted to ferrous metals.

8-5. Perhaps the best known numerical code is the Society of Automotive Engineers (SAE) code. SAE specifications are rather broad and are not complete procurement specifications. The SAE numerical code is the basic code for ferrous metals. It is especially useful in identifying steels by chemical composition.

8-6. The SAE system is based on the use of four- or five-dight numbers. The first number indicates the type of steel; for example, 1 indicates a carbon steel, 2 indicates a nickel steel, etc. In the case of alloy steels, the second number—and sometimes the third—usually indicates the approximate percentage of the principal alloying element. This is usually the one for which the steel is named. The final two numbers—sometimes three—indicate the approximate carbon content in one-hundredth of 1 percent (0.01 percent). The SAE series numbers are given in chart 4. The following examples will help you understand this system:

SAE–1045

1 Type of steel (carbon)
0 Percent of alloy (none)
45 Carbon content (0.45 percent carbon)

SAE–2330

2 Type of steel (nickel)
3 Percent of alloy (3 percent nickel)
30 Carbon content (0.30 percent carbon)

SAE–71650

7 Type of steel (tungsten)
16 Percent of alloy (16 percent tungsten)
50 Carbon content (0.50 percent carbon)

SAE–50100

5 Type of steel (chromium)
0 Percent of alloy (less than 1 percent chromium)
100 Carbon content (1 percent carbon)

CHART 4
SAE NUMERICAL CODE

TYPE OF STEEL	SAE NUMBERS
Carbon Steels	1XXX
Plain Carbon	10XX
Free Cutting, Manganese	X13XX
Free Cutting, Screw Stock	11XX
High Manganese	T13XX
Nickel Steels	2XXX
.50% Nickel	20XX
1.50% Nickel	21XX
3.50% Nickel	23XX
5.00% Nickel	25XX
Nickel-Chromium Steels	3XXX
1.25% Nickel : .60% Chromium	31XX
1.75% Nickel : 1.00% Chromium	32XX
3.50% Nickel : 1.50% Chromium	33XX
3.00% Nickel : .80% Chromium	34XX
Corrosion and Heat Resisting	30XXX
Molybdenum Steels	4XXX
Chromium-Molybdenum	41XX
Chromium-Nickel-Molybdenum	43XX
Nickel Molybdenum	46XX & 48XX
Chromium Steels	5XXX
.60% to 1.10% Chromium	51XX
1.2% to 1.5% Chromium	52XXX
Corrosion and Heat Resistant	51XXX
Chromium-Vanadium Steels	6XXX
Tungsten Steels	7XXXX & 7XXX
Silicon-Manganese Steels	9XXX

53-126

CHART 5
ALUMINUM ALLOY GROUPS

MAJOR ALLOYING ELEMENT	
Aluminum at least 99% pure	1XXX
Copper	2XXX
Manganese	3XXX
Silicon	4XXX
Magnesium	5XXX
Magnesium and Silicon	6XXX
Zinc	7XXX
Other Elements	8XXX
Unused Series	9XXX

53—127

8-7. The American Iron and Steel Institute numerical code (AISI) is essentially the same as the SAE code. For example, SAE–1030 and AISI–C1030 are carbon steels of identical chemical composition. One difference between the two codes is that the prefix of an AISI number indicates the process used in the manufacture of the metal. The C in AISI–C1030 indicates that the steel is a basic open-hearth carbon steel.

8-8. The Aeronautics Division of the SAE Standards Committee has developed the Aeronautical Material Specifications (AMS) code. These specifications are complete procurement specifications. The chemical composition of AMS metals are coordinated as closely as possible with SAE general standards for similar metals. An example of an AMS number is AMS–5045B.

8-9. The American Society for Testing Materials numerical code (ASTM) has much in common with the Aeronautical Materials Specifications code. The ASTM specifications are also complete procurement specifications and have many detailed requirements in addition to the chemical composition of the materials.

8-10. The Federal specifications numerical code was developed to aid in the procurement of supplies used by the departments and independent agencies of the Federal Government. Federal specifications are complete procurement specifications and have many detailed requirements in addition to the chemical analysis of the material.

8-11. The Department of Defense has developed a numerical code which consists of two series: (1) military specifications (MIL) and (2) joint Army-Navy specifications (AN). They can be used as procurement specifications and may have other detailed requirements in addition to those of chemical composition.

8-12. The Aluminum Association numerical code (AA) was developed for aluminum and aluminum alloy products. Specifications are very general in nature; they are not complete procurement specifications.

8-13. The Aluminum Association identification system as shown in chart 5 consists of a four-digit number which indicates the type of alloy, control over impurities, and the specific alloy. The first number indicates the type of alloy. For example, 2 is copper, 3 is manganese, 4 is silicon, etc. The second number indicates the control that has been used. The last two numbers usually indicate an assigned composition. Thus, AA–2024 means:

2 . Type of alloy (copper)
0 Control of impurities (none used)
24 .. Exact composition (AA number 24)

In the case of commercially pure aluminum, the 1000 series, the last two digits of the identification

CHART 6
AIR FORCE COLOR CODE

COLOR	NUMBER	LETTER
BLUE	1	F
GREEN	2	H
OLIVE DRAB	3	O
YELLOW	4	T
ORANGE	5	W
RED	6	A
MAROON	7	B
WHITE	8	C
GRAY	9	D
BLACK	0	S

53 128

number represent the amount of aluminum above 99 percent in one-hundredth of 1 percent. Thus, AA–1040 indicates a commercially pure aluminum containing 99.40 percent aluminum.

8-14. Aluminum alloys varying greatly in hardness and physical condition or characteristics are available. We call these differences "temper." Letter symbols represent the different tempers. The temper designation is separated from the basic four-digit identification number by a dash, for example, 2024–T6. In this case we have an aluminum alloy, 2024, with a T6 temper (solution heat treated and then artifically aged).

8-15. We have discussed seven different specification number systems. The Department of Defense has developed a master code which ties all of these systems together. It is explained in Military Handbook H1B, *Cross Index of Chemically Equivalent Specifications and Identification Code (Ferrous and Nonferrous Alloys)*. The master code groups materials of similar chemical composition which may be represented by one or more specifications. The five-digit code numbers cover both ferrous and nonferrous alloys but are not specification numbers and cannot be used to procure materials. This identification code is extremely valuable, because it is a reference for comparing the material composition of various specifications.

8-16. *Air Force color code*. For ease in identifying metals in storage, the Air Force uses a color code system, which is described in TO 42D–1–3. The material designation numbers are represented by stripes of paint at each end and in the middle of the stock. The stripes are parallel and of equal width, ½ to ¾ inch wide, and placed side by side and in sequence. When read from either end of the bar toward the center, the material designation numbers are shown in proper order. A ¼ inch or less space is left between stripes of the same color appearing side by side. In addition to the basic numerical code, the temper designation of aluminum alloys is represented by stripes. These stripes are located ¾ to 1 inch from the alloy identification stripes. Ten colors are used to represent the various numbers and letters as shown in chart 6. Figure 4 shows the color coding of SAE–1045 steel, and figure 5 shows the color coding of AA–2025–T6 aluminum. Note that if read from either end toward the center, the numbers are in correct sequence.

8-17. **Selection.** The type of metal to use in a repair job is usually specified by a technical order or blueprint, but often you will have to determine which type to use. You will have to consider the intended use and the properties of the metal.

8-18. Internal reactions of a material to external forces are known as the mechanical properties. These properties exist in a definite relationship to

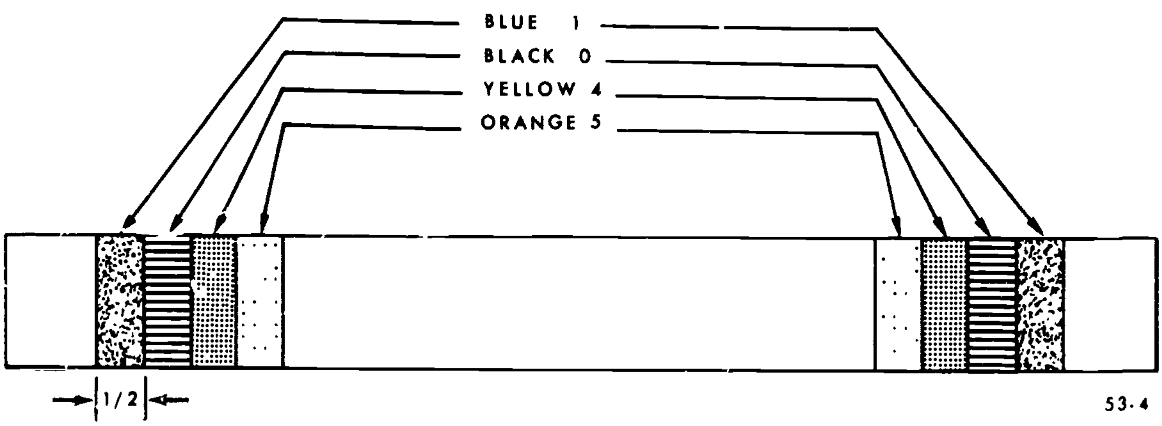


Figure 4. Color code for 1045 steel.

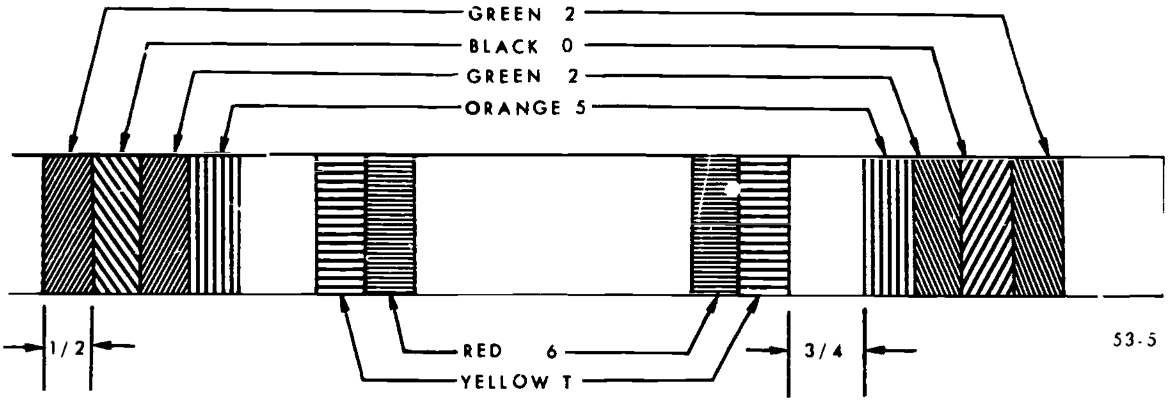


Figure 5. Color code for 2025–T6 aluminum.

one another. You must have a knowledge of these properties to determine which metal to select.

a. Hardness is the resistance a substance offers to deformation or penetration. The hardness of a metal can be controlled by heat treatment.

b. Tensile strength is the resistance that a substance offers to being pulled apart under a slowly applied load. This will increase or decrease as the hardness of a substance increases or decreases. It is usually stated in pounds per square inch of cross-sectional area.

c. Brittleness is the tendency of a material to fracture or break with very little deformation, bending, or twisting. Brittleness is usually not a desirable mechanical property. Normally the harder the material, the more brittle it is.

d. Shear strength is the resistance to an action similar to that produced by a pair of scissors. A shear action is a force acting in a tangential manner which tends to cause the particles of a body to slide over each other. The shear strength of steel is approximately 60 percent of the tensile strength and can also be controlled as tensile strength is; i.e., by varying the degree of hardness.

e. Ductility is the ability of a substance to be elongated without breaking. Metals that are comparatively soft are usually ductile.

f. Toughness is the ability of a material to absorb sudden shock without breaking. Usually the harder the material, the less tough it is.

g. Wear resistance is the ability of a substance to resist cutting or abrasive action resulting from a sliding motion between two surfaces under pressure. A hard material will usually have good wear resistance.

8-19. Suppose that you are required to make a part that will be subjected to wearing action and also to shock and impact. You should select a metal that can be case hardened to provide a good wearing surface and a tough inner core to readily absorb shock. It is always necessary to know the intended use of the part and the forces that will be acting on it. The *Machinery's Handbook,* or any other machinist handbook, is a good source of information in metal selection. Such a source usually contains a list of various metals and their suggested uses and examples of typical tools and the metals recommended for their manufacture. Perhaps the metal suggested is not available, but at least, by consulting such a source, you will have an indication of the type of metal to use and you can then select a metal with similar characteristics from the metals on hand. Drill bushings, for example, are required to hold the rotating drills in accurate alinement and to maintain this accuracy while being subjected to wear caused by the drill flexing and moving off center. Drill bushing metal must be hard to prevent excessive wear and loss of accuracy, and so you should select a metal that can be made extremely hard by heat treatment, such as SAE 1095.

8-20. You will probably never have to decide what type of material to use to manufacture an aircraft part. This is an engineering decision, and normally the type of material to be used is specified by a technical order, a microfilm, or a drawing or blueprint. Keep in mind that extensive research went into the selection of the many metals used for aircraft parts and that each part should be made out of the same material, or better, as the original. Use no substitute material unless you are specifically told to do so. If you are ever in doubt about the selection of a metal, talk to a more experienced machinist or a machine shop technician.

9. Heat Treatment

9-1. Why is it important for you to have a knowledge of the heat treatment of metals? Most of the metals you work with in a machine shop are heat treated or will require heat treatment after machining. Since heat treatment is so often neces-

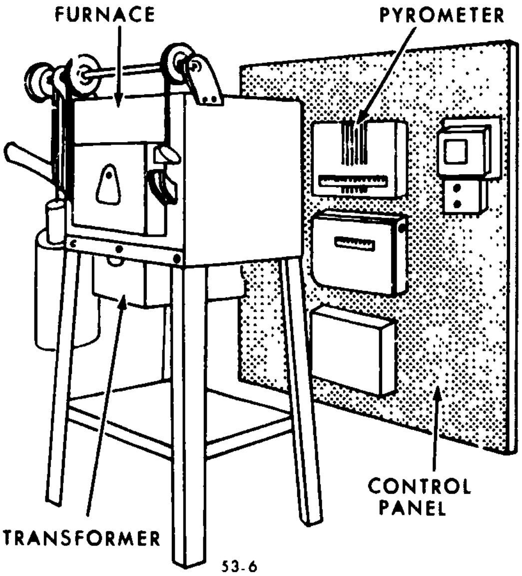


Figure 6. Electric furnace.

sary to obtain the desired properties you need in metal parts, it is important for you to know about it. Heat treating is a metals processing task, but the machinist must determine the need and must arrange for heat treating. Every new part must possess the same properties as the original part. To obtain these properties, heat treatment may be required. In this section we will discuss heat-treating furnaces, heat-treating operations and their effects, and heat-treating requirements.

9-2. **Heat-Treating Furnaces.** Heat-treating furnaces may be classified by the type of fuel used—electricity, gas, oil, or charcoal; by the condition of the internal air, either still or circulated; or by the type of atmosphere in the furnace—oxidizing, carburizing, or neutral.

9-3. *Electric furnace*. Most furnaces used by the Air Force are of the still air, electrically heated type, as illustrated in figure 6. Electrically heated furnaces are used because they are clean, efficient, and automatic, and they require very little maintenance. They operate on the same principle as electric toasters, electric irons, and electric stoves. An electric current is passed through a conductor having a high resistance, and heat is generated. An accurate temperature control and indicator called a pyrometer, shown in figure 7, is used to maintain constant temperatures. It is set to the desired temperature and indicates when that temperature has been reached. Then it turns the current off and on so that the temperature setting is accurately maintained.

9-4. *Atmospheres*. When steel is exposed to air, it "rusts," or oxidizes. In other words, the oxygen in the air reacts chemically with the surface of the metal. Heating the air in a heat-treating furnace accelerates this chemical reaction and causes the metal to scale. When the amount of scaling is not excessive, ordinary air (oxidizing atmosphere) is used in the furnace. When the amount of scaling is undesirable, it can be controlled by using a carburizing or "reducing" atmosphere. Increasing the carbon content prevents scaling. A carburized atmosphere may be produced outside and forced into the furnace. At times an atmosphere which is neither oxidizing nor reducing may be needed. A neutral atmosphere may be used; however, this type of atmosphere is very difficult to maintain.

9-5. **Heat-Treating Processes and Their Effects.** There are five basic heat-treating processes; hardening, case hardening, annealing, normalizing, and tempering.

9-6. *Steps in a heat-treating process*. All five of the heat-treating processes can be performed on ferrous metals. Most nonferrous metals can be annealed, and many, such as certain aluminum or magnesium alloys, can also be hardened by heat treatment. However, nonferrous metals are never tempered, normalized, or case hardened. There are three steps in each of the five operations:

(1) Heating is the first step. Certain changes take place at known temperatures. The metal must be heated until it is hot enough for the desired changes to occur.

(2) Soaking is the second step. This involves letting the metal remain at the required temperature until it is evenly heated throughout. The more mass a part has, the longer it must be soaked.

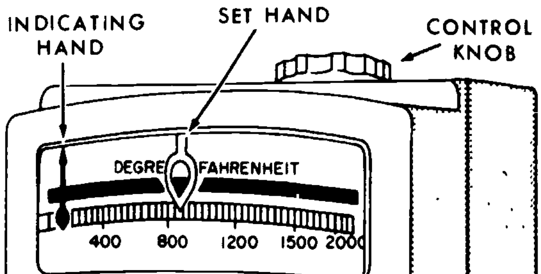


Figure 7. Pyrometer.

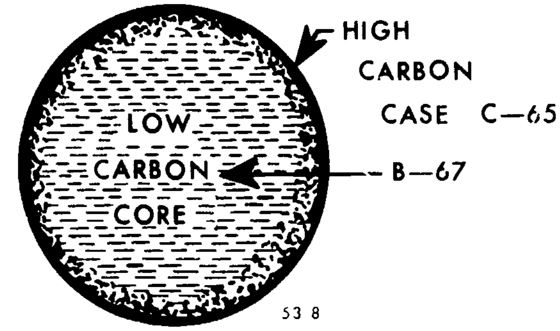


Figure 8. Case hardened steel.

(3) Cooling is the final step. Cooling back to room temperature can be done in several ways. The substance used to cool the part is called the cooling medium.

9-7. *Effects.* We will now explain the effects of the five basic heat-treating operations:

a. Hardening causes metal to better resist deformation and penetration. However, as a metal hardens, it usually becomes more brittle, and it is sometimes necessary to take steps to reduce the brittleness.

b. Case hardening forms a hard surface over a tough core, as shown in figure 8. This is usually done by increasing the carbon content of the metal near its surface. Low carbon steels are often case hardened.

c. Annealing is a process in which metals are softened, usually to make them easier to form or machine.

d. Normalizing removes internal stresses in metals caused by forging, machining, casting, or welding. Metal is tougher in this condition than in any other.

e. Tempering follows the hardening operation, especially with high carbon steel; it reduces brittleness and increases elasticity. Most metals lose some of their hardness wnen tempered, but high-speed steel increases in hardness when tempered.

9-8. **Design in Relation to Heat Treatment.** The serviceability of a tool or part is directly affected by its design and heat treatment. Failure of a part is often due to improper design and heat treatment, which may cause the part to crack, warp, or break. The internal stress produced by machining and heat treating the part may reduce its strength by as much as 90 percent, causing it to fail under a relatively light load. Failure of a newly designed part is usually caused by failure to consider its heat treatment requirements. Designs which have abrupt changes in mass or size may cause serious stresses to develop during heat treatment. A part properly designed will heat and cool evenly.

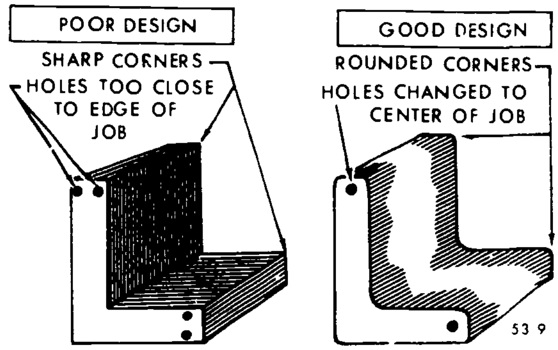


Figure 9. Corner and fillet design.

9-9. The rate of cooling is affected by differences in mass and size and also by the shape and surface finish. When a part is quenched, even though it is uniform in cross section, the shape may cause it to cool unevenly and to develop stresses. Sharp corners and edges are rounded whenever possible, since they produce a great concentration of stresses. Holes located near an edge also produce stress, as do holes located off center in thick parts. Holes can be drilled on the opposite side to balance the mass of the part. Tool and chatter marks are also stress risers and can be removed by filing and polishing before the heat treatment operations. Figures 9 and 10 illustrate good and poor design in relation to heat treatment.

9-10. **Determining Required Heat Treatment.** The heat treatment required for any part depends upon many factors. You should consider the type of metal, the composition of the metal, and the use the part will be put to. In many cases the required heat treatment is specified by technical orders or are given on the blueprint. For information on the temperatures and controls required for successful heat treating, consult TO 1–1A–9, *Aerospace Metals.* When you make a tool or a part, you must determine which of the mechanical properties it must have if it is to be efficient. All of the mechanical properties are present to a degree in all metals. Most of them can be developed to a higher

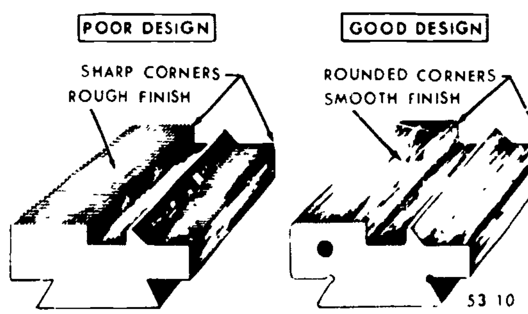
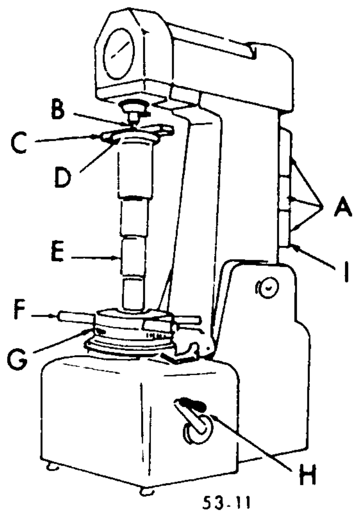


Figure 10. Corner and surface design.



A. WEIGHTS
B. PENETRATOR
C. SPECIMAN
D. ANVIL
E. ELEVATED SCREW
F. HANDWHEEL
G. ZERO ADJUSTER
H. TRIP LEVER
I. WEIGHT PAN

Figure 11. Rockwell hardness tester.

degree than normal. Increasing one will affect another. You determine which of the mechanical properties are the most important and which are the least desirable by determining the job that the tool must do. An example is a cutting tool which must be harder and have more wear resistance than the material it is to cut; but, as hardness and wear resistance increase, brittleness increases, and shock resistance decreases. A brittle tool with little or no shock resistance is of no value. Another example is a part designed to break under a given load or sudden jolt. Brittleness, then, is of great importance.

9-11. The heat treatment of an aircraft part is usually specified by a technical order, drawing, blueprint, or microfilm; you will seldom if ever, make the decision as to the heat treatment for such items. At times, however, you will manufacture items that do not have the required heat treatment specified, such as locally designed tools and equipment. When the heat treatment is not specified, you can usually find a suitable heat treatment in a reference publication such as *Machinery's Handbook*. For example, this handbook has a listing "Reamers, tool steel for." Turning to the given page number, you will find that the recommended steel for hand reamers is drill rod and that the recommended hardness is Rockwell C63–66. If the item requires annealing or normalizing, this information will be given in the remarks section of the same book. This is all the information that is normally given.

9-12. If you are unable to find information pertaining to the selection and heat treatment of metal for the particular item you are to machine, consult the metals processing technician. The metals processing technician is required to perform the actual heat treatment; however, you must give him enough information so that he can decide exactly what procedures to use. First of all you must be able to tell him the type of metal being heat treated. The more completely you can identify the composition of the metal, the better. If you know the specification number of the metal, tell him.

9-13. You should also tell the metals processing technician the desired Rockwell hardness number, or at least the intended use of the item. You should be able to tell the metals processing technician, "This is a piece of SAE–1095 steel that has to be hardened to a Rockwell hardness rating between C–63 and C–66," or "This is a hardened SAE–4130 steel that must be annealed for further machining." This information will help the metals processing technician to determine the exact procedures to use to obtain the desired results.

10. Inspection of Metals

10-1. This section covers hardness testing, corrosion control, and nondestructive inspection. As explained in Chapter 1, there are separate ladders in the metal working career field for corrosion control and nondestructive inspection. Hardness testing is a metals processing task. Although you are not primarily responsible for these three areas, you should have some knowledge of them.

10-2. **Hardness Testing.** Let's suppose that a work order comes to your shop which requires the fabrication of an aircraft replacement part. You may work from a blueprint which specifies the required dimensions, type of metal, heat treatment, and hardness, or you may find all these specifications in a technical order. In either case you select the specified type of metal and make the part. But the finished part may not possess the desired properties. For example, it may not be hard enough. Metal is usually furnished in a soft or annealed condition so that it can be machined. Therefore, after you have finished making the part, you send it to the metals processing shop to be heat treated. The metals processing technician heat treats it as specified in the blueprint or technical order. Then he tests it to be sure that the specified hardness has been obtained.

10-3. The Air Force uses the Rockwell hardness tester because it is simple to operate, it can test a great variety of metals of varying degrees of hardness, and it does not depend upon the judgment of the operator for accuracy. Figure 11 illustrates a Rockwell hardness tester, while figure 12 shows some of the attachments which are used with it. The principle of Rockwell hardness testing is based on the degree of penetration of an indentor

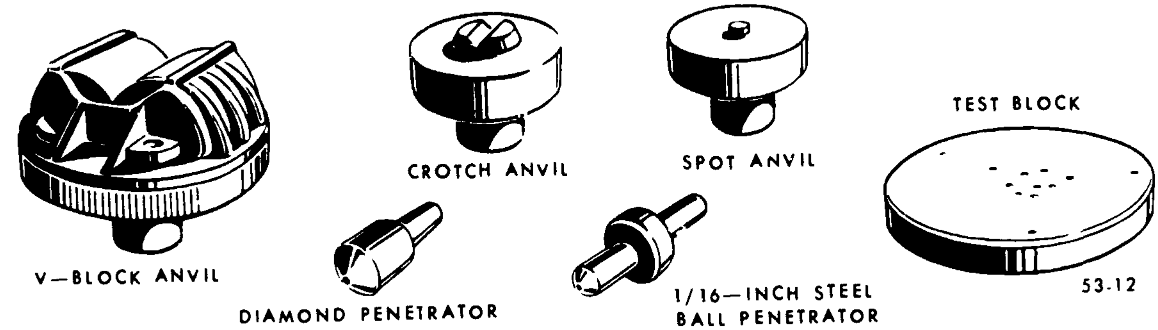


Figure 12. Rockwell hardness tester attachments.

into a material under a given static load. When the lever shown on the right side of the base in figure 11 is tripped, a predetermined load or weight forces a penetractor into the metal being tested. The hardness value depends upon the depth of penetration and is indicated on the dial. The shallower the penetration, the higher the hardness number. The penetrator can be either a diamond or a hardened steel ball.

10-4. Before the major load is applied, the test speciman must be securely locked in place to prevent slipping and to properly seat the anvil and penetrator. To do this, a load of 10 kilograms (approximately 22 pounds) is applied before the lever is tripped to apply the major load. This preliminary load is called the minor load. The minor load is 10 kilograms, regardless of the major load or the penetrator used. When the tester is set properly, the machine automatically applies the 10-kilogram minor load.

10-5. The load that is applied to force the penetrator, shown in figure 12, into the metal is known as the major load and is measured in kilograms (1 kilogram is approximately 2.2 pounds). The major load can be 60 kilograms, 100 kilograms, or 150 kilograms, depending upon the material being tested. After the pointer has come to rest, the hardness of the material can be read from the dial graduations. Figure 13 shows a dial face.

10-6. **Corrosion Control.** The deterioration of metal because of its reaction to its environment is called corrosion. A good example of corrosion is a rusty nail. The rust is formed by the reaction of the metal with the oxygen in the air. This reaction is called oxidation. The oxidation rate increases as the moisture content and temperature of the air increases. Other elements besides oxygen cause corrosion. Salt air, for instance, which surrounds many air bases in the US and overseas, is highly corrosive because of the reaction of the chemicals in salt with metals.

10-7. *Identification.* Corrosion always starts on the surface of a metal. Many times you can detect it with the naked eye. We will list several common types of corrosion and briefly explain how you can identify them.

a. Pitting corrosion is the most common type and is found on aluminum and magnesium in the form of a white or grey powdery deposit. When this deposit is removed, small holes, or pits, are visible on the surface.

b. Intergranular corrosion is usually caused by poor heat treatment. All metals are composed of very fine grains, each with a different composition at the center than at its boundary. A small corrosion cell develops when a corrosive agent comes in contact with the metal surface. This corrosion progresses rapidly along the grain boundaries and appears as a bulky mass.

c. Exfoliation corrosion is a form of intergranular corrosion. It also starts at the surface and progresses along the grain boundaries under the surface, expanding enough to cause the face of the metal to lift up.

d. Galvanic corrosion occurs when dissimilar metals are in contact. The electrical potential difference between the metals, in the presence of an electrolyte such as water, develops and electrochemical cell and corrosion occurs. You can recog-

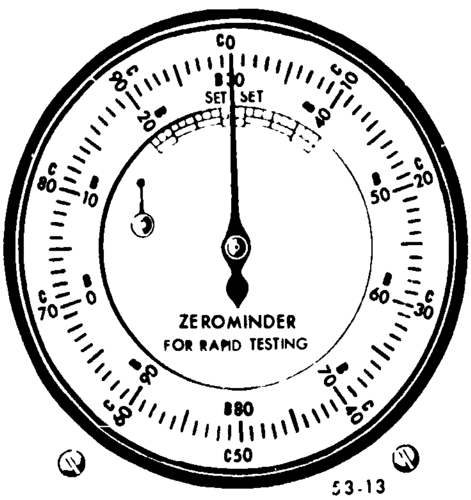


Figure 13. Dial face of a Rockwell hardness tester.

nize this type of corrosion by the buildup of material at the joint between the metals.

e. Stress corrosion is caused by corrosion occurring at a point of high tensile stress in the metal. The combined corrosion and concentration stress causes the metal to crack.

10-8. *Prevention.* Scheduled inspections and preventive maintenance are essential in combating corrosion. Coating aluminum alloy sheet with commercially pure aluminum, which is then called Alcad, is one method that is used to prevent corrosion. The most important measure in preventing corrosion is the removal of foreign matter from unprotected metal surfaces. Keeping protective finishes in good condition is another corrosion control measure.

10-9. One method of protecting a metal from corrosion is to plate its surface with another more corrosion-resistant metal. In some instances, plating also produces a more wear-resistant surface. Plating is applied by a process called electroplating, in which a pure metal and the part to be plated are placed in a liquid solution called an electrolyte. An electric current is passed from the pure metal through the electrolyte to the part, causing tiny particles of the pure metal to be deposited on the surface of the part. The process is continued until the plating reaches the required thickness. The plating thickness required varies, but in most cases, it is extremely thin. The surface may be plated with two or more metals to make the final plating adhere better or to reduce the cost. For example, in the chromium plating of steels, the steel is often plated first with copper, then with nickel, and finally with chromium. Electroplating is the responsibility of the metals processing shop. There are several types of electroplating, with the most common being cadmium, copper, nickel, and chromium.

10-10. Other metal surfaces treatments which aid in corrosion control are anodizing, metal spraying, hot dipping, and painting. Some corrosion-resistant metals require no protection. You should refer all metal surface treatment problems to the metals processing shop.

10-11. **Nondestructive Inspection.** Nondestructive inspection is any form of inspection in which the part inspected is not damaged or destroyed. Visual inspection is an aid to nondestructive inspection. It is a quick and economical method of detecting small defects before they can cause failures. Its reliability depends upon the ability and experience of the inspector. He must know how to search for structural failures and how to recognize areas where such failures are likely to occur. Defects that would otherwise escape the naked eye can often be detected with the aid of magnifiers.

10-12. Visual inspection of materials, parts, and complete units is no longer the most important method of determining their condition. Various nondestructive inspections are used to detect variations in structure, changes in surface finish, and the presence of physical defects. Although nondestructive inspection, AFSC 536X0, is a separate ladder of the metalworking career field, you should have a basic understanding of nondestructive inspection methods.

10-13. Training is extremely important in nondestructive inspection. Present nondestructive inspection methods, such as those of magnetic particle, fluorescent and dye penetrant, eddy current, ultrasonic, and X-ray inspection are becoming more and more exacting. The importance of the decisions of the person who performs these inspections cannot be overemphasized.

10-14. The physical location and supervisory control of nondestructive inspection equipment are important in insuring efficiency and economy in performing inspections. Nondestructive inspection equipment should be placed in one location and under the supervision of one trained supervisor.

10-15. In field maintenance activities all parts that require inspection should be taken to an inspection laboratory. A central location prevents duplication of expensive equipment and permits more efficient use of inspection personnel. The quality of inspection equipment maintenance is also improved since one trained person is responsible for carrying out the maintenance program.

Preparatory Shop Work

THIS CHAPTER covers the basic knowledge and skills that you need to know before you begin to work with machine tools. Shop mathematics is involved in every machine shop task. To perform most of these tasks, you must be able to read blueprints and shop drawings. Geometric construction is required to make shop drawings and for layout work. Finally you must know how to use common hand and measuring tools.

11. Review of Shop Mathematics

11-1. Nearly every item you make will require the use of mathematics. Measuring, calculating cutting speeds, machining tapers, manufacturing gears, and many other tasks make use of mathematics. Arithmetical operations involving whole numbers should not cause you any trouble. Machinists constantly work with fractions and decimals. In this section we will first identify some commonly used mathematical symbols and terms which every machinist must know in order to work at his trade. Our purpose is simply to make certain that you know what these symbols and terms are. After identifying these symbols and terms, we will explain fractions and decimals, stressing their addition, subtraction, multiplication, and division.

11-2. **Mathematical Symbols and Terms.** Symbols such as ÷, ∠, and π and terms such as "product," "quotient," and "dividend" are often used in mathematics. If you have been out of school for some time, you may not remember what they mean.

11-3. *Symbols.* The following symbols often appear in data that a machinist uses. Not all of them are used in this section of the text. However, since you will be using all of them later in your work, you should be sure that you know what they mean.

+ plus
− minus
× multiplied by
÷ divided by
= equals
// parallel
⊥ perpendicular
∠ angle
∟ right angle (90°)
π pi (3.1416)
° degree
% percent

11-4. *Terms.* Mathematical terms are used to quickly and accurately identify numbers being discussed. For example, it is much simpler to say, "the product of 6 and 3 is 18," than to say, "the number obtained by multiplying 6 by 3 is 18." The "product" also shows that 6 and 3 are multiplied, not added, subtracted, or divided. The following information will show you the location and terms which identify the numbers used in addition, subtraction, multiplication, and division:

Addition (+)	Addend *Addend* Sum
Subtraction (−)	Minuend *Subtrahend* Difference
Multiplication (×)	Multiplicand *Multiplier* Product
Division (÷)	Quotient Divisor /Dividend

11-5. **Fractions.** A fraction is a number that shows the number of equal parts of a unit. For example, 3/4 means that three of the four equal parts of a unit are used; 7/32 means that seven of the thirty-two equal parts are used.

11-6. *Definitions.* It will be easier for you to understand the operations if you know the definitions of the terms. These are as follows:

a. Denominator. The denominator is the number which shows into how many equal parts the unit has been divided. The denominator is written below the line. *Example:* 3/4; here 4 is the denominator.

b. Numerator. The numerator is the number which shows how many equal parts of the unit have been used to make a fraction. The numerator is written above the line. *Example:* 2/3; here 2 is the numerator.

c. Terms. The terms of a fraction are the denominator and the numerator. *Example:* 1/2; here 2 is the denominator; 1, the numerator.

d. Common fraction. A common fraction is a fraction which has both terms expressed. *Example:* 3/4; here both 3, the numerator, and 4, the denominator, are given.

e. Proper fraction. A proper fraction is one whose numerator is less than its denominator. *Example:* 3/8, 5/16; here 3 and 5 are less than 8 and 16, respectively.

f. Improper fraction. An improper fraction is one whose numerator is greater than its denominator. *Example:* 4/2, 5/3; here, 4 and 5 are greater than 2 and 3, respectively.

g. Unit fraction. A unit fraction is one whose numerator is one. *Example:* 1/2, 1/4; here each numerator is 1.

h. Mixed number. A mixed number is a number composed of a whole number and a fraction. *Example:* 3¾, 7½; here 3 and 7 are whole numbers, while 3/4 and 1/2 are fractions.

i. Complex fraction A complex, or compound, fraction is a fraction in which one or both of its terms is a fraction or mixed number. *Example:* 3¾ over 6, 15/3 over 32, 3¼ over 7⅝.

11-7. *Reducing a fraction.* Reducing a fraction means changing a fraction from one form to another without changing its value. *Example:* 1/2 changed to 2/4 or 4/8; here the relative value of the terms is the same in all three forms of the fraction. The value of a fraction is not changed by multiplying or dividing its numerator and denominator by the same number. *Examples:*

$$1/2 \times 4/4 = 4/8$$
$$1/2 = 4/8$$
$$8/12 \div 4/4 = 2/3$$
$$8/12 = 2/3$$

11-8. When both the numerator and denominator cannot be divided by the same number, the fraction is said to be in its lowest terms. *Example:* 1/5, 2/3; here no common number can be divided into 1/5 or 2/3; thus each of these fractions has been reduced as far as possible—to its lowest terms. An improper fraction may be reduced to a whole number or a mixed number by dividing the numerator by the denominator. *Example:*

$$16/8 = 16 \div 8 = 2$$
$$95/8 = 95 \div 8 = 11\tfrac{7}{8}$$

11-9. *Addition.* You will frequently be required to add fractions. A common use of addition is to find the overall length of an item. The lengths of the sections are often fractions of an inch or whole inches and fractional parts. To find the overall length of an item, you must therefore add the lengths of the various sections. Fractions cannot be added unless they have a common denominator. *Example:* 5/8, 3/8, 6/8. The common denominator should be the least common denominator, which is the lowest denominator into which all the denominators will divide. In the example shown, 8 is the least common denominator.

11-10. To reduce fractions to fractions having a common denominator, reduce each fraction separately. Divide the least common denominator by the denominator of the fraction and multiply the quotient obtained by the numerator of the fraction. The product is the numerator of the reduced fraction, and the denominator is the least common denominator. For example, reduce ⅞ to a fraction having a denominator of 16, assuming that ⅞ is one of a series of fractions (¼, ½, ⅜, etc.) having 16 as a least common denominator. Divide the least common denominator (16) by the denominator of the fraction (8) and multiply the quotient obtained (2) by the numerator of the fraction (7). The product (14) is the numerator of the reduced fraction, and the denominator is the least common denominator (16). Thus:

$$7/8 = 14/16$$

11-11. After you have reduced all of the fractions to fractions having a common denominator, you may then add their numerators and place them over the common denominator. If the answer is an improper fraction, change it to a mixed number, with the fractional portion expressed in its lowest form. When adding mixed numbers, add the whole numbers and fractions separately and the results together.

11-12. A typical example of work requiring the addition of fractions is a shaft consisting of four sections, respectively, $\frac{3}{16}$, $1\frac{7}{16}$, $\frac{3}{4}$, and 4½ inches in length. What is the overall length of this shaft? First, change the fractions to fractions having a common denominator; in this case the least common denominator is 16. Then add the numerators of the new fractions ($\frac{30}{16}$ inch) and change them to a mixed number, with the remaining fraction in its lowest form (1⅞ inches), thus obtaining the overall length of the shaft (6⅞ inches). *Solution:*

$$\begin{array}{rcl} 3/16 \text{ inch} & = & 3/16 \text{ inch} \\ 1\frac{7}{16} \text{ inches} & = & 1\frac{7}{16} \text{ inches} \\ 3/4 \text{ inch} & = & 12/16 \text{ inch} \\ +\ 4\frac{1}{2} \text{ inches} & = & 4\frac{8}{16} \text{ inches} \\ \hline & & 5\frac{30}{16} \\ & & 30/16 = 1\frac{14}{16} = 1\frac{7}{8} + 5 \\ & & = 6\frac{7}{8} \text{ inches} \end{array}$$

11-13. *Subtraction.* You will have many occasions to subtract fractions in your shopwork. For example, layout work frequently requires the subtraction of known dimensions to determine the length of unknown dimensions. Fractions must be reduced to their least common denominator before they can be subtracted. Then their numerators must be subtracted and the difference placed over the common denominator. The resulting fraction should then be reduced to its lowest form. For example, subtract ¾ from ⅞. *Solution:*

$$\begin{array}{r} 7/8 = 7/8 \\ -\ 3/4 = 6/8 \\ \hline 1/8 \end{array}$$

11-14. Subtract mixed numbers by first subtracting the fractions, and then subtracting the whole numbers. If the fractional subtrahend is larger than the fractional minuend, borrow a whole number from the whole minuend, change the number borrowed to a fraction having the same least common denominator as the original fraction, and add it to the original fractional minuend. Next, subtract the fractional subtrahend from the new fractional minuend. Then do the whole numbers subtraction operation. For example, a brace originally 9⅜ inches long is to be shortened $2\,^{13}/_{16}$ inches. How long will the brace be after having been shortened? *Solution:*

$$\begin{array}{r} 9\tfrac{3}{8} = 9\tfrac{6}{16} = 8\tfrac{22}{16}\text{ inches} \\ -\ 2\tfrac{13}{16} = 2\tfrac{13}{16} = 2\tfrac{13}{16}\text{ inches} \\ \hline 6\tfrac{9}{16}\text{ inches} \end{array}$$

11-15. *Multiplication.* You will occasionally use formulas that will require the multiplication of fractions. To multiply two or more fractions together, multiply the numerators together and multiply the denominators together. Place the product of the numerators over the product of the denominators and reduce the resulting fraction to its lowest terms. For example, multiply ¾ by $^{8}/_{9}$ as follows:

$$8/9 \times 3/4 = 24/36 = 2/3$$

11-16. You can obtain the answer more easily (especially if several fractions are to be multiplied) by cancellation. Cancellation is the division of a numerator and a denominator by a number common to both to reduce the size and number of fractions in an equation. To multiply ¾ by $^{8}/_{9}$, using cancellation, arrange the fractions as for normal multiplication. *Example:*

$$3/4 \times 8/9$$

11-17. Now of course you can divide a numerator and denominator by a common number; for example, you can divide both 8 and 4 by 4. Doing this, substitute the quotients obtained for the numbers you have divided and continue cancelling as long as possible. *Example:*

$$2/9 \times 3/1 =$$

Again, you know that you can divide both 9 and 3 by 3. *Example:*

$$2/3 \times 1/1 = 2/3$$

11-18. It is not necessary to rewrite the formula after each cancellation operation, as we have done for clarification purposes with the sample formula. A much simpler and convenient method is to strike through the number being divided and to write the quotient above or below the old number. *Example:*

$$5/8 \times 16/18 \times 21/25 \times 15/16 = \overset{1}{\cancel{5}}/8 \times \overset{1}{\cancel{16}}/\underset{\underset{2}{\cancel{6}}}{\cancel{18}}$$

$$\times\ \overset{7}{\cancel{21}}/\underset{\underset{1}{\cancel{5}}}{\cancel{25}} \times \overset{\overset{1}{3}}{\cancel{15}}/\underset{1}{\cancel{16}} = 7/16$$

11-19. *Division.* You divide fractions by inverting the divisor, changing the division sign to a multiplication sign, and multiplying. When a fraction is inverted, the numerator becomes the denominator, and the denominator becomes the numerator. For example, divide $^{5}/_{7}$ by ¾.

$$5/7 \div 3/4 =$$

Invert the divisor, ¾, which then becomes $^{4}/_{3}$. Change the division sign to a multiplication sign and solve.

$$5/7 \times 4/3 = 20/21$$

11-20. **Decimals.** A machinist needs a thorough knowledge of decimals because the dimensions of parts are often expressed as decimals. Objects having close tolerances are dimensioned with decimals rather than fractions because more accurate measurements are possible with decimals. The smallest fraction commonly used is $^{1}/_{64}$ inch; smaller fractions are difficult to measure by ordinary means. Measurements of $^{1}/_{1000}$ inch, or smaller, are easily made if decimals are used.

11-21. You not only must be able to measure objects dimensioned with decimals but also must be able to solve mathematical problems in which decimals appear, such as gear calculations—to determine the amount of materials to remove and to determine the depth of cut to take—and to handle other problems too numerous to list. In fact, you must add decimals in making measurements with a micrometer.

11-22. *Reading decimals.* A decimal is actually a fraction having a denominator of 10, or a power of 10 (100, 1000, 10000, etc.) that is written in a different form. For example, the fraction $\frac{1}{10}$ is written as 0.1 in decimal form; $\frac{1}{100}$ is 0.01; and $\frac{35}{1000}$ is 0.035. The period in front of the number is called a decimal point and shows that the number is a decimal and not a whole number. A decimal is a fraction whose numerator is the number that follows the decimal point and whose denominator is a 1 followed by as many zeros as there are numbers following the decimal point. For example, 0.7 means $\frac{7}{10}$, 0.39 means $\frac{39}{100}$, and 0.035 means $\frac{35}{1000}$. You can see, then, that each place to the right of the decimal point represents a zero in the denominator; thus, one place, one zero; two places, two zeros; etc.

11-23. The number of places to the right of the decimal point determines the reading or meaning of the decimal. The places have definite meaning, as shown in the following listing:

0.0	=	tenths
0.00	=	hundredths
0.000	=	thousandths
0.0000	=	ten thousandths
0.00000	=	hundred thousandths
0.000000	=	millionths

11-24. If a number extends two places to the right of the decimal point, it is read as hundredths; three places, as thousandths; etc. For example, 0.020 extends three places to the right and should be read as twenty thousandths. Again, 0.125 would be read as one hundred twenty-five thousandths. Zeros may be added to, or removed from, the end of a decimal without changing its value. For example, 0.05 = 0.0500 = 0.05000.

11-25. *Addition.* You will be required to add decimals frequently. The measurement obtained with a micrometer is the sum of the decimal readings taken from the barrel and thimble of the micrometer. To add decimals, position the numbers to be added in a column with each succeeding decimal point directly below the one above. Add zeros to decimals whenever a decimal has fewer places than another to help avoid errors. Place the decimal point in the sum directly below the other decimal points. For example: What would the overall length of an object be if it consists of sections that are 1.255 inches, 0.875 inch, 0.375 inch, 4.5 inches, and 3.25 inches in length?

11-26. To solve this problem, arrange the numbers in a column so that the decimals are in alinement, add zeros to make the decimals even in length, and then add the column as follows:

$$\begin{array}{r} 1.255 \\ 0.875 \\ 0.375 \\ 4.500 \\ +\ 3.250 \\ \hline 10.255 \end{array}$$

11-27. The diameter of a shaft measured with a micrometer results in a reading of 0.525 inch on the barrel and 0.019 inch on the thimble. What is the diameter of the shaft? Add the two decimal readings to obtain the answer as follows:

$$\begin{array}{r} 0.525 \\ +\ 0.019 \\ \hline 0.544 \end{array}$$

11-28. *Subtraction.* To subtract decimals, place the decimal points in a column and then subtract. For example: A shaft is being machined in a lathe that must have a diameter of 0.618 inch when completed. The micrometer reading shows the diameter to be 0.88 inch. How much more material must be removed from the diameter of the shaft to obtain the desired dimension? *Solution:* Subtract the desired dimension from the micrometer reading, as follows:

$$\begin{array}{r} 0.880 \\ -\ 0.618 \\ \hline 0.262 \end{array}$$

11-29. *Multiplication.* Multiply decimals as if the numbers were whole numbers, then place the decimal point in the product. You can determine the location of the decimal point by adding the number of decimal places in the multiplicand and multiplier, and, beginning at the right, counting off the same number of decimal places in the product. For example, let's multiply 43.286 by 6.04. First multiply the numbers, disregarding the decimal point, as follows:

$$\begin{array}{r} 43.286 \\ 6.04 \\ \hline 173144 \\ 2597160 \\ \hline 26144744 \end{array}$$

Since the multiplicand has three decimal points and the multiplier has two, the product will have five. Starting at the right, count off five decimal places and insert the decimal point (261.44744).

11-30. To divide decimals, the divisor must be a whole number. If the divisor is a decimal, move the decimal point to the right the number of places necessary to change the decimal to a whole number,

Chart 7
Decimal Equivalent

DECIMAL EQUIVALENTS									
Fraction	16th	32nd	64th	Decimal	Fraction	16th	32nd	64th	Decimal
			1	.015625				33	.515625
		1	2	.03125			17	34	.53125
			3	.046875				35	.546875
	1	2	4	.0625		9	18	36	.5625
			5	.078125				37	.578125
		3	6	09375			19	38	.59375
			7	.109375				39	.609375
1/8	2	4	8	.125	5/8	10	20	40	.625
			9	.140625				41	.640625
		5	10	.15625			21	42	.65625
			11	.171875				43	.671875
	3	6	12	.1875		11	22	44	.6875
			13	.203125				45	.703125
		7	14	.21875			23	46	.71875
			15	.234375				47	.734375
1/4	4	8	16	.250	3/4	12	24	48	.750
			17	.265625				49	.765625
		9	18	.28125			25	50	.78125
			19	.296875				51	.796875
	5	10	20	.3125		13	26	52	.8125
			21	.328125				53	.828125
		11	22	.34375			27	54	.84375
			23	.359375				55	.859375
3/8	6	12	24	.375	7/8	14	28	56	.875
			25	.390625				57	.890625
		13	26	.40625			29	58	.90625
			27	.421875				59	.921875
	7	14	28	.4375		15	30	60	.9375
			29	.453125				61	.953125
		15	30	.46875			31	62	.96875
			31	.484375				63	.984375
1/2	8	16	32	.500	1	16	32	64	1.000

53--129

then move the decimal point of the dividend to the right the same number of places. If the dividend does not have enough decimal places, add as many zeros as necessary. Place the decimal point in the quotient directly over the decimal point in the dividend. For example, let's divide 0.78 by 0.964. First arrange the numbers as for simple division, as follows:

$$0.964 \,/\, \overline{0.78}$$

Next, change the divisor to a whole number by moving the decimal point to the right, in this case three places, as follows:

$$964 \,/\, \overline{0.78}$$

After this, move the decimal point in the dividend to the right three places. Since there are only two places in the example, you must add a zero to do this as follows:

$$964 \,/\, \overline{780.}$$

Now, place the decimal point in the quotient directly over the decimal point in the dividend, as follows:

$$964 \,/\, \overline{780.}$$

Then divide. If the quotient is not a whole number, add zeros to the dividend and continue the division to as many decimal places as desired, as shown:

```
          0.809
  964 / 780.000
        771 2
          8 800
          8 676
            124
```

11-31. *Changing decimals to fractions.* You can change a decimal to a common fraction by writing the decimal in its fractional form and reducing it to its lowest terms. For example, change 0.5, 0.25, 0.125, and 0.375 to fractions, as follows:

$$\begin{aligned} 0.5 &= 5/10 &= 1/2 \\ 0.25 &= 25/100 &= 1/4 \\ 0.125 &= 125/1000 &= 1/8 \\ 0.375 &= 375/1000 &= 3/8 \end{aligned}$$

11-32. A more convenient means of changing decimals to fractions is to use a decimal equivalent table, such as the one shown in chart 7. To use this table, locate the decimal being changed and the fractional equivalent shown next to it. For example, to find the fractional equivalent of 0.4375, locate the decimal on the table. The fractional equivalent, $7/16$, appears next to it.

11-33. *Changing fractions to decimals.* You may not have a table of decimal equivalents at hand, or the fraction you wish to change to a decimal may not appear on the table; for example, $1/40$. What should you do in such a situation as this? Since the line in a fraction means "divided by," in reality a fraction which says that the numerator is divided by the denominator. For example, $1/4$ really means 1 divided by 4. To change a fraction to a decimal, you simply do the indicated division. To change $1/40$ to a decimal, for example, divide 1 by 40, as follows:

$$40 \,/\, \overline{1}$$

Since 40 is greater than 1, add a decimal point after the numeral 1 and as many zeros as necessary and divide, as shown:

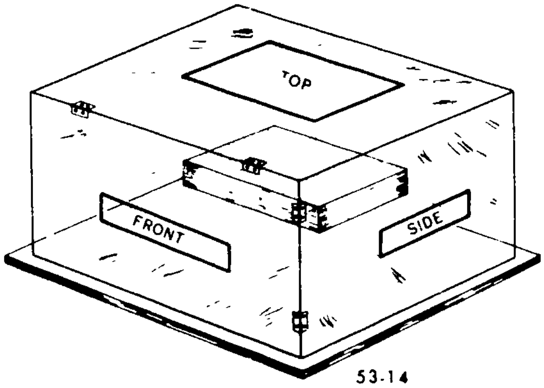
```
       0.025
  40 / 1.000
         80
         200
         200
```

Thus, $1/40$ equals 0.025.

12. Blueprint Reading and Mechanical Drawings

12-1. You will work constantly with blueprints and drawings. Metal parts, whether simple or complex, are machined to exact dimensions. It would be difficult for a design engineer to describe, verbally or in writing, these dimensions. He depends on a blueprint or a drawing to do the job for him. The purpose of a blueprint or drawing is to indicate the form and size of an object. This is done in such a way that the person using the drawing can clearly understand what the engineer or designer had in mind. You will be required to work from blueprints and drawings and to interpret them correctly. You may even be required at times to design parts or to make your own working drawings. You are not required to be a draftsman. However, the more you know about blueprints and mechanical drawings, the easier it will be for you to read and interpret them.

12-2. **Blueprints.** You undoubtedly have heard the term "blueprint" many times, but you may not know exactly what it is, how it differs from a drawing, or why it is used. A blueprint is a reproduction or print of a drawing. Blueprints are used because the time required to make several copies of a drawing by hand prohibits their use in the quantities needed. Blueprints are widely used because hundreds of copies of a drawing can be



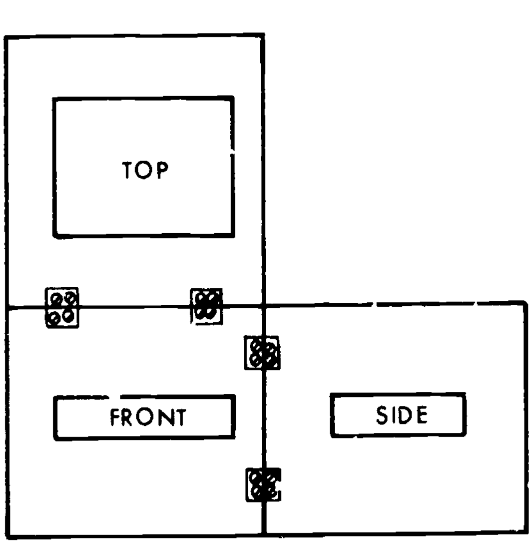
53-14

Figure 14. Projected view of an object.

made in a short time. Because of the blue color, with white lines, the prints are called blueprints. The term "blueprint" is commonly used when referring to reproductions which may also be brown with white lines, black with white lines, or white with blue lines. This depends upon the type of printing paper and chemicals used.

12-3. The storage of an enormous number of blueprints in the Air Force has made it necessary to put many of them on microfilm. There are many types of engineering drawings. If you need one of these in your work, your shop foreman will obtain it for you. Technical orders contain drawings which show how to repair all types of Air Force equipment. You will use this type of drawing more frequently than any other.

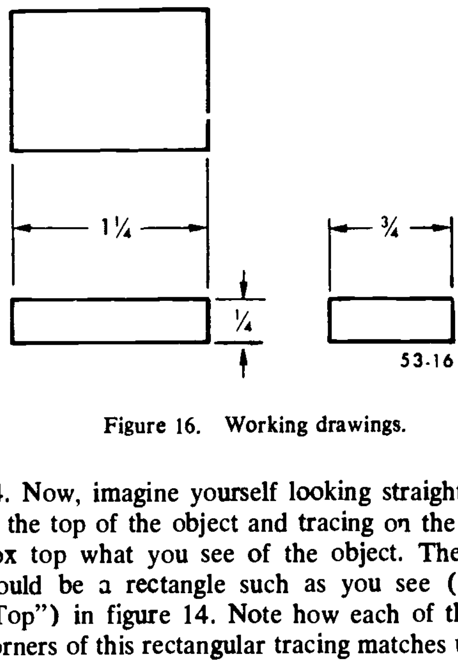
12-4. **Mechanical Drawings.** A mechanical drawing is a drawing made with the aid of instruments. It differs from an artist's drawing in that no attempt is usually made to give the drawing perspective. Perspective is a part of the appearance an object presents to the eye. An artist's drawing attempts to represent an object exactly as the eye sees it, with portions near the viewer appearing larger than portions farther away. In mechanical drawing the object is drawn in such a way as to leave no doubt as to its exact size and shape, even though the drawing may seem to be out of proportion. The most common type of mechanical drawing is the orthographic projection, sometimes known as a three-view drawing.



53-15

Figure 15. Orthographic projection onto paper.

12-5. *Orthographic projection.* For machine shop purposes, the best way to draw many objects is to project them on paper in some combination of front, top, and side views. This method of showing several faces of an object at the same time is called orthographic projection. To understand this three-view type of projection, imagine that the object is resting inside a rectangular-shaped box having transparent plastic sides, as you see in figure



53-16

Figure 16. Working drawings.

14. Now, imagine yourself looking straight down at the top of the object and tracing on the plastic box top what you see of the object. The result would be a rectangle such as you see (labeled "Top") in figure 14. Note how each of the four corners of this rectangular tracing matches up with the corners of the object as shown by the vertical dashes known as projection lines. Note, too, how the front and right-side tracings appear as projections on the plastic box of what you would see if you were looking directly at the object from those particular angles.

12-6. To understand view alinement, imagine swinging the hinged sides of the plastic box around, flattening them out into a single plane as though you were laying them flat on drawing paper. Note in figure 15, that the top view alines directly over the front view, and the side view appears directly to the right of and in line with the front view. This is an orthographic projection, showing the exact size and shape of the object.

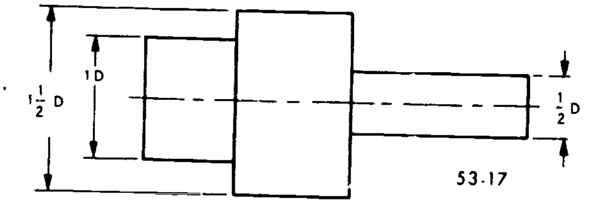


Figure 17. One-view drawing.

When dimensions are added, as in figure 16, you have a working drawing. You can work from an orthographic projection much more easily than from a picture of the object as your eye sees it.

12-7. *Freehand sketching.* The importance of freehand sketching cannot be overestimated. You use freehand sketches to express original ideas. You may have a general idea of what you want to make and what its function will be when it is completed, but you need some other people's ideas as to the design of the part. With your sketch you can show what your general plans are. As your plan takes a more definite shape, you may erase and change the design as often as necessary until you have a satisfactorily designed part. Much of the work done in the machine shop is planned with the aid of freehand sketches.

12-8. **Interpretation of Drawings.** The ability to "read" (understand) a drawing becomes easier as you become more experienced. The presentation views ' types of lines used, the indications of holes a... threads, the sectional views, and the dimensions all help you in the correct interpretation of a d ving

12-9. *Presentation of views.* The number of views and the position of the views aid you in reading a drawing. While drawings usually present three views of the object, the number of views necessary will depend upon the nature and shape of the object. For instance, no matter from what angle you view a ball, its shape is still the same round sphere; therefore, one view is sufficient to show the size, and the sperical shape can be explained by a short note. Cylindrical work is often represented by one view, as you can see in figure 17. Here, the centerline running through the middle of the piece and the letter D (for diameter) in the dimensions indicate that the piece is cylindrical in shape. Since a side view consisting of three circles would not show anything not already known, its omission makes the drawing easier to read. However, any extra machining operations, such as drilled holes, keyways, threads, and counterbores, could require a second, or even a third, view for describing accurately their size and location. Figure 18 shows a drilled bushing. The bushing requires both a top and front view if you are to clearly establish the location and nature of its central hole. It has no other features that would require a side view.

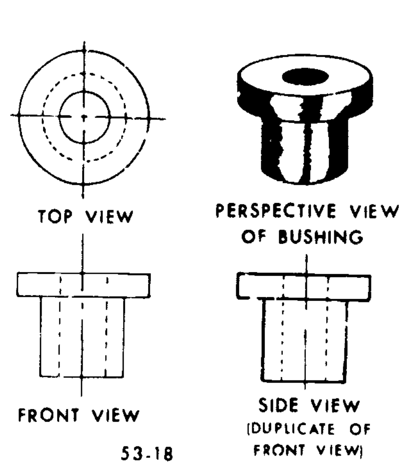


Figure 18. Two-view drawing.

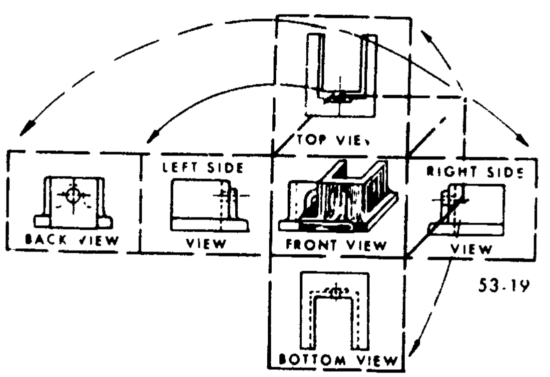


Figure 19. Relationship of views.

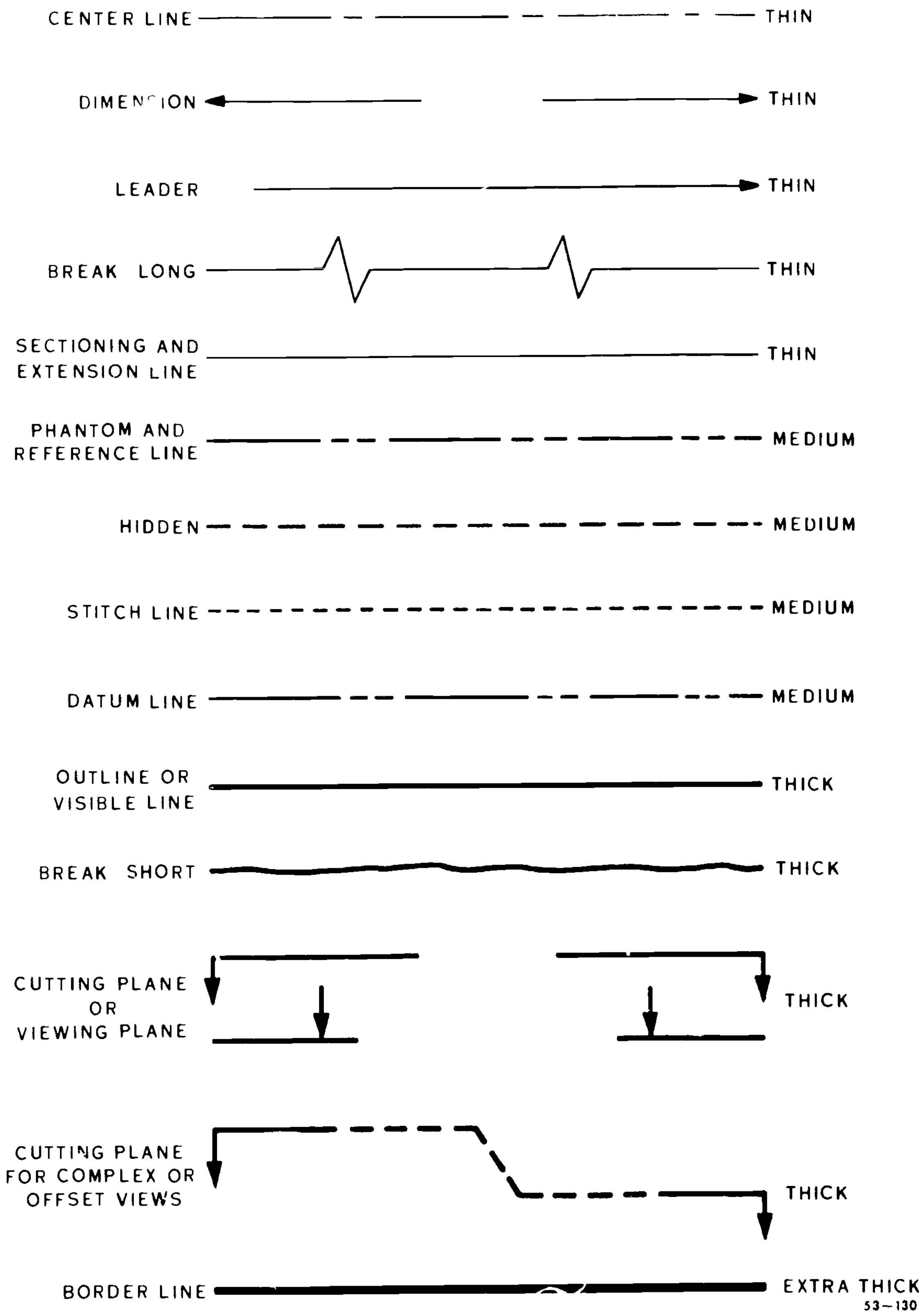
12-10. If the object has a complicated or irregular shape, more than three views may be needed. The number of views needed will depend upon how much detail the machinist needs. Detail in any one view should be kept to a minimum to avoid cluttering up the view and making it difficult to read.

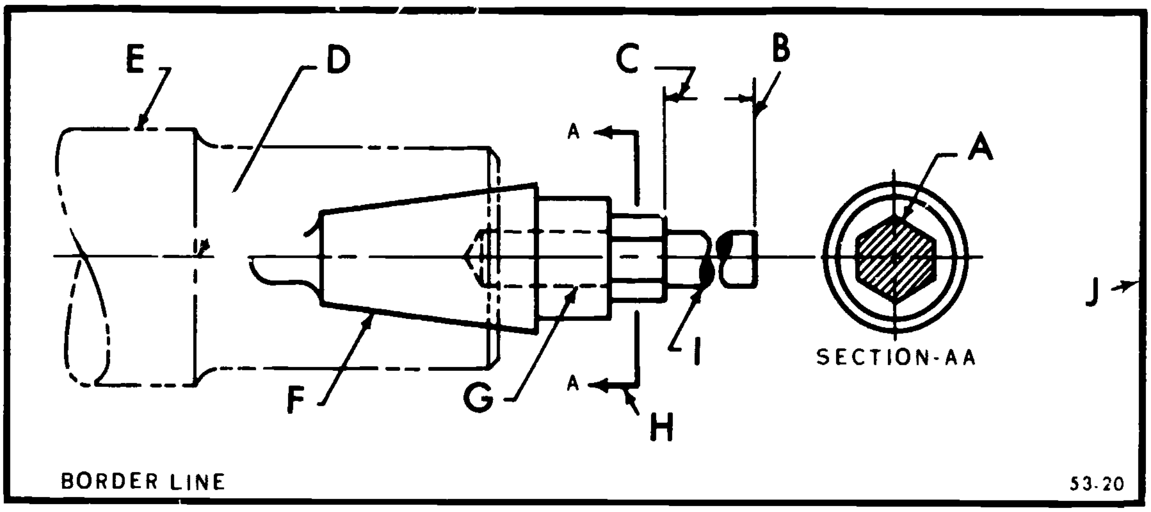
12-11. Views are drawn in the correct relationship with each other. The top view is placed above the front view, and the right-side view is placed to the right of the front view, as shown in figure 19. A view placed out of position is confusing. There would be much repetition of information if all six views were used to describe an object. A multiview drawing should not contain more views than necessary to describe the object fully. In presenting views the following principles are observed:

a. Views showing essential contours are selected.

b. Views with the least amount of invisible lines are preferred.

Chart 8
Line Thickness, Composition, and Meaning





A. SECTIONING LINES
B. EXTENSION LINE/LEADER LINE
C. DIMENSION LINE
D. CENTER LINE
E. PHANTOM LINE
F. OUTLINE
G. HIDDEN LINE
H. CUTTING PLANE LINE
I. BREAK LINE
J. BORDER LINE

Figure 20. Use of various lines.

c. Right-side views are preferred to left-side views, unless the left-side view conveys more information.

d. The top view is preferred to the bottom view, unless the bottom view conveys more information.

e. In presenting views the available space should be considered.

f. The principal view is the one that shows the characteristic contour of the object. It is good practice to use this view as the front view on a drawing, regardless of the natural front of the object.

12-12. *Types of lines and uses.* The types of lines used to draw the views of an object help you to interpret a drawing. These lines are made in definite, standard ways. The relative thickness of the line (fine, medium, thick) and the composition (broken, solid, dashes, etc.), as shown in chart 8, signify various meanings, which must be understood if you are to correctly read a drawing. The composition and use of various lines are as follows:

a. Centerlines consist of long and short dashes, alternately and evenly spaced with a long dash at each end and short dashes at points of intersection of centerlines. Very short centerlines, as in figure 20, may be broken if there is no confusion with other lines. Centerlines are also used to indicate the travel of a center.

b. Dimension lines terminate in arrowheads at each end, as shown in chart 8 and figure 20. They are broken where the dimension is inserted.

c. Leader lines indicate a part or area to which a number, note, or other reference applies. Usually the leader lines terminate in an arrowhead, as in chart 8.

d. Break lines are used when an object is uniform in shape, such as a pipe or shaft, and the length prevents its being shown completely on a drawing. A portion is removed from its midsection, as shown in figures 20, 21 and 22, to enable both ends of the object to be seen. Short breaks are indicated by solid, freehand lines, as in figure 21. For long breaks, full, ruled lines with freehand zigzags are used, as in figure 22. Shafts, rods, and tubes have the ends of the break drawn as indicated in figure 21.

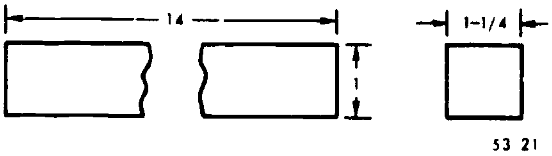


Figure 21. Short break lines.

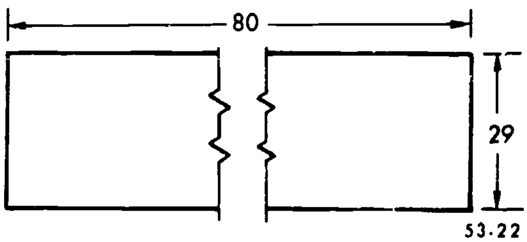


Figure 22. Long break lines.

e. Phantom lines are used to indicate alternate positions of parts of an object, repeated detail or the locations of absent parts, as in figure 20. They are made by alternating one long and two evenly spaced short dashes with a long dash at each end.

f. Sectioning lines are used to indicate the exposed surfaces of an object in a sectional view, as in figure 20. They are usually full, thin lines but may vary with the type of material shown

g. Extension lines are used to indicate the extent of a dimension, as in figure 20. They do not touch the outline of the object but terminate within one-sixteenth of an inch from it.

h. Hidden lines consist of short, evenly spaced dashes to show hidden features of an object, as shown in figure 20. They always begin with a dash in contact with the line from which they start, except when a dash would form a continuation of a full line.

i. Stitch lines, shown in chart 8, are used to indicate the stitching or sewing lines on an article. They consist of a series of very short, evenly spaced dashes, about one-half the length of the dashes used for hidden lines.

j. Outline or *visible* lines are solid, thick lines used to represent all visible lines on the object, as in figure 20.

k. Datum lines, as in chart 8, are made by alternating one long and two short evenly spaced dashes of medium thickness. They are used to show surfaces not present in the drawing from which positions are located.

l. Cutting plane lines are used to indicate a plane in which a sectional view is taken. *Viewing plane* lines, as shown in chart 8, are used to indicate the plane from which a surface is viewed.

m. Border lines are extra heavy lines used to frame or inclose the entire drawing and to give the drawing a "finished" appearance, as shown in figure 20.

12-13. *Dimensions.* To manufacture an object, the size of the object and all of its parts must be given and understood. You must be able to correctly interpret the dimensions on a drawing; otherwise, the drawing will only show the shape of the object, and the full meaning will be lost. An understanding of the various types of dimensions and the methods used to represent them are essential in interpreting drawings.

12-14. There are three general groupings of dimensions, as shown in figure 23. The three types are as follows:

a. Detail or size dimensions give size of diameters, widths, lengths and heights of the object.

b. Position or location dimensions specify the location of features of an object with respect to each other, such as the distance between a surface and a center, or between two surfaces.

c. Overall dimensions give the entire length, or height, or width of an object. They are the total of

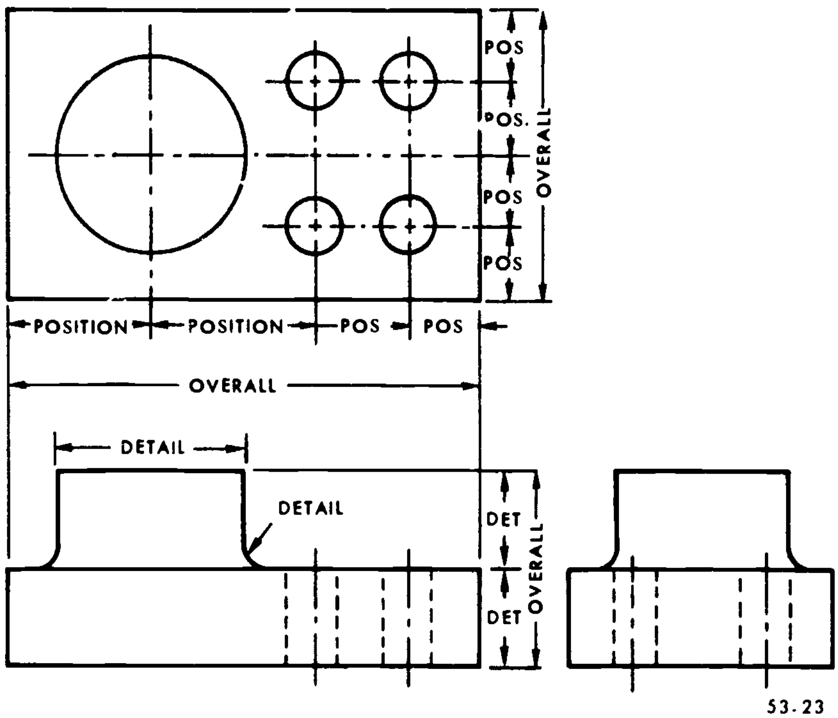


Figure 23. Types of dimensions.

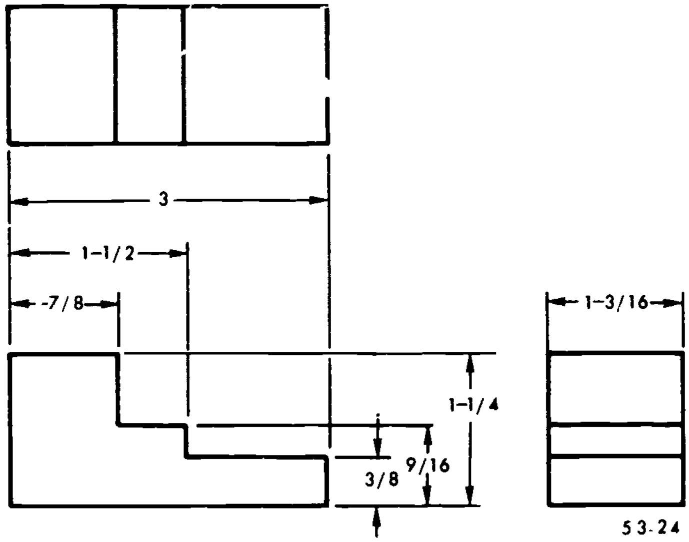


Figure 24. Dimensioning methods.

all smaller dimensions in any one direction and are usually located on the outside of detail and position dimensions.

12-15. The following rules are used as guides when drawings are dimensioned; however, objects frequently have shapes, sizes, and complicated portions which may require deviating from these rules:

a. Dimensions applying to related views are placed between them.

b. Dimensions are not placed directly on the object.

c. Dimensions that apply to more than one view are placed on the view which most clearly illustrates the features being dimensioned.

d. Hidden edges are not used for dimensioning.

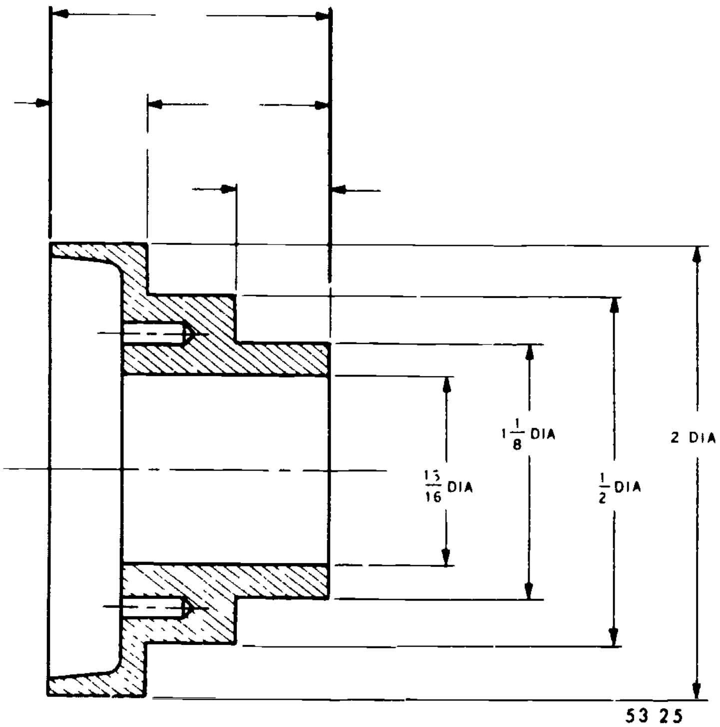


Figure 25. Dimensioning methods shown in profile.

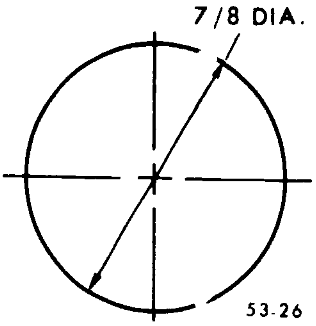


Figure 26. Dimensioning large diameters.

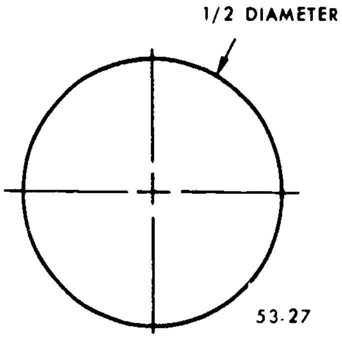


Figure 27. Dimensionir.g small diameters.

e. All needed dimensions are given withcut unnecessary duplication.

f. Small dimensions are placed near the object and increasingly larger ones are placed farther away, as shown in figure 24.

g. The symbol or abbreviation for inches is omitted from dimensions. The measurements are understood to be in inches unless otherwise stated.

h. Dimensions and lettering are positioned so that they can be read from the bottom of the drawing.

12-16. Figure 25 illustrates the method used to dimension diameters shown in profile. Note the use of the abbreviation of diameter. When diameters appear on a drawing as circles, the dimensioning methods shown in figure 26 and 27 are used. Figure 27 shows the method used for smaller diameters. Centerlines are used for both types.

12-17. Radii are dimensioned by drawing a radial dimension line through the origin of the radius. On small radii, however. the dimension line may be drawn on the side opposite the center. The letter R always follows the dimensions of the radius. Figure 28 illustrates the methods used.

12-18. Examples of the dimensioning of angles are shown in figure 29. The dimension of the angle

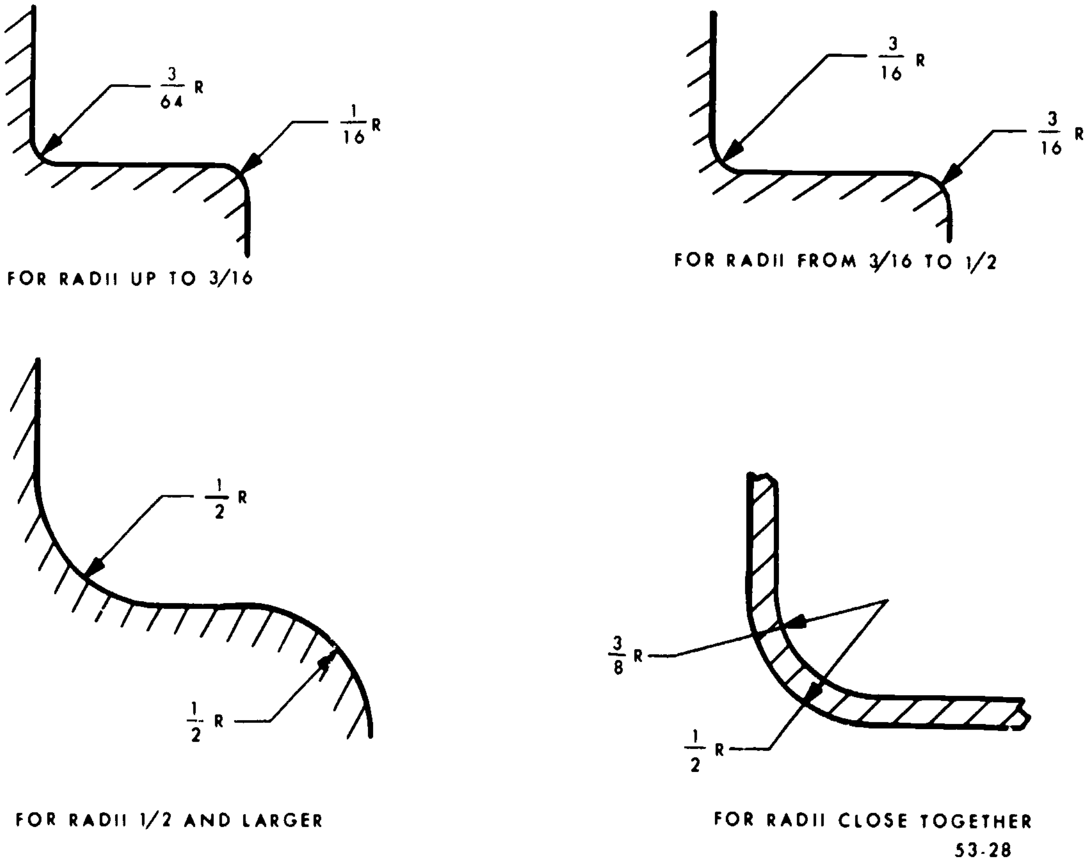


Figure 28. Dimensioning arcs and radii.

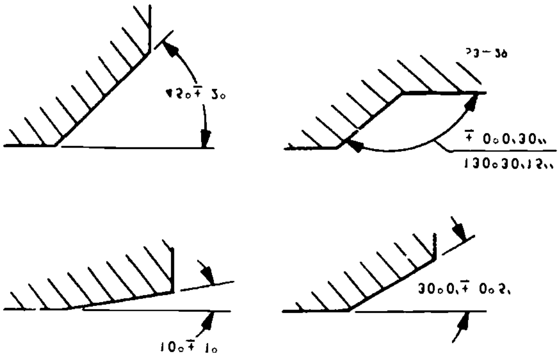


Figure 29. Dimensioning angles and chamfers.

is placed inside the extension lines whenever the spacing permits. Figure 30 illustrates the recommended dimensioning of chamfers and the optional method used for 45° chamfers.

12-19. *Limits, tolerance, and allowance.* To make parts interchangeable and to eliminate working to unnecessarily close measurements, the designer must determine acceptable variations in size. The drawing, tells you what the "allowable errors" are. These permitted differences in size are known as limits, tolerance, and allowance. You must understand what these terms mean and how they are shown in order to correctly interpret a drawing.

12-20. Limits are the extreme dimensions to which an object may be made and still be acceptable. You can find the maximum limit by adding the amount a piece may be made larger than specified (indicated by a plus sign in front of the permitted variation). You can find the minimum limit by subtracting the amount a piece may be made smaller than specified (indicated by a minus sign

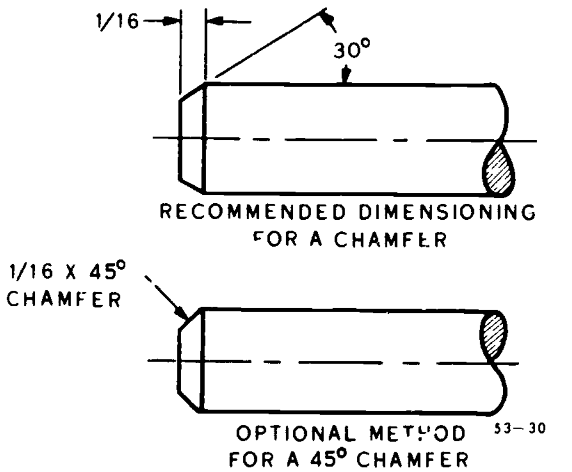


Figure 30. Methods of dimensioning chamfers.

in front of the permitted variation). Figure 31 shows how the limits of a piece are determined.

12-21. Tolerance is the total amount of variation permitted. You can determine it by adding the amount a part may be made larger to the amount it may be made smaller. In figure 31 the tolerance is 0.005 inch. When a piece may be made only larger or only smaller, it has a unilateral tolerance, as shown in figure 32. When the piece may be made either larger or smaller, it has a bilateral tolerance. Figure 33 illustrates how bilateral tolerances are shown when they are unequal. When they are equal, they are often shown as illustrated in figure 34.

12-22. Allowance is the intentional difference between mating parts. If the mating parts have clearance between them, the allowance is positive. When the part is larger than the hole it goes in, the allowance is negative. The term "allowance" is also applied to excess stock deliberately left on a piece of material to be further machined to the final size. For example, a part that is to be heat

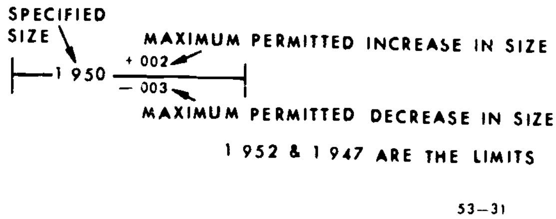


Figure 31. Limits.

treated is usually made oversize and is ground to the final size after heat treating. This excess stock is called grinding allowance.

12-23. *Holes and threads.* You will be required to drill, countersink, counterbore, and tap holes, and to machine threads. We will explain these operations later in the course. To do these operations, you must be able to "read" the drawing and recognize these operations by their illustrations.

12-24. The following are some of the guides used for dimensioning holes:

a. The depth of blind holes is given. The depth does not include the drill point.

b. Countersunk holes have the included angle and diameter of the countersink given.

c. The abbreviations C'Drill (counterdrill), C'Bore (counterbore), and C'Sink (countersink) are used along with the size of the hole. Figure 35 shows the methods used to represent various types of holes and the methods used to dimension them.

12-25. Threaded holes and shafts may be illustrated as they actually appear, as shown in figure 36, part A; by the conventional method, part B; or

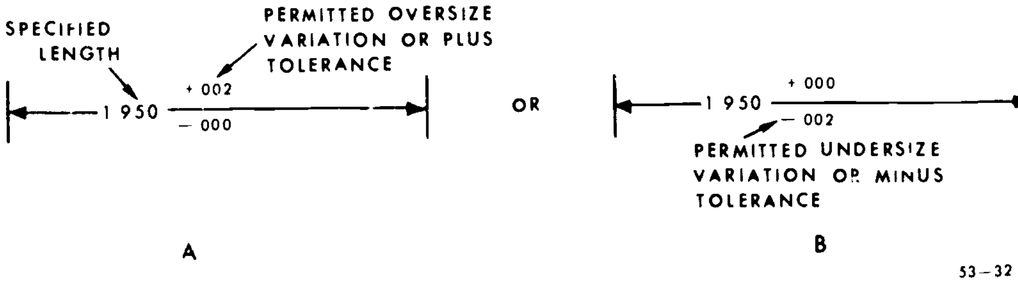


Figure 32. Unilateral tolerance.

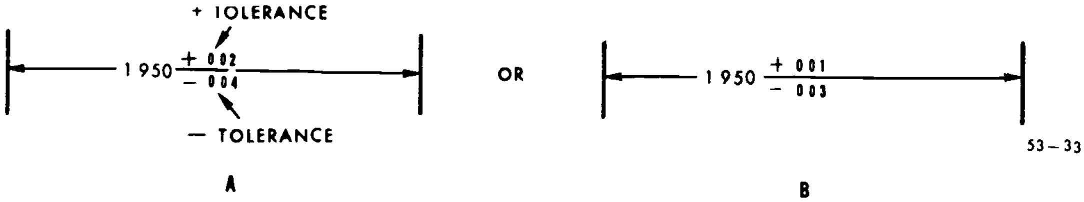


Figure 33. Unequal bilateral tolerance.

by the simplified method, part C. In any case, the diameter of the thread, the number of threads per inch, the thread series, and the class of fit are given, as illustrated in figure 36, to the right of part C. The abbreviation LH is used following the thread designation for left-hand threads only, for example, 1/2-13-NC-3-LH.

13. Geometric Construction

13-1. To make an accurate shop drawing of a part or to lay out work, you must know how to construct geometric figures. In this section we will explain how to construct and divide lines, angles, and arcs, using only a pencil, a straightedge, and dividers. You will then be able to apply this knowledge in the construction of any geometric figures that you may need in drawing and in layout work. Dividers are illustrated in figure 37.

13-2. **Bisecting Lines.** To bisect a line (divide into two equal parts), use the ends of the line, points A and B in figure 38, as pivot points for the dividers and draw arcs greater than one-half of line AB that intersect on each side of the line. A line drawn between the points of intersection of the arcs, points C and D, bisects the line AB and also is perpendicular to it.

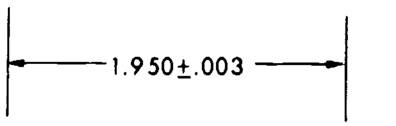


Figure 34. Equal bilateral tolerance.

13-3. **Bisecting Arcs.** To bisect an arc, use the ends of the arc, points A and B in figure 39, as pivot points, and using a radius greater than one-half of arc AB, draw arcs that intersect at C and D. A line drawn between points C and D bisects arc AB.

13-4. **Bisecting Angles.** Using any desired radius and point A in figure 40 as a pivot point, draw arc ED. With the same divider setting, and points E and D as pivot points, draw arcs that intersect at point O. A line drawn from point O to point A bisects angle CAB.

13-5. **Trisecting Right Angles.** To trisect a right angle (divide into three equal angles), use point B in figure 41 as a pivot point, and any desired divider setting, and draw arc DE. Then, using the same setting and point D as the pivot point, draw an arc that intersects arc DE at point F. Using the same setting and point E as the pivot point, locate point G. Lines drawn from points F and G to point B trisect angle ABC. It must be remembered that this method works only on a right angle.

13-6. **Erecting a Perpendicular from a Point off a Line.** Using point P as a center and a radius greater than the distance from P to line AB, as shown in figure 42,A and B, draw an arc which intersects the line at points C and D. With points C and D as the pivot points, draw arcs which intersect at point E. You may use a radius longer than distance PF, as in figure 42,A, or shorter than PF, as in figure 42,B. A line drawn from point E through point P and extended to line AB, as in fig-

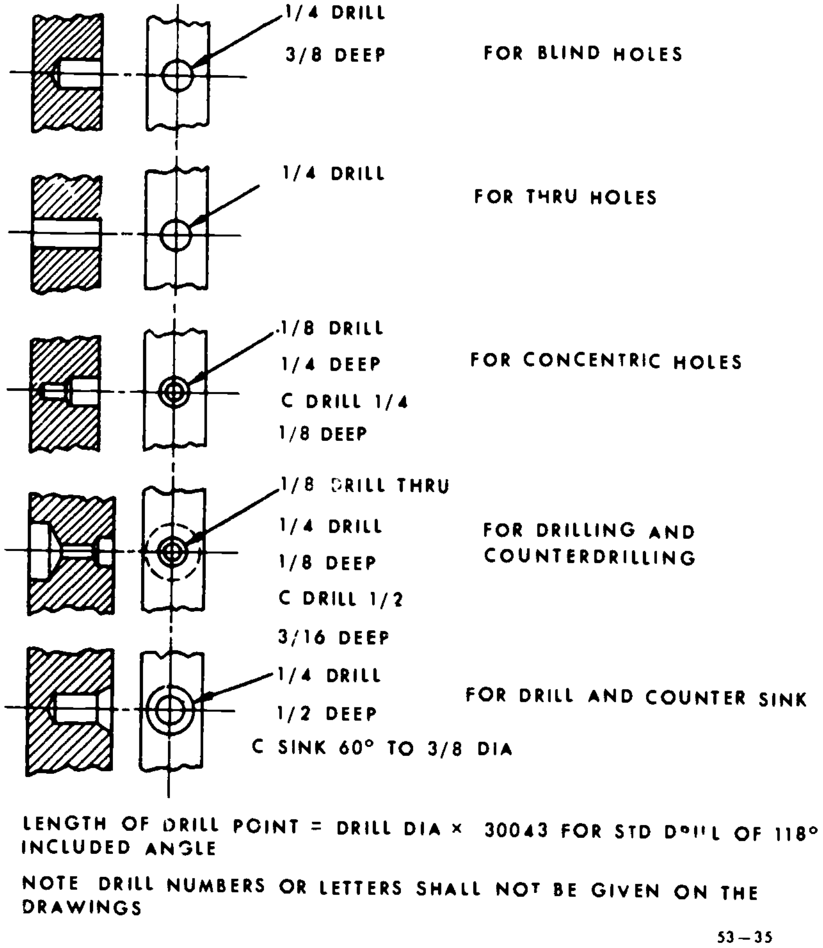


Figure 35. Dimensioning holes.

ure 42,A, or drawn from point P to point E, as in figure 42,B, is perpendicular to line AB.

13-7. **Erecting a Perpendicular from a Point or a Line.** Using point P, as shown in figure 43, as a pivot point, and with the divio s set at any desired radius, draw arcs that intersect line AB at points C and D. Using points C and D as pivot points, and with the dividers set at a radius greater than one-half of distance CD, draw arcs that intersect at point E. A line drawn from point E to point P is perpendicular to line AB.

13-8. **Dividing a Line into Equal Parts.** You can divide a line, such as AB in figure 44, into any desired number of equal divisions by drawing a line, such as BC, at any angle from point B. With

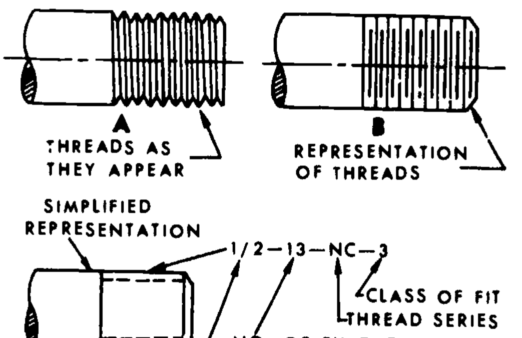


Figure 36. Dimensioning threads.

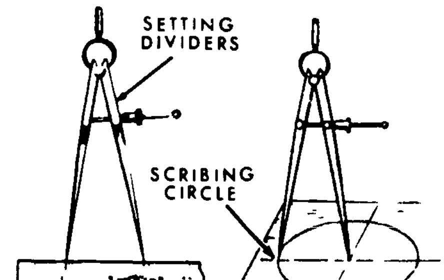


Figure 37. Using dividers.

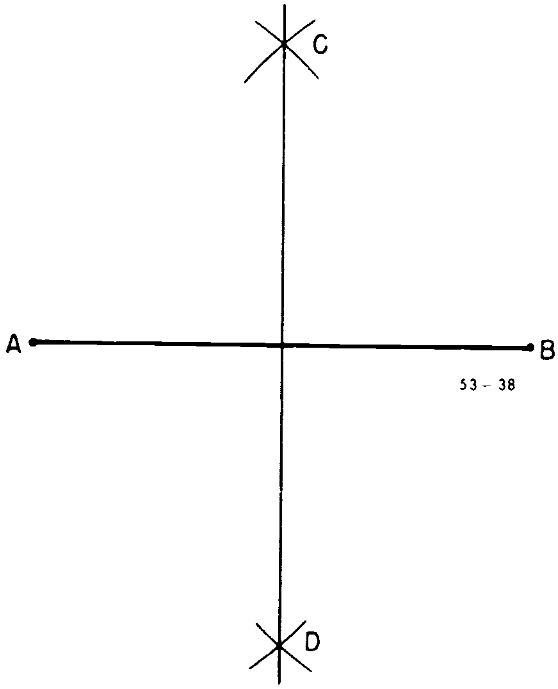


Figure 38. Bisecting a line.

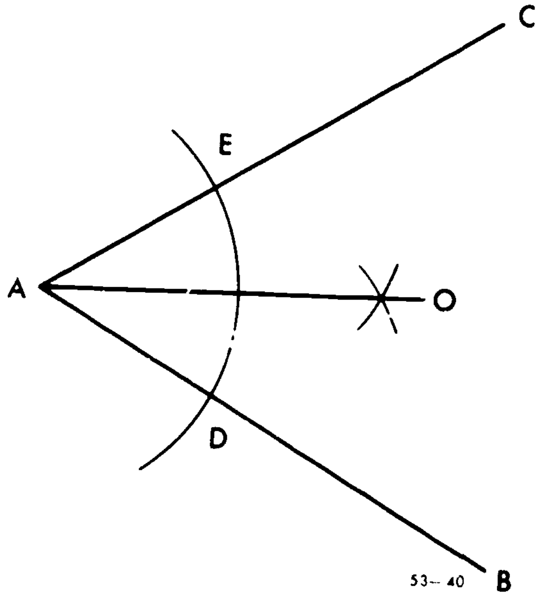


Figure 40. Bisecting an angle.

the dividers set at any desired setting, step off the desired number of divisions on line BC. Connect the last division mark, point C, and point A. Lines drawn parallel to line AC and through each of the division marks on line BC divide line AB into the desired number of equal divisions.

13-9. **Constructing a Line Parallel to a Line.** With the dividers set at the desired distance from line AB, figure 45, and using any two points on line AB as pivot points 1 and 2, construct arcs (the farther apart points 1 and 2 are, the more accurate the results will be). Line CD is drawn tangent to the arcs and is parallel to line AB.

13-10. **Constructing an Arc Tangent to Intersecting Lines.** With the dividers set at the desired radius, construct parallel lines to intersecting lines AB and BC, as shown in figure 46. Using point E as a pivot, the desired arc is drawn, which is tangent to the original lines.

13-11. **Constructing an Arc Tangent to a Straight Line and a Circle.** With the dividers set to the length of the radius of the circle, as in AB in figure 47, plus the length of the desired radius (BF), construct arc EF, using the center of the circle as a pivot point. With the dividers set to the desired radius (BF), construct a line parallel to line CD and extend it to intersect arc EF. Using point E as a pivot point, and with the dividers set to the desired radius (BF), construct the arc tangent to the line and the circle.

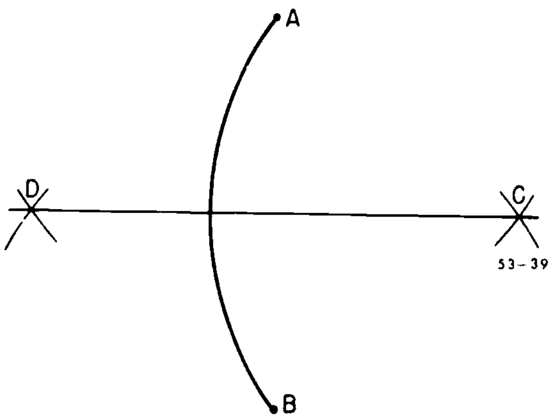


Figure 39. Bisecting an arc.

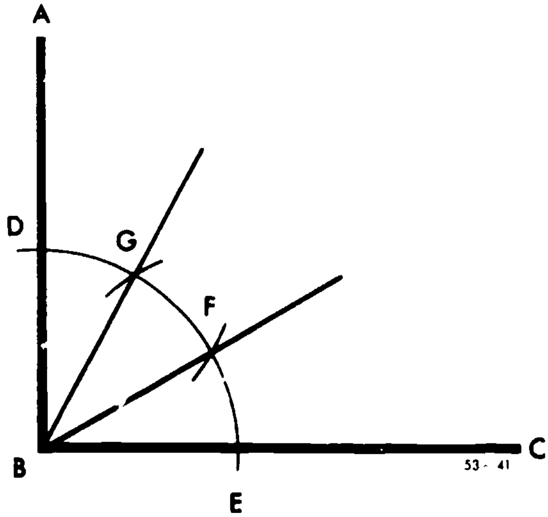


Figure 41. Trisecting a right angle.

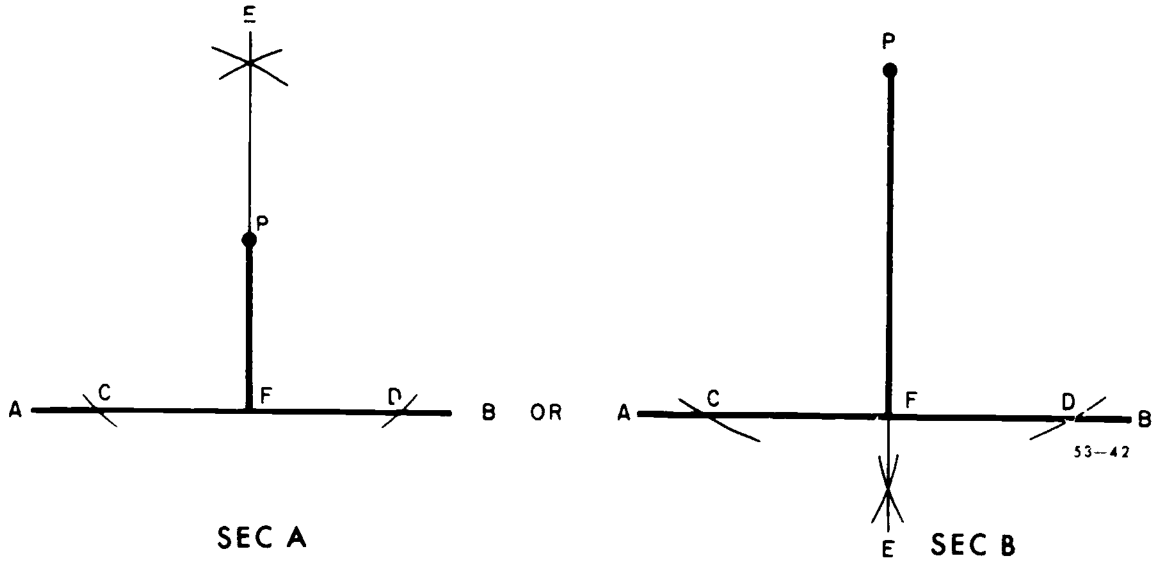


Figure 42. Erecting a perpendicular through a point off a line.

13-12. **Constructing an Arc of a Given Radius Within a Right Angle.** With the dividers set at the desired radius, and using point A as a pivot point, draw an arc which intersects the sides of the right angle, such as angle CAB in figure 48. With the same divider setting and using points E and D as the pivot points, draw intersecting arcs which locate point F. Using point F as the pivot point, construct an arc having the desired radius tangent to both sides of the right angle.

13-13. **Constructing Regular Hexagons.** A regular hexagon is a figure having six equal sides and equal angles. Draw the horizontal centerline of a circle, and using point O, as in figure 49, as the pivot point, draw a circle of the desired diameter. Leaving the dividers set to the radius of the circle and using points 1 and 4 as pivot points, draw arcs locating points 1, 6, 3, and 5. Draw chords connecting points 1 and 2, 2 and 3, 3 and 4, etc., to form the sides of the regular hexagon.

14. Handtools

14-1. Many people think that handtools are so simple that no one needs to bother pointing out the right and wrong way to use them. The truth is that "more tools are ruined by abuse than are worn out by use." In this section we will discuss the care and use of common and metal cutting handtools.

14-2. **Care Of Tools.** The proper storage and care of tools are the marks of a true craftsman. A machinist who allows his tools to be damaged through neglect or carelessness is almost certain to do his work in the same haphazard way. The following simple rules will help you to obtain the maximum service possible from your tools. At the same time, they will help you to do your job more easily and more quickly:

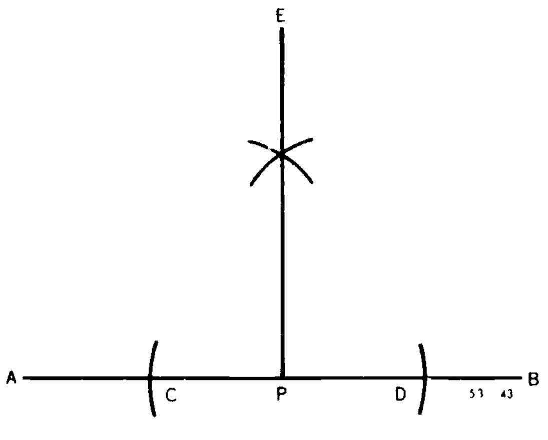


Figure 43. Erecting a perpendicular from a point on a line.

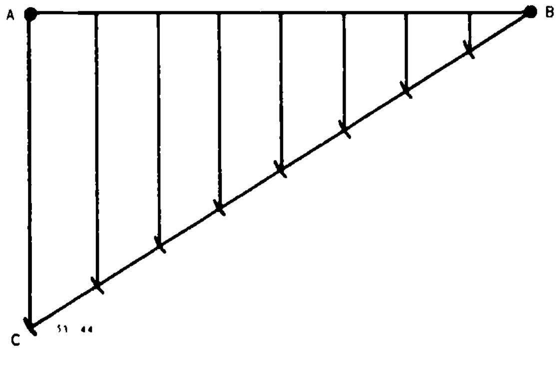


Figure 44. Dividing a line into equal parts.

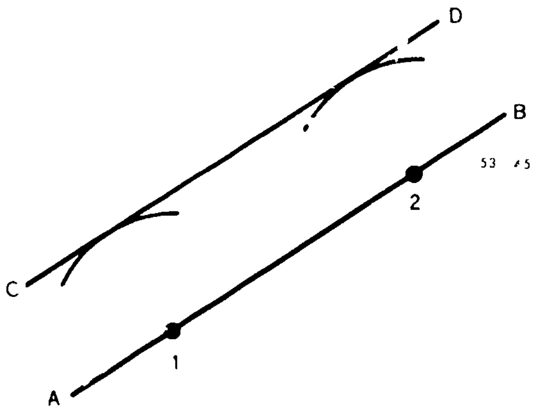


Figure 45 Constructing a line parallel to a line

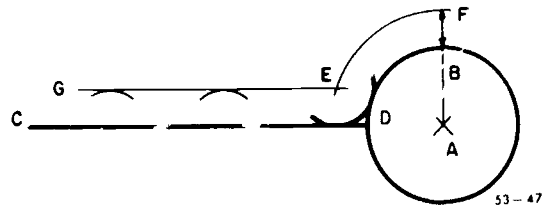


Figure 47. Constructing an arc tangent to a straight line and a circle.

a. Keep tools as clean as possible when you use them; never put one away dirty.

b. Arrange your toolbox so that there is a place for everything, and keep everything in its place.

c. Maintain the tools properly by repairing damaged tools as soon as possible. Keep such tools as chisels and punches sharp and free of dangerous burrs.

d. Apply a light film of oil to the tools to prevent rust. Wiping each quickly with a slightly oily rag will usually do the job.

e. Restrict a tool to its intended use.

f. Keep a checklist when you work on the line. Before leaving the job, make sure that every tool you took with you to the job is in your possession. More than one missing tool has been located—sometimes disastrously—when an aircraft engine was started! By keeping track of your tools, you will help prevent FOD (foreign object damage).

g. Keep your toolbox securely locked and in a safe place when you are not using your tools.

14-3. **Common Handtools.** You are probably already familiar with some common handtools. Many persons who work with handtools daily do not know how to select the proper tools for the job or how to use them properly. The first handtool we will discuss is the screwdriver.

14-4. *Screwdrivers.* Many machinists forget that there is a proper screwdriver for every job. Too often small screws are driven with a large screwdriver and large ones with a small screwdriver; the result is a damaged screw slot. The screwdriver has only one purpose—to loosen and tighten screws. With its slender steel shank and wood or plastic handle, the screwdriver is designed to take considerable twisting force or torque. But it is not designed to be used as a lever or prybar.

14-5. The sides of a correctly ground screwdriver blade should be practically parallel; however, most manufacturers taper the blade out to the shank body. A good trick is to grind the blade on an abrasive wheel so that the faces taper in very slightly for a short distance back of the tip. A screwdriver blade ground in this way will stay down in the screw slot even under high torque.

14-6. The most common types of screwdrivers are the cabinet, or standard, offset, close quarter, and cross-point, as shown in figure 50. The standard screwdriver is suitable for most ordinary work. The blade tip must have sharp corners to fit the slot; otherwise it is likely to slip and damage the slot in the screwhead. It is also important that the screwdriver be held firmly against the screw to prevent it from slipping and injuring the worker or damaging the work. The offset screwdriver is use-

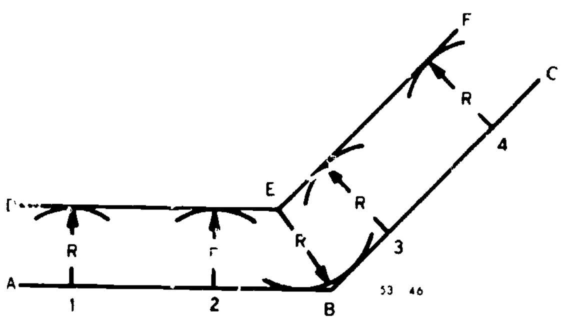


Figure 46. Constructing an arc tangent to intersecting lines.

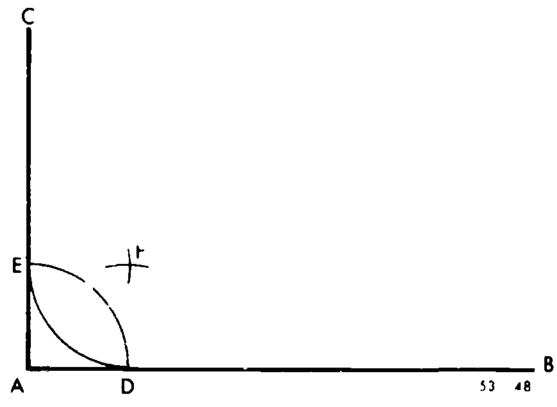


Figure 48. Constructing an arc within a right angle.

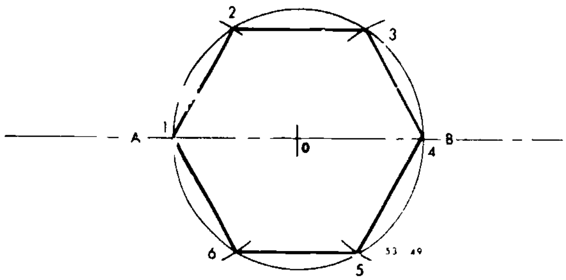


Figure 49. Constructing a regular hexagon.

ful in tight corners where the standard straight type will not enter. The close quarter screwdriver is also useful in a similar situation. The cross-point screwdriver is made with a special-shaped blade to fit the cross-slot screws.

14-7. *Hammers*. The machinist uses two types of hammers: the ball peen hammer and the plastic or rawhide faced mallet, shown in figure 51. Use the face of the 8-ounce hammer for striking prick and center punches and for other light duty applications, and the face of the 16-ounce hammer for heavy-duty work. The ball-shaped peens are useful for peening or riveting. If the surface of the work could be damaged by using a steel hammer, use a plastic or rawhide mallet. Most beginners have a tendency to hold the handle too close to the head, "choking" the hammer. Holding the hammer like this reduces the force of the blow and makes it harder to hold the hammerhead upright.

14-8. Most accidents involving hammers are caused by a loose head which in turn is caused by neglect on the part of the user. As shown in figure 52, the hole in the hammerhead is oval shaped with a bellmouth at each end. The handle is tapered to fit snugly into the smaller bellmouth to prevent the head from slipping in the direction of your hand. When inserted in the hole the small

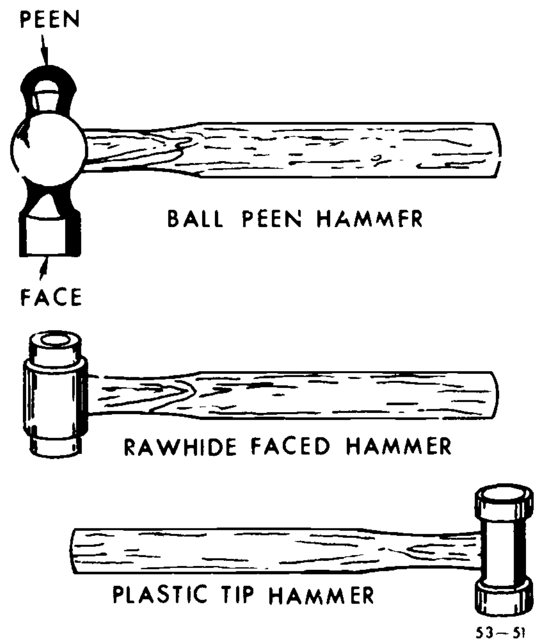


Figure 51. Hammers and mallets.

end of the taper can be spread to fill the other bellmouth completely by means of a steel wedge driven firmly into the end of the handle. Check the security of this wedge from time to time to see that it is still spreading the wood so that the head will not slip off the end of the handle. You may on occasion have to lightly grind the face of the hammerhead in order to keep it flat and perpendicular to the centerline or to remove dangerous burrs that have formed around the edges.

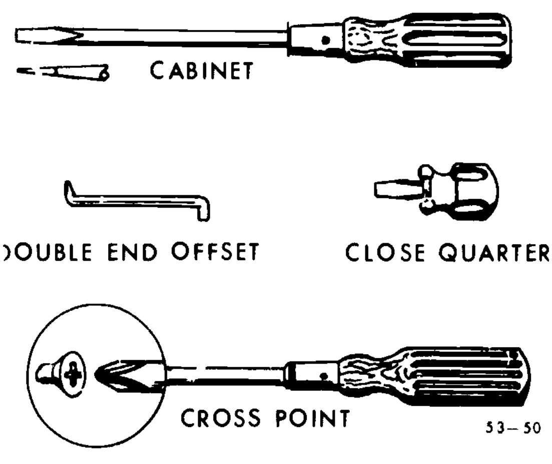


Figure 50. Screwdrivers.

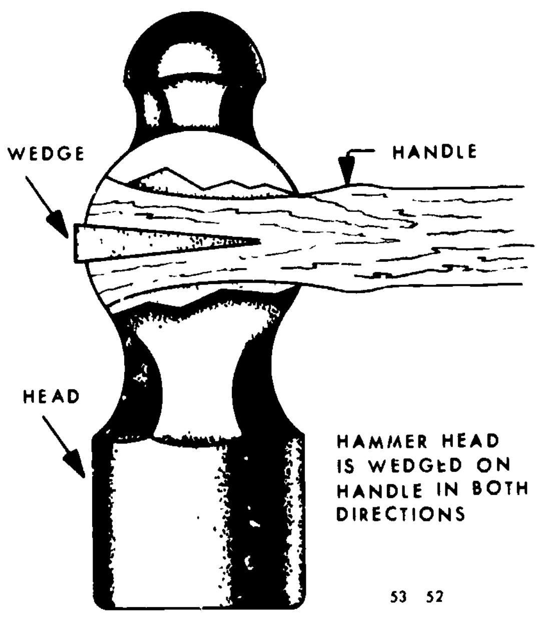


Figure 52. Securing head on handle.

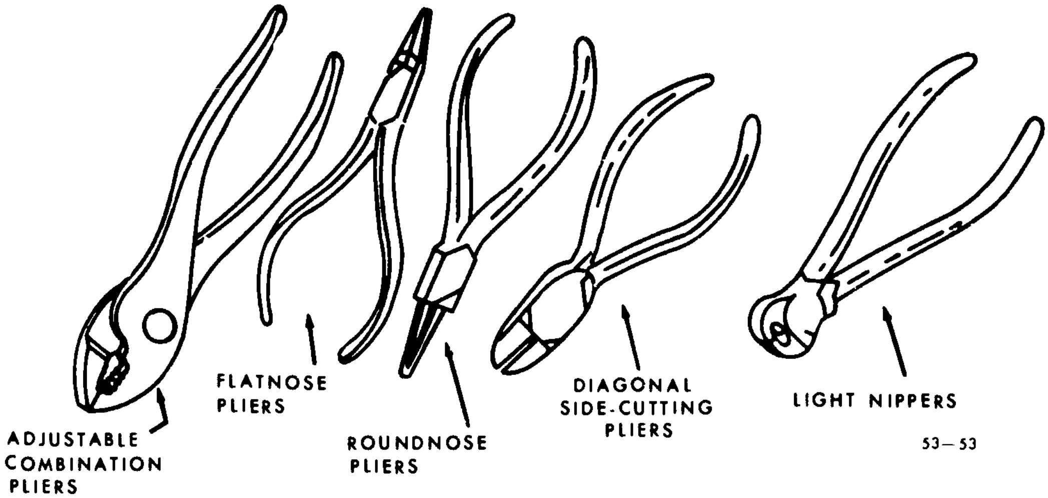


Figure 53. Pliers.

14-9. *Pliers.* The more common types of pliers are shown in figure 53. The various roundnose and flatnose pliers make it possible to bend and form metal in a variety of shapes or to work in close quarters. Side-cutting or diagonal-cutting pliers, often called dikes, are used to cut soft metal rods and wire. Never use pliers in place of a wrench. The teeth of the pliers will quickly mar or damage a nut or finished object. If you use pliers correctly, there is little danger of injury. Do not permit them to become greasy or oily, either on the jaws or on the handles. If you let the handles become oily or greasy, your hand cannot control the grip, and you may skin your knuckles. Always grasp pliers in such a way as to exert pressure on the extreme end of the handles.

14-10. *Wrenches.* Wrenches are tools for tightening or loosening nuts and bolts, or for grip-

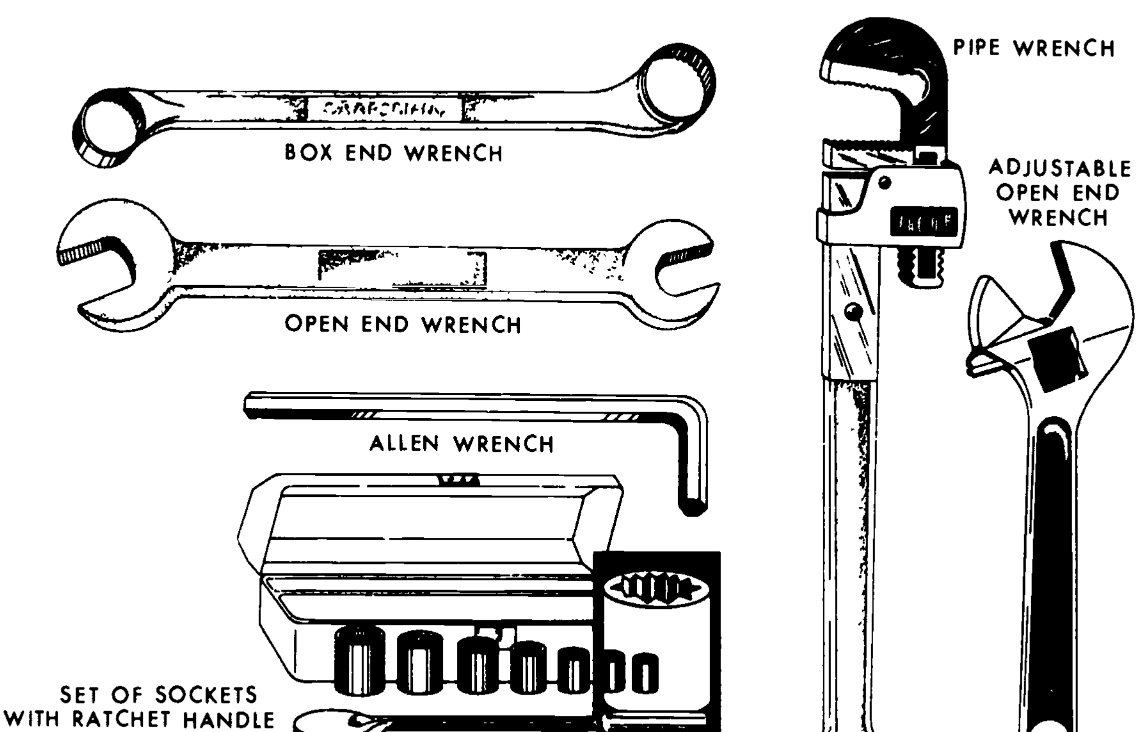


Figure 54. Wrenches.

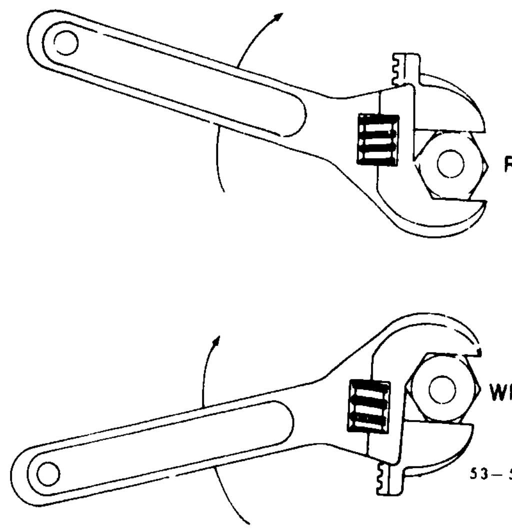


Figure 55. Direction of force.

ping material such as pipe or round stock. They are classified as adjustable, socket, open-end, pipe, and Allen wrenches, as shown in figure 54. Figure 55 shows the correct way to use an adjustable wrench. Always keep the movable jaw on the side toward which the wrench is moving.

14-11. *Open-end wrenches* are solid, nonadjustable wrenches, with openings in each end. About ten open-end wrenches are commonly found in a toolbox. The openings vary from $\frac{5}{16}$ inch to 1 inch in width. The size of the openings between the jaws determines the size of the wrench. The smallest wrench in the ordinary set has a $\frac{5}{16}$-inch opening in one end and a $\frac{3}{8}$-inch opening in the other. Because of this combination, the wrench is called a $\frac{5}{16}$–$\frac{3}{8}$-inch open-end wrench. These figures refer to the distance across the flats of the nut and *not* to the bolt diameter. Wrench openings usually measure 0.005 to 0.015 inch larger than the nominal sizes marked on the wrenches so that they will more easily slip on and off boltheads and nuts. The smaller the openings in the wrench, the shorter the overall length. This proportions the lever advantage of the wrench to the size of the bolt or stud. With a given amount of pull on a wrench, a short length produces less twisting or torque. It also reduces the possibility of shearing the bolt or stud or stripping the nut.

14-12. Open-end wrenches have their heads and openings set at an angle to the body (offset). Most are designed to be offset about 15°, but some wrenches are designed for a 22.5° offset. This design helps in working in close quarters. An elementary trick is to "flop" the wrench after every stroke; flopping is turning the wrench over so that what had been the upper face is now the lower face. The angle of the head is reversed to fit the next two flats of the nut. Flopping makes it easier to loosen or tighten a nut. You can turn a nut even when the swing of the wrench is limited to 30°.

14-13. Some special types of open-end wrenches have the angle of opening set at 75°; others are set at an angle of 90°. There are also special thin open-end wrenches which have extra long handles that permit working in narrow spaces. Never use this type of wrench for any job that requires much torque; the handles are not designed for heavy leverage.

14-14. The best feature about *box-end wrenches* is that they can be used in close quarters. They are called box-end wrenches because they box or completely surround the nut or bolthead. Box-end wrenches usually have 6 or 12 notches or points arranged in a circle. A 12-point box-end wrench can be used to loosen or snug up a nut with a minimum handle travel of only 30°, whereas a 60° swing would be necessary in an open-end wrench. Another advantage of the box wrench is that there is little or no chance of a wrench slipping off the nut or bolt. If the corners of a hex-headed nut or bolt are damaged so that a 12-point box-end wrench slips when pressure is applied, a 6-point box-end wrench should be used because of its better gripping power.

14-15. *Adjustable-jaw wrenches* have one stationary jaw and one adjustable jaw. The length of the handle determines the size of the wrench. For example, a 6-inch adjustable-jaw wrench has a 6-inch handle, although the jaws open to only $\frac{3}{4}$ inch. As the length of the handle increases, there is a proportional increase in the size of the opening of the jaws. The adjustable wrench is used frequently; however, it is not intended to take the place of standard open-end, box end, or socket wrenches.

14-16. The *socket wrench* helps greatly in making work easier. There is great variety of socket wrenches for every possible use and position. One thing to keep in mind in using socket wrenches is that they should never be overstressed. Never use an extension on a socket wrench handle to increase torque. Always use a socket that is big enough for the job. Use the correct drive size. The drive size refers to the size of the square boss used to turn the socket. Do not be in such a hurry that you use a socket with a $\frac{1}{4}$-inch drive when, by taking only a minute longer, you can find a socket with a $\frac{3}{8}$-inch drive that is built especially for a heavy job. Socket wrenches normally have 12 points but some of them have 6 points. The advantage of the 12-point socket over the 6-point socket is that the 12-point can be swung only half as far before it can be refitted for another grip on a nut. For this reason, it can be used in close quarters.

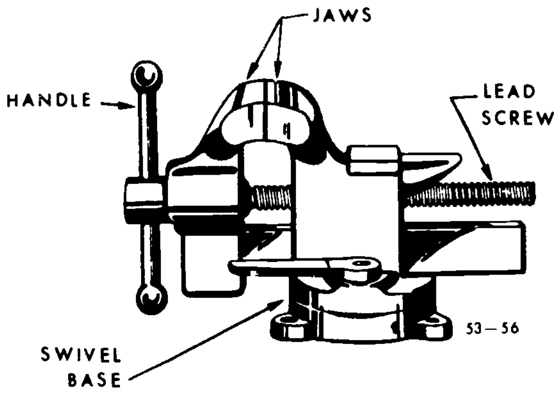


Figure 56. Machin't vise.

By contrast, a 6-point socket holds the nut better, offering less chance of damage to the nut or bolt-head.

14-17. *Torque wrenches* are calibrated tools used to measure the force of pull (pounds) when you are tightening nuts or hose clamps or checking the breakaway torque of various driving units. The torque is expressed in either inch-pounds or foot-pounds. There are two basic types of torque wrenches; indicating and breakaway. As its name implies, the indicating type indicates the amount of torque being applied, either on a dial or by means of a pointer. The breakaway type is more commonly used by the Air Force. This type automatically releases when a predetermined torque value has been reached.

14-18. *Allen wrenches* are another type of wrench you will frequently use. This wrench is six sided, L-shaped, and designed to fit into the recessed head of a setscrew or capscrew. Either end of this wrench will fit into the recess, making its use possible where either a long or short reach is desirable. Allen wrenches are not designed for work when a fairly high amount of torque is needed. In fact, if you apply excessive torque, a small Allen wrench will snap; and a larger one will bend out of shape and perhaps break. An Allen wrench should not be used if its corners are worn, as this condition can damage the recess in the screw. A damaged recess in the screw can make removal of the screw with the Allen wrench difficult or impossible.

14-19. *Machinist vise.* The machinist vise, shown in figure 56, is commonly used to hold work when handtools are being employed. You should place soft metal inserts, called soft jaws, between the vise jaws and the work when you wish to protect the finish of the work.

14-20. *Arbor press.* The arbor press, shown in figure 57, is not really a handtool in the normal sense of the word; however, it is hand operated. The arbor press is a simple machine for applying pressure to remove shafts from gears or pulleys or to press a shaft or mandrel into a hole. It is used extensively to assemble parts, such as bushings, pins, and shafts which require a press or force fit.

14-21. **Metal-Cutting Handtools.** The various metal-cutting handtools are often used for "bench work." They may also be used in conjunction with power tools.

14-22. *Chisels.* Chisels are used for chipping or cutting metal by hand when great accuracy is not essential and when it would be too expensive or impractical to set up the work in a machine. A variety of shapes are available, but your toolkit will contain at least a flat chisel and a cape chisel. Chisels are usually forged from octagonal-shaped tool steel containing sufficient carbon to make them tough enough to withstand hammering and hard enough to maintain a cutting edge. Chisels that have been sharpened correctly will cut any metal softer than the chisel—usually, any material that can be cut with a file. Figure 58 shows some of the common types of chisels.

14-23. Chisels should be ground with an included angle of 60° to 70°. The cutting angle may vary between these limits according to the strength of the material to be cut. Chisels used on hard or tough metals require a stronger cutting edge (70°), while a faster and cleaner cut can be made through softer metals with chisels ground to a sharper (60°) angle. If you also grind flat chisels to a slightly convex cutting edge, there will be less

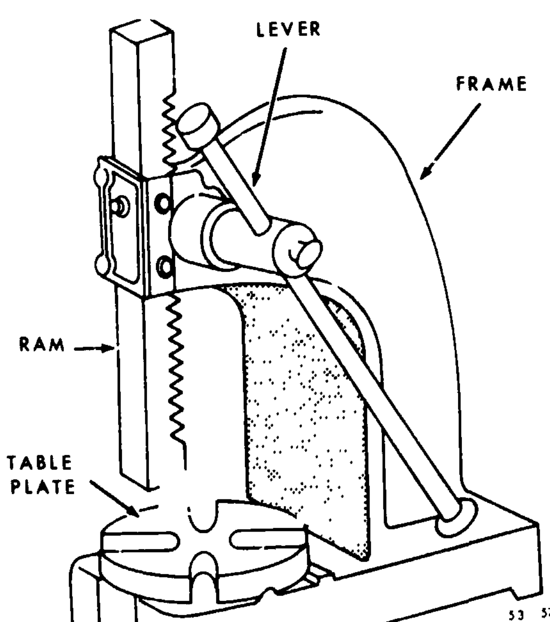


Figure 57. Arbor press

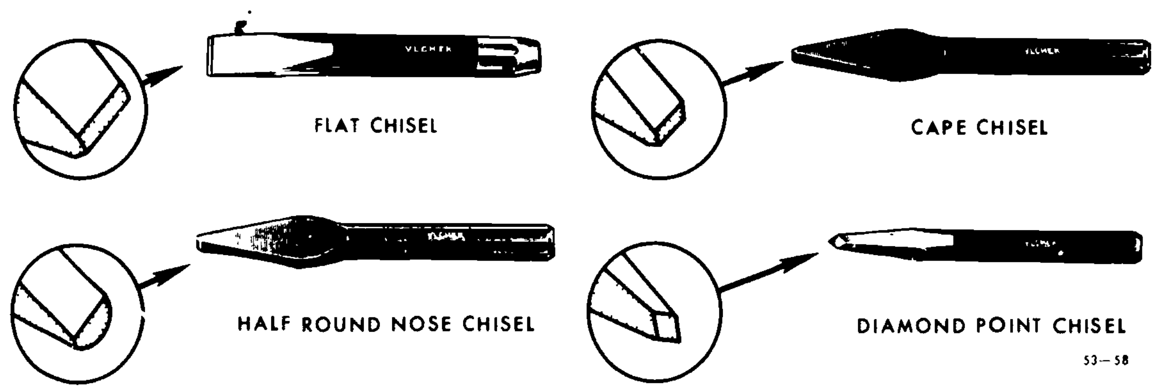


Figure 58. Chisels.

tendency for their corners to dig into the surfaces that are being chiseled. This method of grinding also focuses the impact at the center of the cutting edge where there is more material to withstand the strain than at the corners.

14-24. When you use a chisel, watch the cutting edge of the chisel rather than the head end. With a little practice you can soon acquire the knack of striking the head of the chisel without looking at it. By watching the cutting edge of the chisel, you can control the direction and depth of cut much better. CAUTION: Never allow a mushroom head to form on a chisel. The head end should be ground flat and have a beveled edge to prevent particles from flying off and possibly causing injury.

14-25. *Files.* The file is used for roughing out and finishing surfaces, shaping small parts, slightly reducing the size of parts to make them fit together and removing toolmarks and burrs. They are often used for preparing surfaces for polishing. Files are manufactured in many shapes and sizes. They are identified by their general shape or cross section or by their particular use. Your toolkit will probably include the five files which are best for general machine shop work. They are the 10-inch flat smooth, 10-inch second-cut half-round, 8-inch second-cut round, 10-inch second-cut square, and 10-inch mill bastard. Figure 59 shows the parts of a file. Figure 60 shows the shapes of files. The shape is governed by the cross section. Figure 61 shows a file cuts. Cut refers to both the coarseness (coarse, bastard, second, and smooth) and type of cut (single or double). A double cut file makes a faster but rougher cut than a single-cut file. A 10-inch bastard file is coarser than an 8-inch bastard because the distance between the teeth increases as the length of the file increases.

14-26. Figure 62 shows the proper method of holding a file for crossfiling (conventional). You should reduce the pressure on the back stroke to avoid excessive wear on the teeth. Do not exceed 30–40 strokes per minute. A higher rate may damage the file. Never use a file as a pry or hammer, for it is almost sure to break and injure you or damage the work.

14-27. Figure 63 shows how to round a corner and how to do drawfiling. Drawfiling produces accurate and extremely smooth filed surfaces. CAUTION: Never use a file without a file handle, as

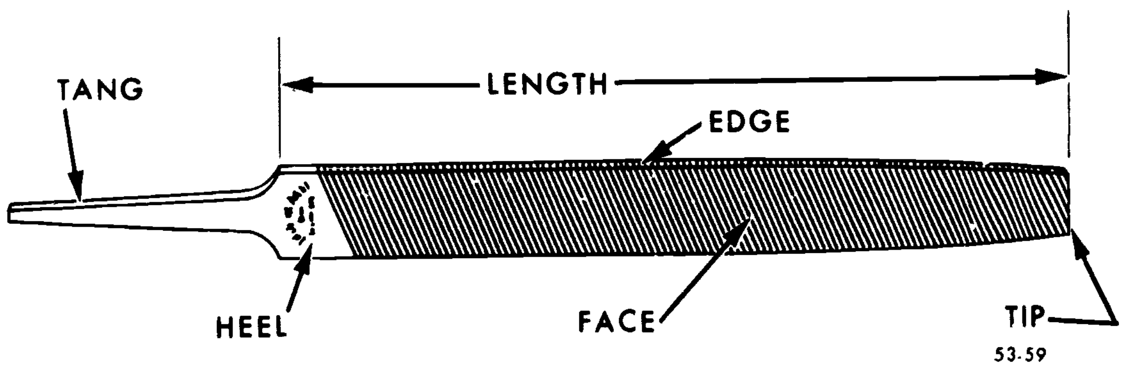


Figure 59. File parts.

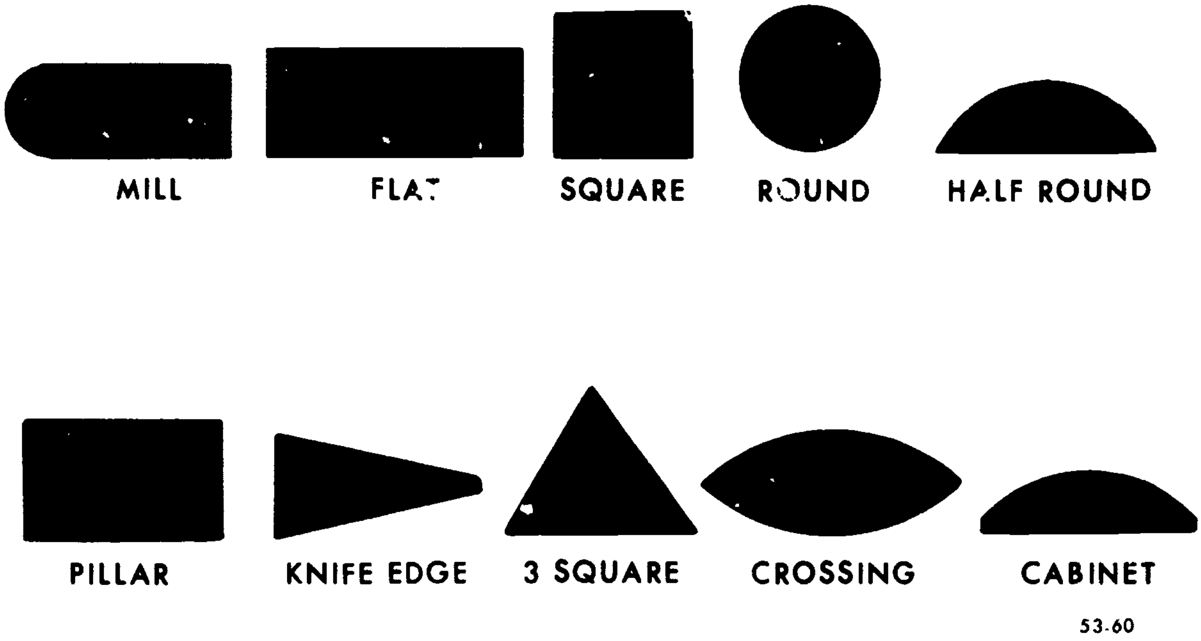


Figure 60. Various file shapes.

shown in figures 62 and 64. The sharp tang can easily penetrate your hand if the file should slip. Use a file brush and card, shown in figure 64, to clean the file and to prevent scratching the work surface due to "pinning" (metal particles lodging in the file teeth). Be sure to clean the file often during use. Also, never place files in contact with each other, as contact can quickly damage the file teeth.

14-28. Some of the more common types of hand files and their uses are as follows:

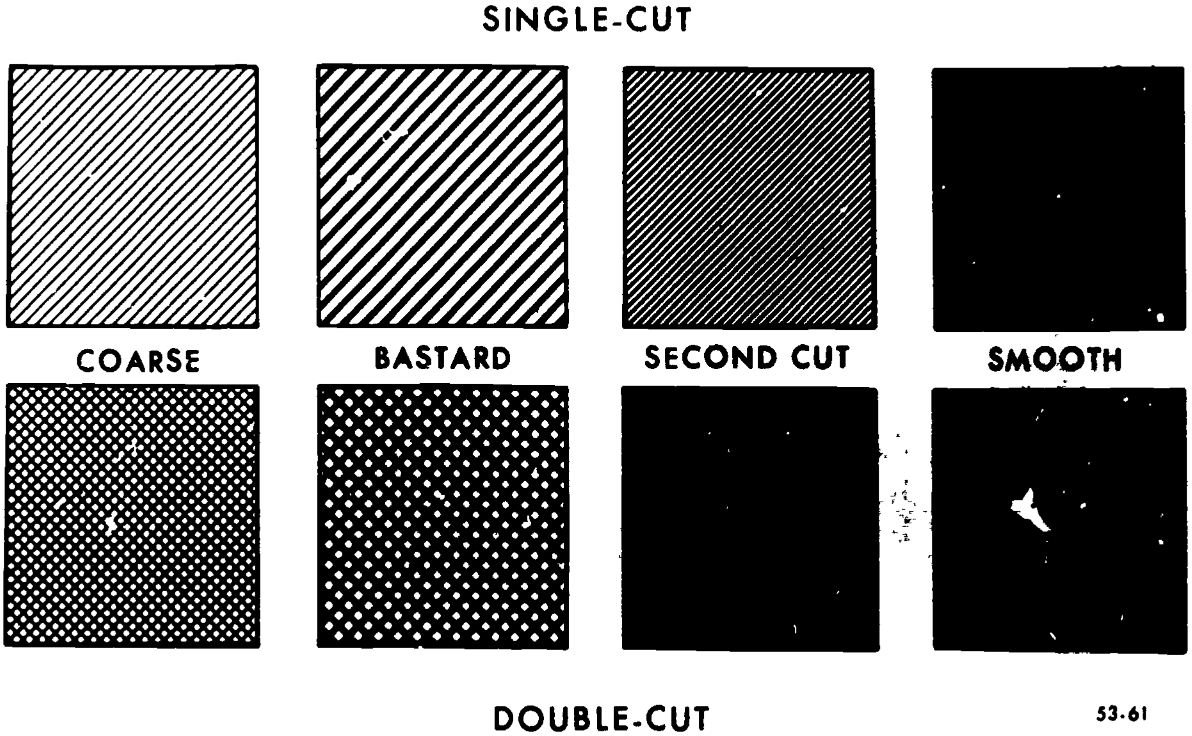


Figure 61. File cuts.

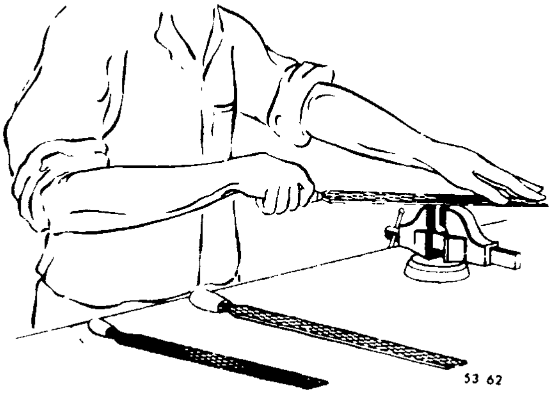


Figure 62. Crossfiling.

a. Flat files are tapered slightly toward the point in both width and thickness. They are double-cut on both sides and single-cut on both edges. They are available in bastard, second-cut, and smooth cuts and are used for all common filing operations.

b. Mill files are tapered slightly in thickness and width for approximately one-third of their length. They are single-cut in bastard, second-cut, or smooth cut. Mill files are available with square edges and with one or two round edges for filing gullets between saw teeth. They are used mainly for sharpening mill or circular saws, edge tools, and machine knives. They are also used for lathe work, for drawfiling, and for finishing brass and bronze.

c. Half-round files are not complete half-circles as the name implies, since the arcs are about one-third of a circle. They are available in nearly all cuts. They are double-cut, tapered in width and thickness toward the point, and have one flat and one oval side. They are used primarily for filing concave surfaces.

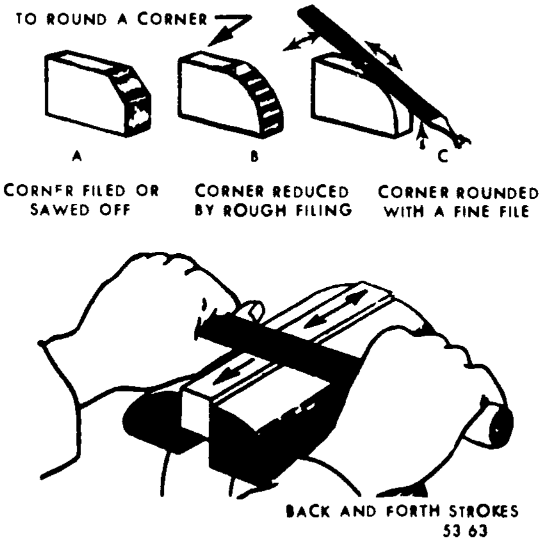


Figure 63. Rounding corners and drawfiling.

d. Round files, often called rattail files, are circular in cross section, are tapered or blunt, and are single- or double-cut. They are made in bastard, second-cut, and smooth cut. Round files are used mainly to file or enlarge circular openings or to file concave surfaces.

e. Square files are square in cross section, are tapered or blunt, and are double-cut on all four sides. They are made in bastard, second-cut, and smooth cut and are used for filing keyways, slots, corners, and general surface filing.

f. Three-square files, often called triangular or three corner files, are triangular in cross section, are tapered or blunt, and have sharp edges. They are double-cut or single-cut and are made in bastard, second-cut, and smooth cut. They are used to file acute internal angles, to clear out square corners, and to repair damaged threads.

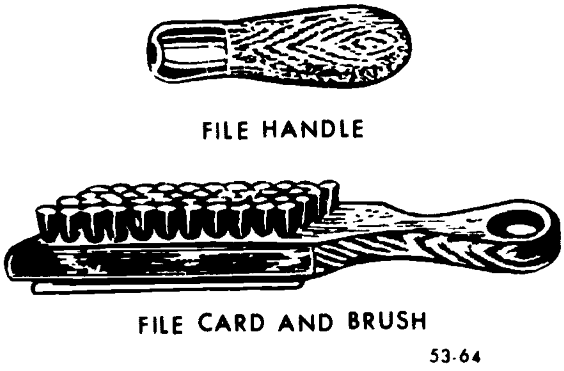


Figure 64. File handle and file card.

g. Knife edge files are shaped like knife blades. They are double-cut on both sides and single-cut on its one edge. Knife files are made in bastard, second-cut, and smooth cut and are used for filing V-grooves and narrow slots.

h. Pillar files are of even width, are tapered in thickness, have one "safe" (uncut) edge, and are narrower than most hand files. They are used mainly for filing keyways and slots.

i. Curved-tooth, or vixen, files are widely used for smooth, rapid filing of cast iron, bronze, lead, babbit, aluminum, zinc, plastics, and sheet metal. The curved teeth readily clear themselves of chips.

j. Lead-float files are used especially for filing lead, babbit, and other soft metals. They have coarse, short-angle, single-cut teeth which shear away the metal rapidly under ordinary pressure.

14-29. *Hacksaws.* Hacksaws are designed to cut metal in much the same manner that a carpen-

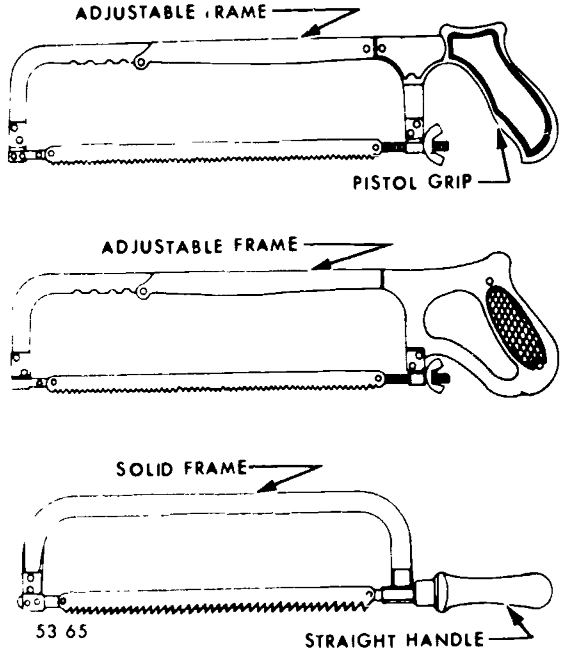


Figure 65. Hacksaw frames.

ter's saw cuts wood. You must understand the correct use in order to cut metal efficiently with the least strain or damage to the saw blade.

14-30. Hacksaw frames are constructed with either a pistol grip or a straight handle and with a solid or an adjustable frame, as shown in figure 65. Although the solid frame is stronger, it will hold only one length of blade. The adjustable frame accommodates blades of 8, 10, or 12 inches in length (as measured between centers of the stud pinholes). In either case, you mount the blade on pins located on turning studs at each end of the frame. These free-turning studs permit setting the blade at a 90° angle to the normal position in order to avoid interference from the frame when you are making long cuts. In the straight-handle type of frame you adjust the blade tension by twisting the handle clockwise. In the pistol-grip type, you adjust the blade tension by turning the wingnut.

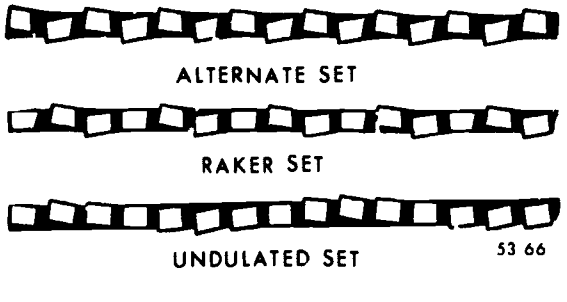


Figure 66. Set of hacksaw blade teeth.

14-31. You should carefully select the proper blade for each cutting purpose. Hacksaw blades are made of a high-grade tool steel that has been hardened and tempered. An *all-hard* blade is hardened throughout; a *flexible* blade has only its teeth hardened and will, therefore, not break as easily under bending stresses. Use an all-hard blade for sawing brass, tool steel, cast iron, and other stock of heavy cross section. Use a flexible blade for sawing hollow shapes and metals of light cross section, such as channel iron, tubing, tin, copper, aluminum, or babbit.

14-32. Set is the angle (or angles) at which the teeth of the blade are set to provide clearance for the rest of the blade in the saw cut. You will notice, if you look at the hacksaw blades in your toolkit, that the blades with coarse teeth have one tooth moved slightly to the right, the next one to the left, the third one to the right, etc. This is the standard, or alternate, set, as shown in figure 66. Medium-coarse teeth may be bent in a raker set (one right, one left, one straight, etc.), while fine-toothed blades usually have an undulated set (whole sections of teeth alternately bent to the right or left). Carpenters reset the teeth of their woodcutting saws from time to time, but when a hacksaw does not cut well or binds excessively

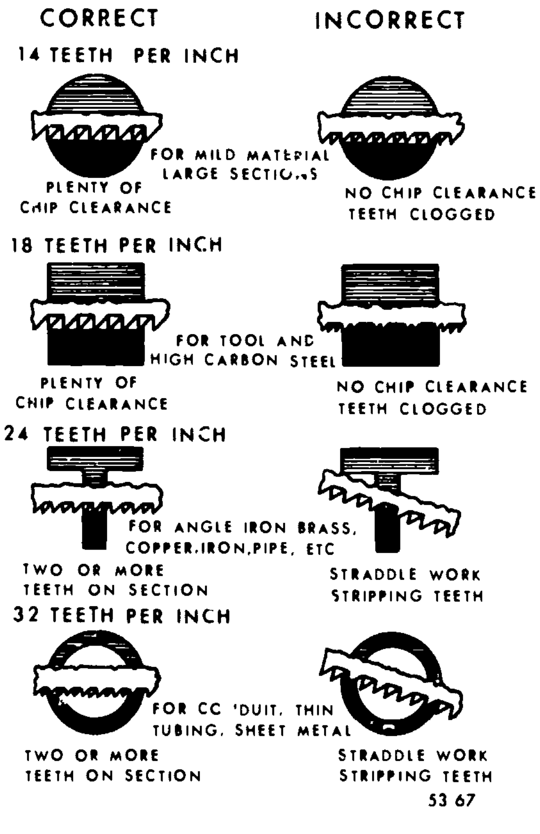


Figure 67. Selection of pitch.

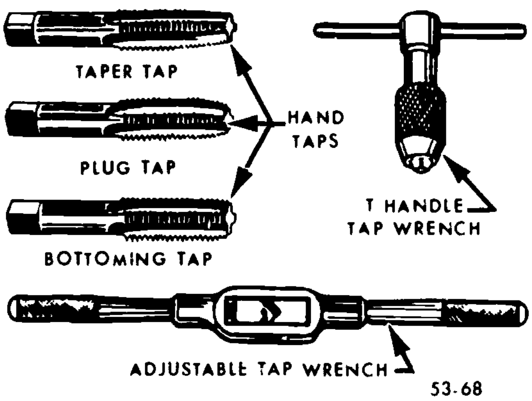


Figure 68. Handtaps and tap wrenches.

from insufficient set, you discard the blade and insert a new one. You do not have to worry about the set of a hacksaw because it is built into the blade. However, your judgment in selecting a blade with proper *pitch* (number of teeth per inch) may well determine how quickly and easily the saw cuts through the material and how long the blade can be used. Blades are made with pitches of 14, 18, 24, and 32 teeth per inch, as shown in figure 67. Follow these recommendations for best results:

- Use a 14-pitch blade on machine steel, cold rolled steel, or structural steel. The coarse pitch makes the sawing free and fast cutting.
- Use an 18-pitch blade on solid stock, aluminum, babbitt, tool steel, high-speed steel, cast iron, etc. This pitch is recommended for general use.
- Use a 24-pitch blade on tubing, tin, brass, copper, channel iron, and sheet metal over 18 gage. If you use a coarser pitch the thin material tends to strip the teeth out of the blade and makes it difficult to push the saw.
- Use a 32-pitch blade on thin-walled tubing, electrical conduit, and sheet metal thinner than 18 gage.
- Select a pitch or manage the sawing so that two or more teeth are usually in contact with the material, as you see in figure 71.

14-33. A small notch filed at the starting point will help you to start the saw blade accurately. Apply pressure on the forward stroke and reduce it on the back stroke. The ease with which a piece of metal may be cut depends upon the speed and pressure applied to the saw. A cutting speed of 40 to 50 strokes per minute is best since this speed does not tire you and also permits you to relieve the pressure on the return stroke. Faster speeds will damage thin blades because the heat generated draws the temper and makes the blade soft. Here are hints that will help you to speed up the sawing and make it easier:

- Apply a little oil to the sides of the blade with your finger to reduce binding when you are making a deep cut.
- Take care to prevent either stripping the teeth or breaking the blade. Some of the causes of breakage are a pitch which is too coarse for the material, the application of too much pressure on the cutting stroke, the slipping of work in the vise, and cutting off at an angle and then trying to straighten the cut by twisting the saw.
- Clamp thin stock between two pieces of wood or soft metal and saw through all three pieces. This will prevent annoying chattering and possible damage to the thin stock.

14-34. *Taps.* A tap is used to cut threads on the inside of a round hole. It is a hardened tool-steel screw with flutes (grooves) cut lengthwise across the threads to form cutting edges. The tap is screwed into the hole, and cutting faces formed by the grooves cut the thread into the wall of the hole. You screw a tap into the hole by using a special wrench to give the necessary leverage. Common types of handtaps are the taper, plug, and bottoming taps, as shown in figure 68. The taper tap is tapered at the end (7 to 9 threads) to aid in starting the thread into the hole. It is used when the thread runs all the way through the hole. The plug tap is also tapered, but for a shorter distance back from the end (2½ to 5 threads) After you start the thread with the taper tap, use a plug tap to cut to

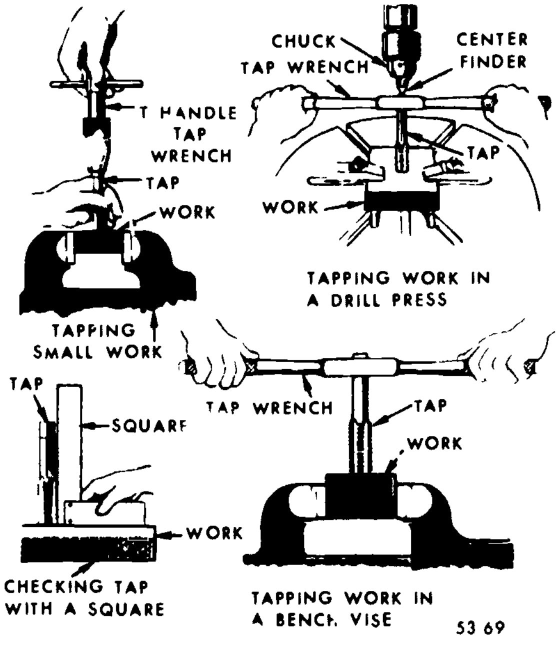


Figure 69. Starting a tap.

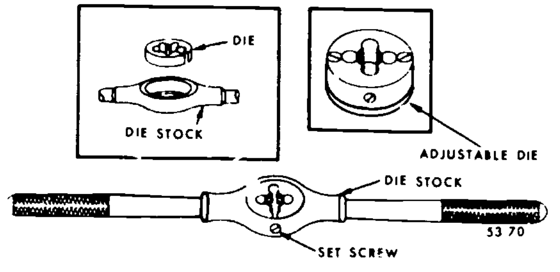


Figure 70. Dies and die stock.

the maximum length. You may also use a plug tap when one end of the hole is closed. The bottoming tap has only one thread on the end that is chamfered or beveled. Use this tap when it is necessary to cut a full thread all of the way to the bottom of a blind hole.

14-35. A hole to be tapped is drilled with a tap drill first. The size of the drill depends upon the thread diameter and number of threads per inch to be cut. Drill sizes are obtained from tables found in any machinist handbook. For example, the tap drill required for a 1/2-inch diameter, 13 threads per inch thread is 0.4216, or 27/64 inch. Note that the tap drill diameter is less than the diameter of the thread. The end of the hole to be tapped should be chamfered to a diameter at least equal to the thread diameter. This assures more accurate alinement and better starting of the tap, and it will prevent a large burr from forming. Three methods of starting a tap are shown in figure 69. Note that one method involves using a T-handle tap wrench and two of them involve using a tap and reamer wrench. In any case the tap should be checked for alinement with the hole, as shown in the lower left view. The tap may break if it is not in alinement with the hole.

14-36. To start a thread, use a taper or plug tap. Use sufficient pressure to insure that the threads will "catch" or start without tearing or reaming the top of the hole. It is important to start the tap straight and keep it so throughout the operation, because taps, especially small ones, will almost surely break if they are bent or strained. If the tap does not enter squarely at the beginning of the operation, straighten it by removing it from the hole and restarting it with pressure applied in the direction from which the tap leans. Be careful not to exert too much pressure at any one time in the straightening process. It is safer to repeat this operation a second or third time, if necessary. When you have properly started and alined the tap, it will follow and feed itself into the hole when you turn the tap wrench.

14-37. Many times it is possible to tap a hole while the work is mounted on a drill press. The tapping is done immediately after the hole has been drilled, while the work is still alined under the spindle of the drill press. This can save time and effort in clamping the work and starting the tap straight. The tap is held in place with a pointed rod mounted in the drill chuck or a lathe center inserted in the spindle. The point of the rod or center is placed in the center hole in the end of the tap and pressure is applied by use of the spindle. The tap can then be turned with a tap and reamer wrench or an adjustable jaw wrench.

14-38. When you are tapping steel or any tough metal, the best method is to take a part turn forward, then a part turn backward to break the chip. Be careful to do this with a steady motion and turning pressure to avoid breaking the tap. Continue this until the hole has been completely tapped. Break the chips from time to time. Long chips may clog the flutes and tear the thread or break the tap. Always use a cutting lubricant to obtain the best results.

14-39. *Dies*. A die is used to cut external threads. Dies are formed in hardened discs of uniform size that can be clamped in a leverage-producing device called a stock. Figure 70 shows the complete assembly ready for use. The procedure for using dies is similar to that for tapping. Hold the work firmly in a vise and file off any burr on the end of the piece to be threaded. To start the thread, place the larger side of the opening in the die over the work and press down on the stock firmly while turning it in a clockwise direction. When the die teeth catch and begin to cut, pour a

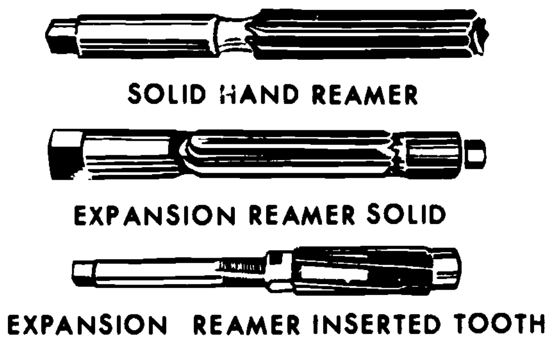


Figure 71. Hand reamers.

few drops of cutting oil on the end of the work and continue turning as if tapping—a part turn forward, followed by a part turn backward—until the thread is cut to the length desired. It is important to lubricate the work frequently. After the die teeth have taken hold, remove feed pressure and allow the threads to pull the die on the work at the proper rate.

14-40. *Reamers.* Reamers are used to make holes that are smooth and true to size and for enlarging cored or drilled holes. We are concerned only with hand reamers in this section. You can feed hand reamers into the hole by hand by applying a tap and reamer wrench that fits the squared shank. Hand reamers may be solid or adjustable, as shown in figure 71. The flutes may be straight or tapered. In reaming a hole, ream only enough material to remove the marks left by the cutting tool during the previous operation and to get the hole to its specified size. The amount of material

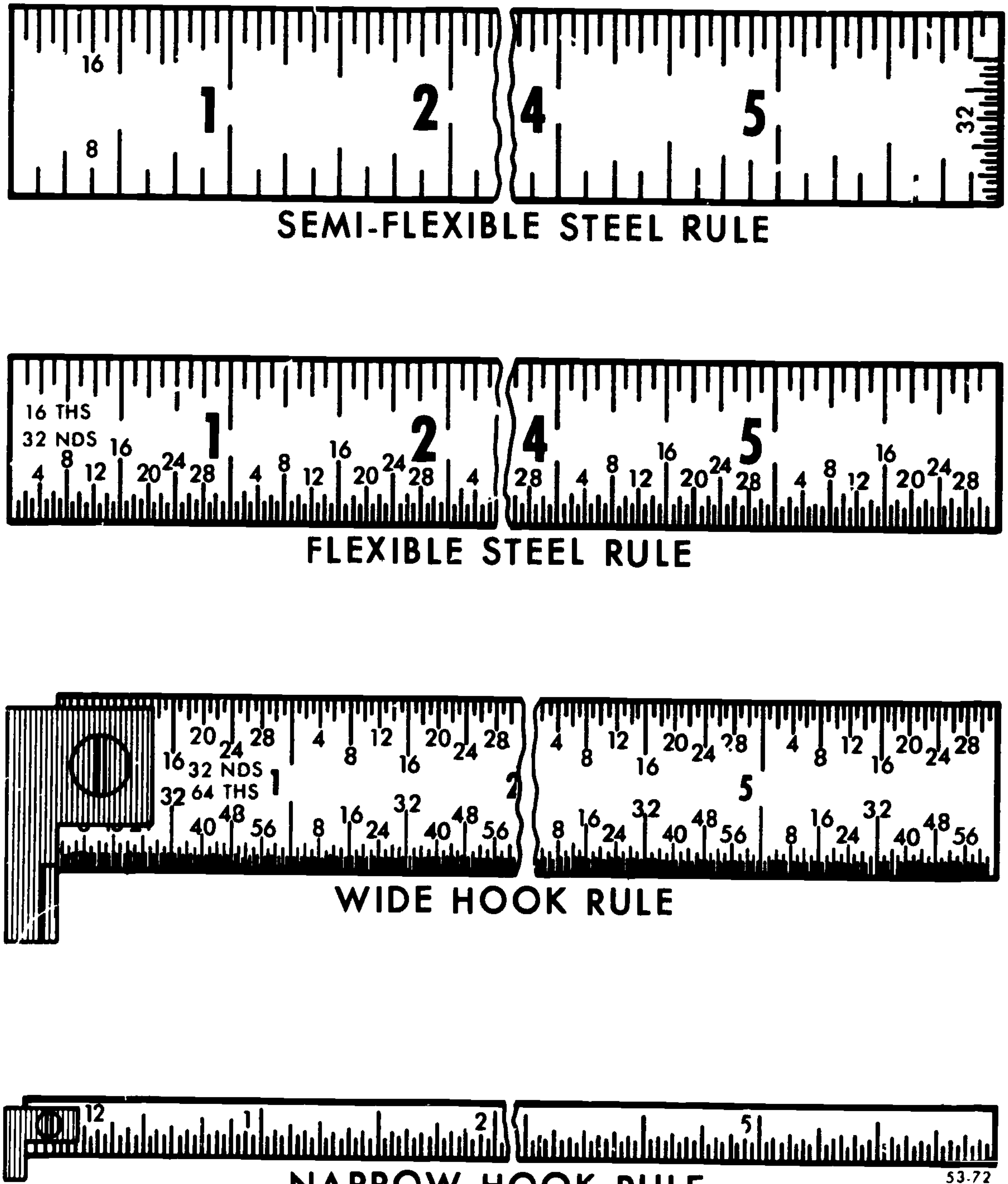


Figure 72. Common machinist scales.

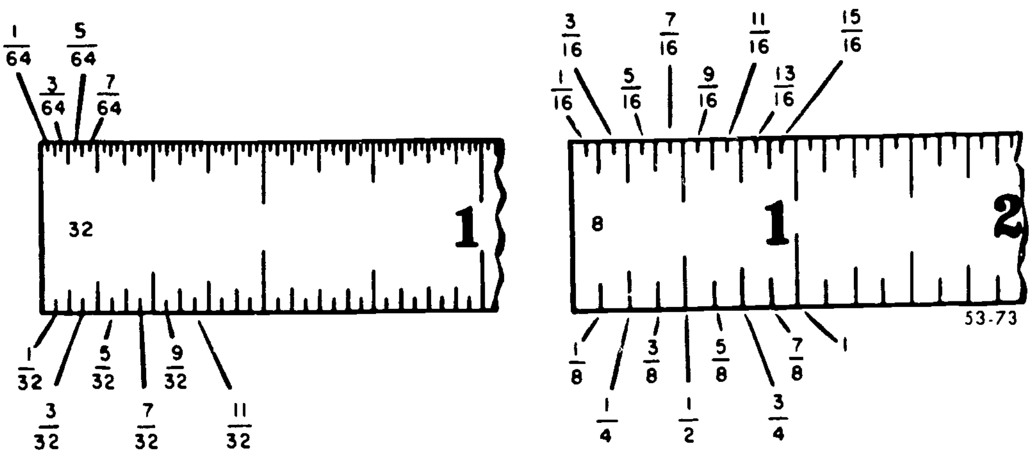


Figure 73. Typical scale graduations.

to be removed by a reamer should not exceed 0.015. This gives the reamer longer life between sharpening and produces holes of more accurate size. To ream steel, always use cutting oil or ordinary machine oil. Cast iron should be reamed dry, although many machinists use a small amount of kerosene or turpentine on the reamer if the cast iron is hard. Never use any kind of lubricant on soft cast iron. The helical flutes of hand reamers are cut left handed so that the reamer will not pull itself into the hole. In any case, always turn both straight and helical fluted reamers clockwise, even when you are removing them from the hole. If you reverse the reamer, you will mar the reamed hole, dull the reamer, and burr the cutting edges. In hand reaming, applying too much pressure will cause the reamer to feed in too rapidly, resulting in possible breakage of the reamer. If more material must be removed than is recommended for one cut, take a series of light cuts rather than one heavy cut. Always check the hole for size after each cut.

14-41. The *solid* and reamer has either straight or helical flutes and is tapered slightly on the end for greater ease in starting. A solid reamer is used to ream a hole to a basic size and is limited to one size of hole. The size of the reamer is designated by its diameter measured across opposite cutting edges.

14-42. There are two types of *expansion* reamers: solid and adjustable blade. The solid type has a body that is bored with a slight taper and is slotted. This permits a slight expension on the reamer end when the adjustment screw is turned. Limits of expansion recommended for different sizes are 1/4 to 15/32 inch, 0.005 inch; 1/2 to 31/32 inch, 0.008 inch; 1 to $1\frac{23}{32}$ inch, 0.010 inch; and $1\frac{3}{4}$ to $2\frac{1}{2}$ inches, 0.012 inch. Use this type of expansion reamer to ream holes to the basic size and, in addition, to produce slight variations in size to obtain various fits. The other type of expansion reamer, the adjustable blade reamer, is probably the most efficient type for hand reaming. The cutting blades in this reamer are removable and are set in slots that are milled at a slight taper with the centerline of the reamer. These blades are held in position by a nut at each end of the blade. You can adjust the size of this reamer within a considerable range over or under the normal size. Adjust the blades by loosening one nut on the threaded body and tightening the other. When a set of blades is worn out, you may substitute a new set without replacing the slotted shank.

14-43. The *line* reamer, as illustrated in figure 71, has a pilot mounted on the fluted end to assist in alining the reamer as it reams two or more holes in line with each other. The diameter of the pilot is 0.005 smaller than the diameter of the reamer.

15. Measuring Tools

15-1. A machinist uses a wide variety of measuring tools. Some are simple in design; others are quite complex. Some of them are calibrated in thousandths and ten thousandths, while others are in fractions of an inch. In this section, we will discuss scales, calipers, micrometers, vernier instruments, dial indicators, and gages.

15-2. **Scales.** Scales or rules are available in a variety of sizes and shapes; each is especially adapted to certain classes of work. For instance, the flexible rule shown in figure 72 is 6 inches long by 1/2 inch wide and is made of thin (1/64 inch) tempered spring steel so that you can bend it a reasonable amount when it is necessary to take measurements in close quarters. Some 6-inch rules, on the other hand, are wider (3/4 inch) and thicker (3/64 inch) for use as rigid straight edges. Hook rules are handy for measuring through holes and other tight places. And, even though all the scales shown in figure 73 are calibrated in fractional parts of an inch, you can also obtain rules marked off in decimals or in metric measurements. Figure 73 shows the graduations most often used on scales.

15-3. **Calipers.** The two tools most commonly used to transfer measurements from the work to a rule or from the rule to the work are spring-joint *outside* and *inside calipers*. As their names imply, they are constructed to take external or internal measurements, as illustrated in figure 74. The C-shaped spring at the pivot end tends to force the legs apart at all times. You can secure a steady and reasonably precise adjustment of the measuring points by screwing or unscrewing the nut located at the side of the caliper. You may take caliper measurements in either of two ways: (1) set the calipers over (or in) the object and then check the unknown measurement against a steel rule, or (2) set the calipers to a measurement on a steel rule and then check the work to see whether or not it matches the measurement. You can set an inside caliper more accurately with an outside micrometer than with a rule. You can also set an outside caliper more accurately with an inside micrometer than with a rule. The increase in accuracy obtained when you use micrometers to set calipers will be evident after you study micrometers.

15-4. **Micrometers.** The smallest measurement you can make with the spring caliper and steel rule is 1/64 inch. To make finer measurements than these (thousandths and ten-thousandths of an inch), you must use a micrometer, sometimes referred to as a mike. We will discuss three types of micrometers. Each micrometer is designed to take a particular type of measurement. Since accurate measurements are extremely important in machine shop work, we will explain in detail how to read micrometers. It is easier to read a micrometer if you know how it is constructed.

15-5. *Outside micrometer.* The outside micrometer is designed to measure external dimensions. It is used more often than any other type. It measures precise distances by recording the endwise travel of a screw during a whole turn or any part of a turn of the screw. The major parts of an outside micrometer are shown in figure 75. Since the thimble of a micrometer can travel a maximum distance of only 1 inch, larger measurements require different frames. For example, a 2-inch micrometer has a measuring range from 1 to 2

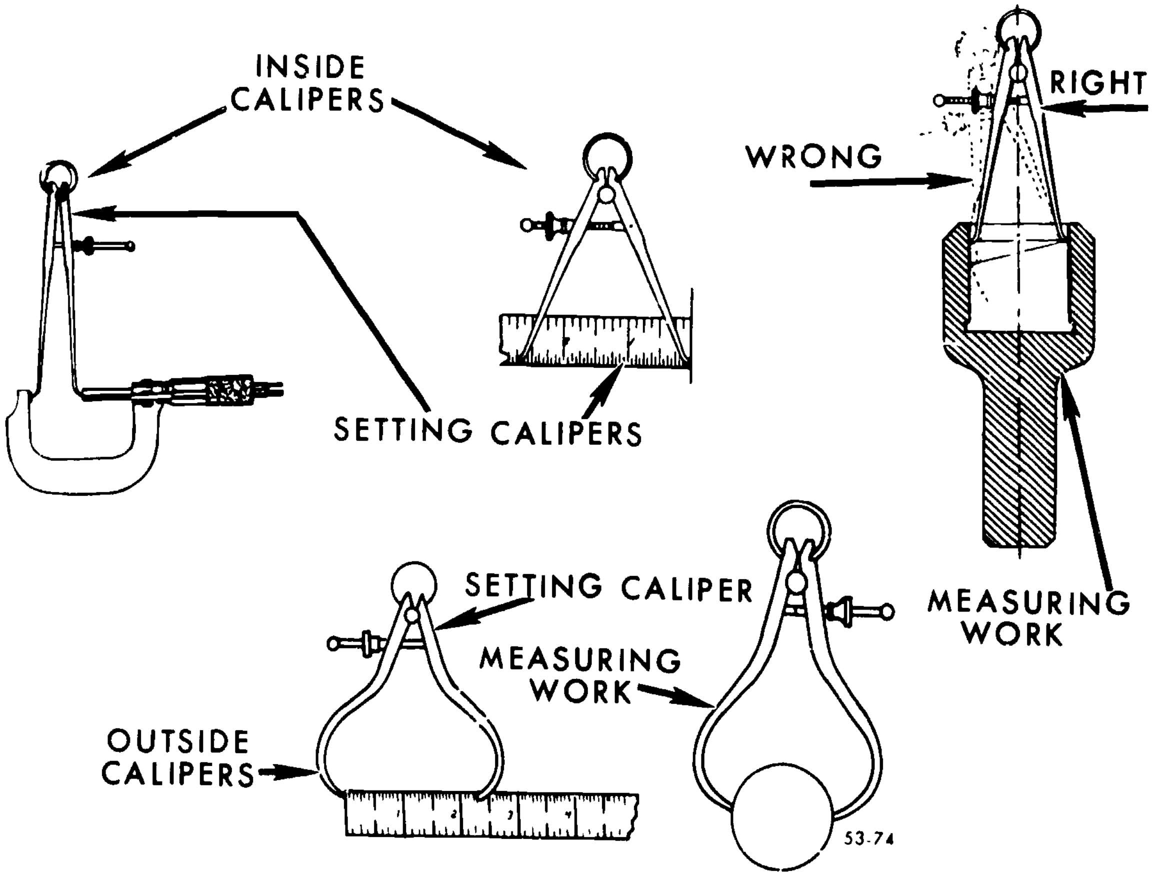
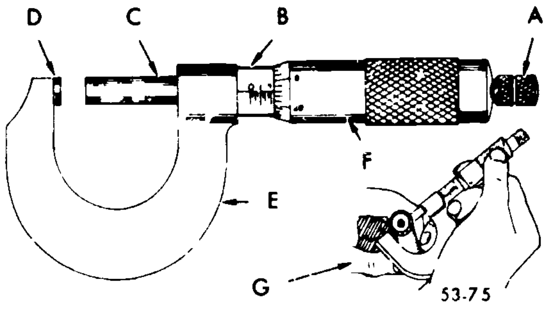


Figure 74. Calipers and their use.



A. RATCHET
B. BARREL
C. SPINDLE
D. ANVIL
E. FRAME
F. THIMBLE
G. MEASURING WORK

Figure 75. Outside micrometer.

inches, and a 6-inch micrometer has a measuring range from 5 to 6 inches.

15-6. A micrometer can measure because of the spindle screw. Since this screw has a pitch of 40 threads per inch, turning the thimble 40 complete revolutions moves the spindle exactly 1 inch. A clockwise turn moves the spindle toward the anvil; a counterclockwise turn moves the spindle away from the anvil. You can easily understand, then, that a single revolution from one screw thread to the next will move the spindle 1/40, or twenty-five thousandths (0.025) of an inch (1.000 inch divided by 40 equals 0.025 inch). Along the barrel in figure 75 you can see a figure 2 over the eighth graduation to indicate 2/10 inch (8 times 0.025 = 0.200). In figure 75 the thimble has traveled no full graduations past the 0 point; therefore the measurement reads 0.200 inch (0.200 inch as signified by the number 2) plus some yet unknown quantity indicated by the spindle having gone past the number 2 graduation

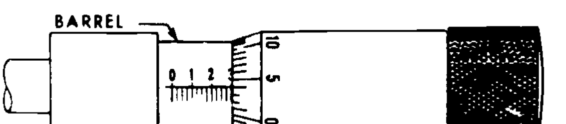


Figure 76. Micrometer readings.

Now look at the graduation on the thimble. Since the circumference of the thimble is divided into 25 equal spaces, each graduation represents 1/1000 inch (0.001 inch). With a reading of 0.200 inch on the barrel and a reading of 23 graduations past the zero point on the thimble, the final measurement in figure 75 is two hundred twenty-three thousandths of an inch (0.223 inch). Figure 76 shows some additional micrometer settings and readings. Study them until you are certain that you understand how the readings are obtained.

15-7. In the illustrations the graduations on the thimble are always shown as lining up exactly with the revolution line. In many cases, however, you will not find this true in actual practice. If the micrometer does not have a *vernier* scale graduated on the barrel, you will have to divide the distance

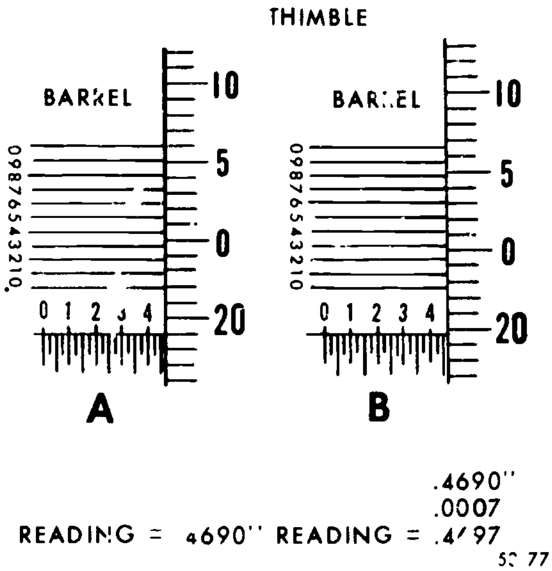


Figure 77. Vernier scale.

mentally between thimble graduations into tenths and guess at the number of ten-thousandths of an inch. However, when the micrometer has a vernier scale, as shown in figure 77, you can obtain a more precise measurement, because one of the thimble graduations lines up exactly with a vernier scale graduation. This happens because a distance equal to only nine thimble divisions has been divided into ten vernier graduations—each vernier graduation is 1/10,000 inch (0.0001) shorter than a thimble division; hence, when thimble division 19 exactly matches the barrel revolution line, as shown in A of figure 77, the zero graduations at each end of the vernier scale exactly match a thimble graduation. This indicates a reading in exact thousandths of an inch of 0.4690 inch. In figure 77,B, the revolution line splits the space between 19 and 20 on the thimble so that we have 0.469

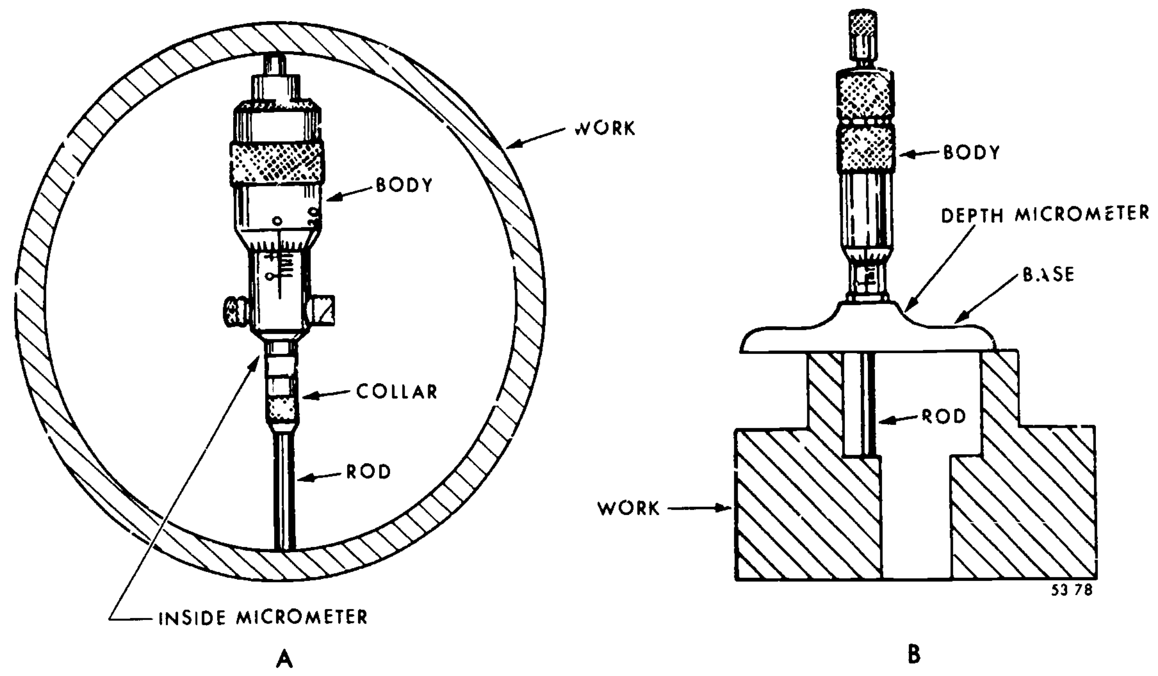


Figure 78. Inside and depth micrometers.

+ inch. Looking at the vernier scale, you find that line 7 on the barrel exactly matches a thimble graduation; thus the final reading is 0.4697 inch.

15-8. *Inside micrometer*. The inside micrometer is designed to measure internal dimensions. You use an *inside* micrometer, as shown in figure 78,A, to measure the inside diameters of pipe, tubing, and holes and distances between surfaces. The measurements obtained may be just as precise as those obtained with an outside micrometer. However, the nomenclature differs slightly from that of other micrometers, and the range of an inside micrometer is usually only 0.500 inch. The *body* of the inside micrometer, often referred to as the *head*, is composed of the units that are called the barrel and the thimble of an outside micrometer. Fully closed, most inside micrometers are 1.5 inches long. This prohibits the measurement of dimensions smaller than 1.5 inches with most inside micrometers. You use extension rods of various lengths when measuring dimensions larger than 2 inches. The maximum measurement possible is governed by the length of the longest extension rod available. The longest single extension rod in most sets is 6 inches; however, some inside micrometers permit the mounting of extension rods on both ends of the body, increasing the maximum measurement that can be obtained. Read inside micrometers in the same manner as you read outside micrometers except that the measurement must also include the length of the extension rod and the length of the body For example, using a 3-inch extension rod and an inside micrometer with a body length of 1.5 inches, you obtain a reading of 0.358. What is the diameter of the hole?

3.000 Length of extension
1.500 Length of body
0.358 Reading obtained
4,858 Actual measurement

15-9. *Depth micrometer*. Depth micrometers are used to measure the depth of holes or the distance from one flat surface to another, as shown in figure 78,B. The range of most depth micrometers is 1 inch. Extension rods allow the measurement of various distances: 1 to 2 inches, 2 to 3 inches, etc. Extension rods are installed in the thimble by removing the thimble cap, inserting the rod, and replacing the thimble cap to hold the rod in place. The extension rod extends through the center of the thimble and barrel and protrudes from the hole in the center of the base. The base is the reference surface in the same manner as is the anvil in an outside micrometer. The end of the extension rod serves as the other measuring surface, just as the end of the spindle does in an outside micrometer. A depth micrometer measures the distance from the bottom surface of the base to the protruding end of the extension rod. With the 0- to 1-inch extension rod installed and the thimble turned to the 0.000 position, the tip of the rod is flush (even) with the flat reference surface of the base. With the 1- to 2-inch extension rod installed, the rod extends exactly 1 inch from the base for a 0.000

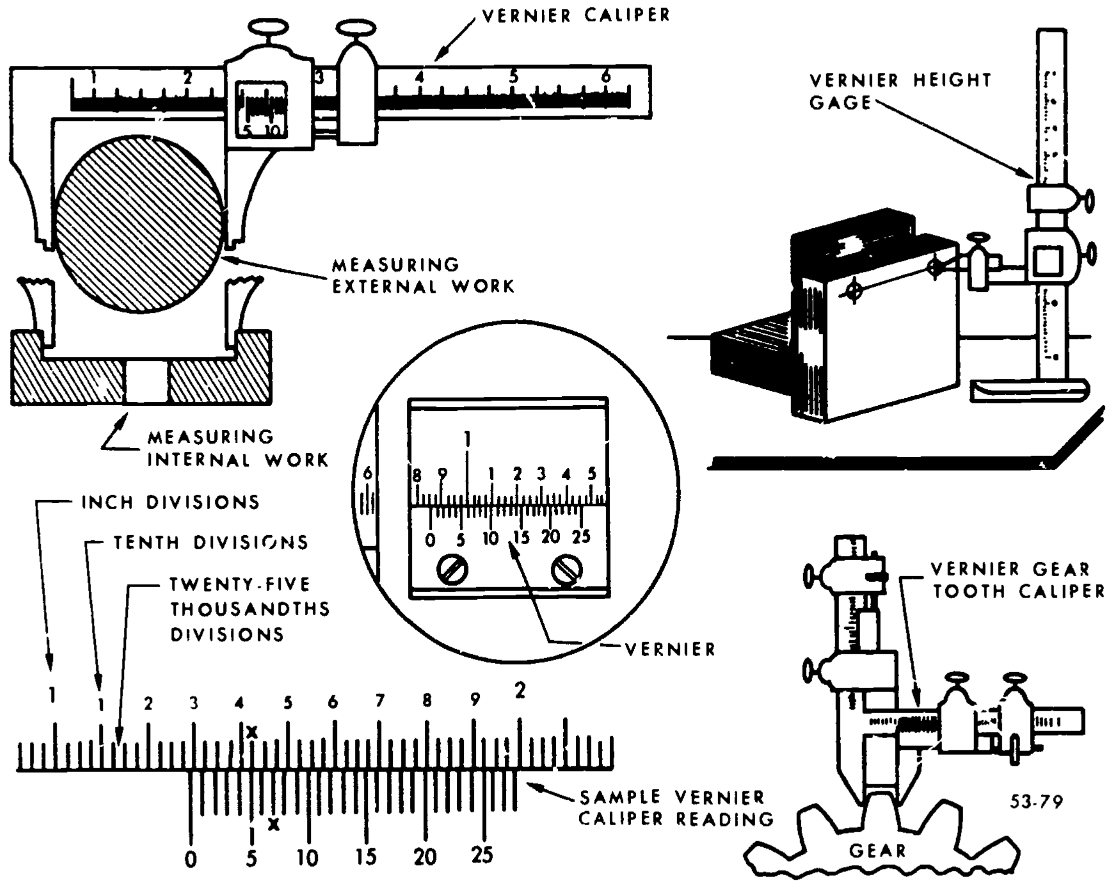


Figure 79. Vernier calipers and height gage.

reading. The graduations on a depth micrometer are reversed from those on an outside micrometer. The 0.000 reading on an outside micrometer is obtained with all of the graduations on the barrel covered by the thimble; while on a depth micrometer, the graduations are all exposed for a 0.000 reading. The measurement obtained with a depth micrometer must include the lowest value of the extension rod being used. For example, if a reading of 0.567 is indicated when a 0- to 1-inch rod is being used, the measurement is actually 0.567. If the same reading is obtained while using a 2- to 3-inch extension rod, the actual measurement is 2.567 inches.

15-10. *Care of micrometers.* Micrometers are precision measuring tools, and they must be handled with care if you are to maintain their accuracy. One of the greatest abuses to which you can subject an outside micrometer is to overtighten the spindle (turn the spindle too hard against the work or the anvil). Use only enough pressure to bring the anvil and the spindle against the surface of the object. A light pressure applied with the thumb and forefinger is adequate. Never tighten the micrometer enough to support its own weight. Many micrometers have a friction mechanism called a ratchet incorporated in the thimble. The ratchet is designed to slip if too much pressure is applied. A knurled ring, located in the frame near the spindle, is designed to act as a spindle lock on some micrometers. Always turn the spindle lock after positioning the spindle; this will allow you to handle the micrometer without disturbing the setting.

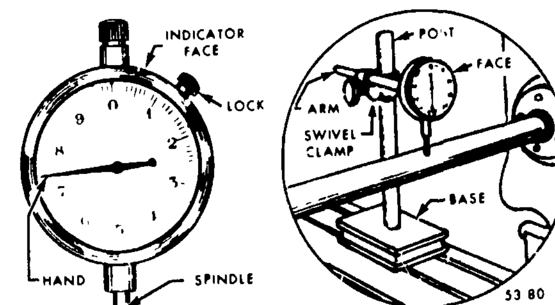


Figure 80. Dial indicator.

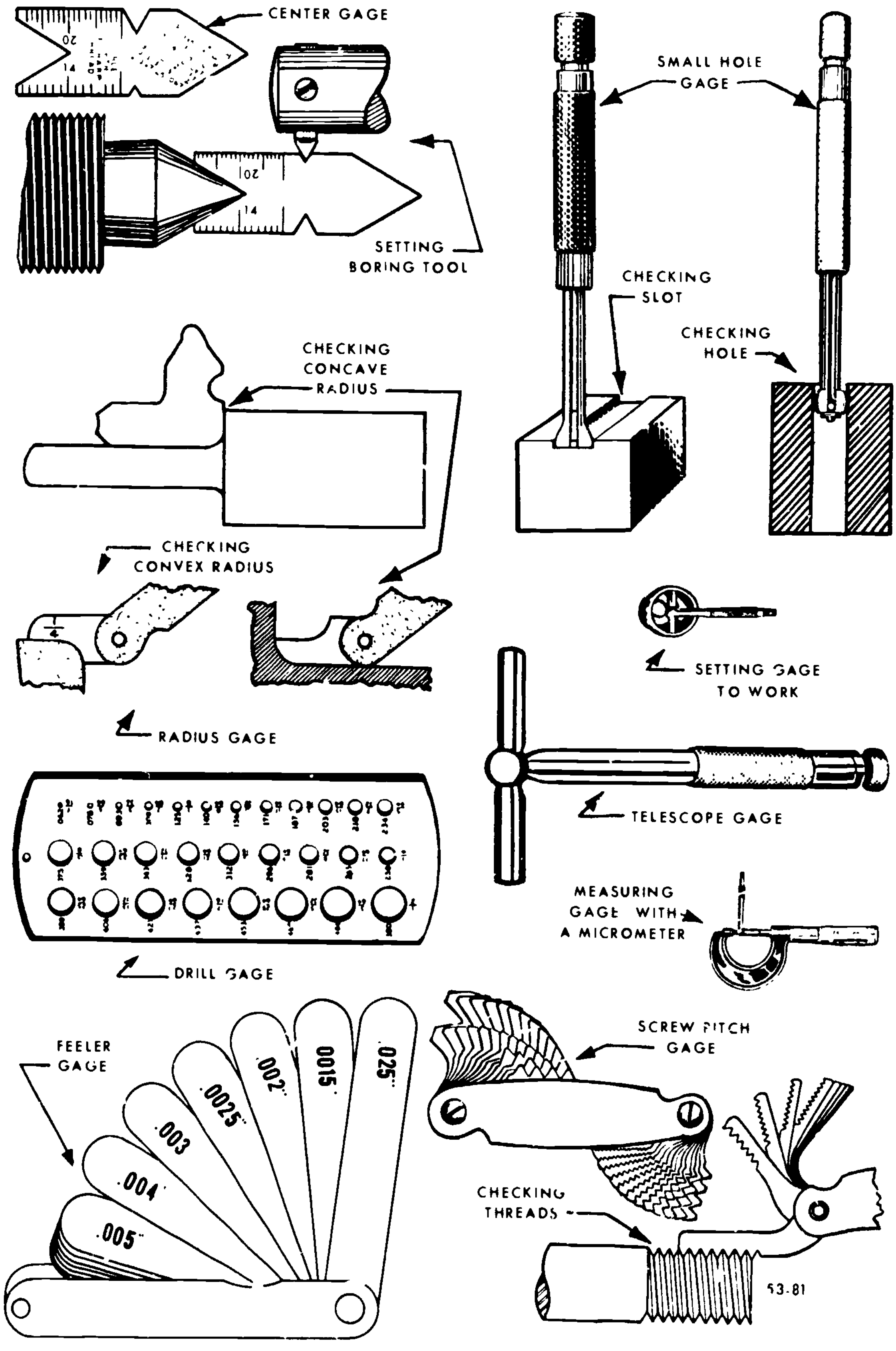


Figure 81. Various gages and their uses.

15-11. Never swing a micrometer by the thimble. When not in use, protect it with a very light film of oil to prevent rusting. Do not store a micrometer with the anvil and spindle closed tightly, as this may impair its accuracy. Keep the micrometer clean and make sure that the surfaces being measured are free of dirt, corrosion, and burrs.

15-12. **Vernier Instruments.** When dimensions of varying lengths must be measured to tolerances in thousandths of an inch, the *vernier caliper* is handier to use than the micrometer. The outside micrometer has only a 1-inch capacity, making it necessary to stock a variety of frame sizes. The vernier caliper capacity is limited only by the length of its scale, which may range from 2 inches in the pocket type to 12 inches or more. In addition, the vernier caliper may be used for both external and internal measurements, without having to use calipers to transfer the measurement. Figure 79 shows a vernier caliper and two other vernier instruments. One is a gear tooth vernier caliper. The other is a *vernier height* gage, which can be used for checking measurements or for precision layout work where tolerances must be held extremely close. All of these instruments operate on the same principle as the vernier graduations on a micrometer except that the vernier scale is calibrated in thousandths of an inch.

15-13. *Dial indicators.* A dial indicator, shown in figure 80,A, is a precision instrument which is used to determine the smoothness or concentricity of a surface. The linear movement of the spindle is amplified by a gear train and is translated into rotation of a pointer over a graduated dial so that a spindle movement of a thousandth of an inch or less can be read on the dial. The scale on the dial reads both to the right and left of zero and thus indicates when you have properly adjusted the instrument to the work. Figure 80,B, shows a dial indicator set up for checking an arbor.

15-14. *Gages.* A machinist usually has an assortment of gages with which he can obtain certain measurements more quickly and easily than with general-purpose measuring tools. Figure 81 shows some of the most useful of these gages.

15-15. A *center gage* is used mainly in grinding and setting thread-cutting tools or in checking the angle of a lathe center. In addition, you can use the printed scales (marked 14, 20, 24, and 32) to check the pitches of the four most commonly used threads.

15-16. A *drill gage* is a flat steel plate containing holes of various sizes under 1/2", each marked with the particular size in a gage number, letter, fraction, or decimal equivalent. You can quickly determine the size of any 1/2" or smaller drill by fitting the fluted end into the drill gage holes until you find the smallest one that will allow the drill to enter. In addition, most drill gages also list information pertaining to the correct relation between drill and tap sizes for commonly used machine screws.

15-17. ***Radius*** (or fillet) *gages* are curved templates, usually fastened together as a group, with which you can check the size and accuracy of concave and convex corners.

15-18. A *screw pitch gage,* as the name implies, identifies the number of teeth per inch on a screw, bolt, nut, pipe, or fitting. By trying to mesh the teeth of successive blades with the unidentified screw heads, you will eventually find one blade that fits exactly. The number on the gage blade indicates the correct thread pitch.

15-19. A *thickness* (feeler) *gage* is a group of steel blades, each of which is accurately ground to a different specific thickness, as marked on each blade, in thousandths of an inch. You can measure the amount of clearance or gap between adjacent surfaces by using either a single blade or a combination of two or more blades. This gage is frequently called a feeler gage because the accuracy of measurement depends upon a skilled sense of touch or feel.

15-20. A *telescope gage* can be used in places in which a micrometer or a vernier caliper would be awkward to handle. With it you can quickly and accurately obtain the inside measurements of slots or holes. This T-shaped tool consists of an adjusting handle and two plungers, one telescoping into the other, which can be locked by turning a knurled screw. Although the gage is not calibrated, you can easily read the measurement obtained by checking the telescope gage with a micrometer or a vernier caliper. When the telescope gage is too large, you can use a small hole gage to measure small holes or slots.

Introductory Machine and Bench Work

THIS FINAL chapter in Volume 1 covers basic information that you need in order to perform nearly all machining operations. It also covers the two most simple machine tools; the power hacksaw and the drill press. We will explain the machinability of metals, the use of cutting lubricants, the power hacksaw, the drill press and its operations, drill grinding, and layout and bench work.

16. Machinability of Metals and Cutting Lubricants

16-1. The machinability of a metal is the ease with which it can be cut or machined. Some metals are more easily machined than others. Of course, the cutting tool must be hard than the metal being machined. Cutting lubr s, or coolants, are used to make the machining of metals easier.

16-2. **Machinability.** Many factors determine the machinability of a metal, such as its composition and grain structure and the heat treatment it has undergone. Some metals, such as aluminum and magnesium, are naturally more machinable than others. When the composition of a metal is changed by adding alloying elements, its machinability may change. When a large amount of sulphur is added to carbon steel, it is easier to machine and is known as a free-cutting or free-machining alloy (SAE–1100 group). Adding tungsten, chromium, or nickel has the opposite effect, and the alloy formed is more difficult to machine. Some grain structures have a lower shear strength than others. This makes them machine more easily. When the grain structure is changed by adding alloying elements or by heat treating, the machinability is changed also. A piece of hardened steel has a grain structure that is more difficult to machine than the same piece after being annealed. As a general rule, nonferrous metals are easy to machine, the most notable exceptions being titanium and nickel.

16-3. Metals can be rated according to their machinability, and this is sometimes done, but industry generally has not found it practical to develop machinability ratings as an aid to the machinist. From experience and testing, it has been found that different groups of metals should be machined at speeds which fall within different cutting foot speed ranges. We will explain cutting foot speed in the sections on the power hacksaw and the drill press. The recommended cutting foot speed range is 80 to 110 for low carbon steels, 60 to 80 for medium carbon steels, and 50 to 60 for high carbon steels. These speeds emphasize the fact that the harder it is to machine a metal, the slower the speed at which it is machined. Cutting foot speeds for all types of metals are listed in *Machinery's Handbook* and in any machinist handbook. The machinist compensates for differences in the machinability of metals by regulating the speed at which a cutting tool cuts the metal and the rate at which it is fed into the work. Other factors that you must consider are the type of machining operation, such as drilling, boring, or reaming, and the type of cut, such as rough or finish.

16-4. **Cutting Lubricants.** If we were to make a list of machine shop villains, heat would be near the top. Heat causes cutting edges to fail, tools to dull, and poor finishes to be produced. Heat is produced by friction; thus, if we can reduce friction, we can help to prevent a temperature rise. Also, if we can remove the heat that is present, we can operate machines at faster speeds and obtain better results.

16-5. Cutting lubricants help to reduce the friction between the chip and the cutting tool, which reduces the amount of heat generated. These lubricants also help to cool the tool, thus lengthening its life, and the work, which prevents arpage, by carrying away the heat. They also improve finish and wash away chips. Flooding the work and tool with the cutting lubricant will usually give you the best results. High pressure is usually not required; a large volume of oil or coolant is more desirable, as shown in figure 82. Flooding is often not practical, however, because the machine is not equipped with a coolant pump or the process is too messy.

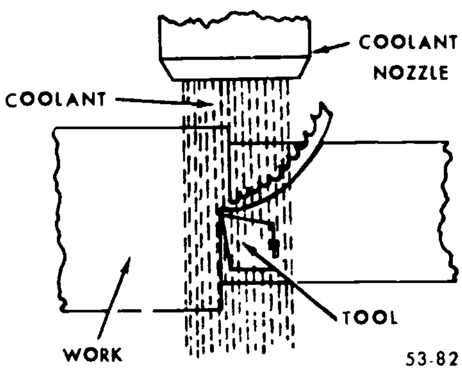


Figure 82. Application of coolant.

In such an event, apply the cutting oil to the work with a brush or oil can. Be sure to apply the cutting lubricant as close to the cutting edge of the tool as safety will permit. Use the cutting lubricants recommended in chart 9 for a given metal and machining operation.

16-6. *Lard* oil has been a favorite cutting lubricant in the past, but it tends to become rancid with age. Other types of fatty oils, such as peanut oil or olive oil, are often used. Fatty oils are often used for threading operations. Lard oil and other fatty oils are recommended for machining copper or copper alloys, since the more chemically active oils may stain the work.

16-7. *Sulphurized* oils are mineral or fatty oils with sulphur added. These oils are nearly black in appearance and have a strong odor. The sulphur helps to reduce friction and prevents the welding of the chip to the tool. The sulphur forms a sulphide film on the surface of the work. This film has a low shear strength which reduces friction. These oils are excellent when cutting pressures are high.

16-8. *Chlorinated* oils have about the same properties as sulphurized oils plus the added advantage of being lighter bodied. This makes them better coolants and "wets" the work and tool better. Chart 9 gives recommended lubricants for various metals and applications. Chlorinated oils form a chloride film, which produces the same results as a sulphide film. Avoid letting moisture come in contact with chlorinated oils. Hydrochloric acid may be formed, which may damage the work or the machine or injure you.

16-9. If cooling the work and the tool were the only requirement, water alone would be sufficient; however, water causes corrosion. To avoid this, use a soluble oil mixture instead. Soluble oil consists of a cutting oil and a substance that allows it to mix with water. This provides excellent cooling, provides some lubrication, and prevents the formation of corrosion.

16-10. Before you operate any type of metalworking machine, check to see whether or not the coolant reservoir is filled with the proper coolant. Take care to prevent the coolant from draining on the shop floor and creating a safety hazard, especially when you are sawing flat stock, pipe, or tubing. The coolant tends to travel along the top of flat stock and to drain onto the floor. Place flat pieces of scrap stock across the material to divert the coolant into the saw. Place containers under the ends of pipe or tubing to catch any coolant which may flow through.

17. Power Hacksaw

17-1. Before you can make or repair a part, you must have one or more pieces of metal of the correct length. Most metal stock received in the shop is in lengths that are much too long to be used "as is" and must be cut off to the length of the individual parts. This is usually done with a power hacksaw. Therefore, you should have a knowledge of the power hacksaw and its operation.

17-2. **Description.** It would be impossible to describe every make of power hacksaw used by the Air Force; however, each has certain characteristics in common with the others, and information that applies to one make usually applies to others. Do not be discouraged if the machine being described is different from those in your shop. If you can identify and understand the functions of the major components of one make, you can easily locate and identify their counterparts on a different make. The following information pertains to the major components of a typical power hacksaw. Refer to figure 83 for the location and illustration of these parts.

17-3. *Base.* The base of the saw usually contains a coolant reservoir and a pump for conveying the coolant to the work. The reservoir contains a series affles which cause the chips to settle to the bo. .i of the tank. A table which supports the vise and the metal being sawed is located on top of the base.

17-4. *Vise.* The vise is adjustable so that various sizes and shapes of metal may be held. On some machines the vise may be swiveled so that stock may be sawed off at an angle. The size of a power hacksaw is determined by the largest piece of metal that can be held in the vise and sawed.

17-5. *Frame.* The frame of the saw supports and carries the hacksaw blade. The machine is designed so that the saw blade contacts the work only on the cutting stroke. This action prevents unnecessary wear on the saw blade. The cutting stroke is usually the draw or back stroke. Many machines have a device that automatically turns off

Chart 9
Cutting Lubricants

Metal	Turning	Milling	Drilling	Tapping
Aluminun.	Mineral Oil with 10% Fatty Oil	Soluble Oil	Soluble Oil	Lard Oil
Alloy Steel	Sulphurized Mineral Oil	10% Lard Oil with 90% Mineral Oil	"	30% Lard Oil 70% Min. Oil
Brass	Mineral Oil with 10% Fat	Soluble Oil	"	10% to 20% Lard Oil with Mineral Oil
Tool Steels	25% Lard Oil with 75% Mineral	Soluble Oil	"	25% to 40% Lard Oil with Mineral Oil
Low Carbon	"	Soluble Oil	"	"
Copper	Soluble Oil	"	"	Soluble Oil
Monel	"	"	"	25% to 40% Lard Oil with Mineral Oil
Malleable Iron	"	"	"	Soluble Oil
Bronze	"	"	"	20% Lard Oil with 80% Mineral Oil
Cast Iron	Dry	Dry	Dry	Dry
Magnesium	10% Lard Oil with 90% Mineral Oil	Mineral Seal 0. 1	Light Mineral Oil	20% Lard Oil with 80% Mineral Oil

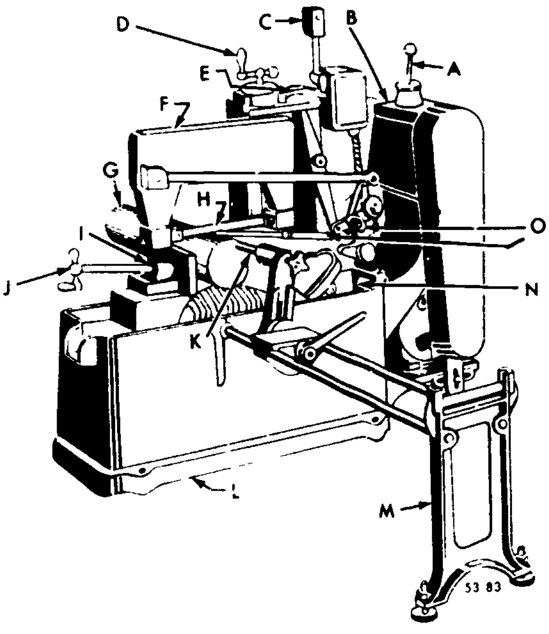
NOTE Never use soluble oil when machining magnesium. Water causes magnesium to burn more intensely.

53—131

the feed and returns the frame to the up position when the work is sawed through.

17-6. *Speed change mechanism.* The shift lever allows the number of strokes per minute to be changed so that a variety of metals may be sawed at the most effective speeds. Some saws have a diagram on them showing the number of strokes per minute when the shift lever is in different positions; others are merely marked "F," "M," and "S" (Fast, Medium, and Slow). CAUTION: On most saws the motor must be turned off before the shift lever is moved. The saw may be damaged if this is not done because the transmission is not synchromesh.

17-7. *Adjustable feed clutch* The adjustable feed clutch is a ratchet and pawl type mechanism that is coupled to the feed screw. The feed clutch may be set to a desired amount of feed in the -



A SHIFT LEVER
B TRANSMISSION
C. STARTING SWITCH
D. SAW FRAME RAISING LEVER
E. FEED LEVER
F. BLADE FRAME
G. WORK
H. SAW
I. VISE
J. VISE LEAD SCREW HANDLE
K. WORK STOP
L BASE
M. WORK STAND
N. COOLANT RESERVOIR
O. COOLANT NOZZLE AND VALVE

Figure 83. Power hacksaw.

sandths of an inch. Because of the ratchet and pawl action, the feed takes place at the beginning of the cutting stroke. The clutch acts as a safety device and permits slippage if too much feed pressure is put on the saw blade. It may also slip because of a dull blade or if too large a cut is attempted. This slippage helps prevent excessive blade breakage.

17-8. **Power Hacksaw Operation.** To saw with maximum efficiency and to reduce blade breakage and wear, (1) the sawing speed must be correct, (2) the work must be properly positioned and secured, and (3) the correct saw blade must be used. Since you will use the power hacksaw for nearly every job you do in the shop, you should thoroughly understand its operation.

17-9. *Speed and feed.* Selecting the proper speed and feed for sawing is important. If the saw is operated too slowly, time is wasted; if too rapidly, the saw blade wears excessively or it may break. Remember, however, that the condition of the hacksaw may require operating it at lower speeds and feeds than those recommended.

17-10. The *speed* (number of strokes per minute) of the hacksaw is determined by the type of metal being machined. A rule of thumb is: the harder the material, the slower the speed. Chart 10 gives the recommended speeds for various types of metals. CAUTION: Never attempt to saw hardened steel. The blade may snap, causing possible injury.

17-11. *Feed* is the distance the teeth penetrate the metal per stroke. It is usually stated in thousandths of an inch. For example, if the feed is set at 0.010, the depth of the saw cut is increased

CHART 10
RECOMMENDED SPEEDS AND FEEDS

MATERIAL	WIDTH OR DIAMETER IN INCHES					SPEEDS
	0-2	2-4	4-6	6-8	8-10	
MILD STEEL	.012	.009	.006	.003	.003	126-149
LOW CARBON	.012	.009	.006	.003	.003	101-120
TOOL STEEL HIGH-CARBON ALLOYS	.009	.006	.006	.003	.003	78-92
HIGH-SPEED STEEL STAINLESS STEEL	.009	.006	.003	.003	.003	63-75
BRASS ALUMINUM	.018	.012	.009	.006	.003	126-149
CAST IRON	.015	.012	.009	.006	.003	126-149

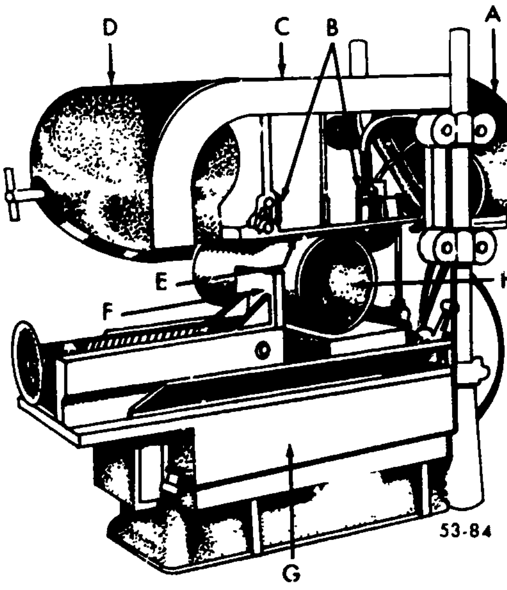
53-132

0.010 inch on each stroke. Improper feed is one of the most frequent causes of hacksaw blade failure. If the feed is too light, it can cause rapid dulling of the blade teeth. The use of too heavy feeds cause blade breakage, chipped out teeth, or crooked cuts. Chart 10 lists the recommended feeds for various metals.

17-12. *Power hacksaw blades*. Power hacksaw blades come in a variety of pitches (number of teeth per inch) and thicknesses. The thickness of blades varies from 0.030 inch to 0.100 inch. The thicker blades are intended for heavy duty applications involving heavy feeds. Hacksaw blades vary in pitch from 2½ teeth to 18 teeth per inch. A blade having 14 teeth per inch is a good choice for general work. An 18-pitch blade is preferred for sawing tubing and pipe. Blades with a pitch of 6 to 10 teeth can be used effectively with aluminum and brass. Coarse-pitched blades are used on softer materials and fin-pitched blades on harder ones, but never less than 3 teeth should be in contact. If the pitch is too coarse, the teeth may straddle the metal and be ripped off. When a hacksaw blade is replaced, the new blade must be placed under tension by tightening the blade tightening nut. If insufficient tension is applied, the blade will not saw straight, or it may break. The teeth must race in the direction of the cutting stroke. NOTE: After you replace a worn or broken blade on a partially sawed job, do not attempt to saw with a new blade in the original cut. The new blade will usually stick and be damaged or broken. The work should be turned over and a new cut should be started from the opposite side.

17-13. *Holding work*. Work to be sawed should be held by the vise. Before the vise is tightened, determine the location of the cut by measuring from the teeth of the blade to the end of the material or by alining a layout line with the edge of the saw teeth. Be sure that the cut is not made on the wrong side of the line. Because of "runout' (the blade not sawing straight down), 1/16 inch extra length should be left on the work piece to enable the ends to be machined square. Long or heavy pieces of stock must be supported near the ends by workstands so that the cut is made perpendicular to the work axis. If the stock does not extend the full length of the vise jaws, a piece of scrap of the same size is placed at the opposite end of the vise. This prevents the vise jaw from tightening unevenly and possibly working loose.

17-14. When a rectangular piece is sawed, the widest side should be parallel to the saw blade. This allows the maximum amount of material to be removed per stroke and spreads the wear over a greater number of teeth. If identical pieces are to be sawed from several bars of metal, it is often



A. DRIVE WHEEL COVER
B. SAW GUIDES
C. FRAME
D. IDLER WHEEL COVER
E. SAW BAND
F. VISE
G. BASE
H. STOCK

Figure 84. Horizontal bandsaw.

possible to clamp several pieces at once in the vise, thereby saving time.

17-15. Most saws have a work stop, such as the one shown in figure 83. This is a device that may be swung up and clamped at any desired distance from the saw blade. Identical pieces may be sawed by merely sliding the stock into contact with the stop gage and clamping the vise. This eliminates the need to measure or lay out each piece separately.

17-16. **Horizontal Bandsaw.** The horizontal bandsaw shown in figure 84, is not a power hacksaw. Some machine shops may not have a power hacksaw, but may have a horizontal bandsaw, which uses a saw band in the form of an endless band. It derives its name from the fact that the band travels horizontally, except where it passes around two wheels that are mounted on the saw frame. Rollers and guides twist the blade so that the teeth are correctly positioned for cutting. The endless saw blade provides a continuous cutting action and distributes the wear on the teeth evenly along its entire length. Since the kerf (the groove made by the blade) is very narrow because of the thinness of the saw blade, less stock is wasted with this type of cutoff saw than with any other type.

17-17. **Machine Lubrication.** Before operating any piece of machinery, you should lubricate it. Failure to lubricate machinery can cause rapid and costly breakdown. The correct type of lubricating oil to use can be found in technical orders, operator handbooks, or on the information plate located on the machine. Follow these recommendations carefully:

(1) Check all dipsticks and gages and maintain the oil at the proper level.

(2) Use progressive lubrication; start at one point on the machine and methodically work your way completely around it, being careful not to overlook any oil cups, oiling holes, or covered lubricating points. This will prevent missing any places that should be oiled, a mishap which may easily occur if you use a hit and skip system.

(3) Avoid overlubrication. A few drops of oil applied weekly does more good than flooding the machine once a month.

(4) Immediately wipe up any oil that has spilled or which runs out of the oiling points. This helps to keep the machine clean and also to prevent possible injury caused by persons slipping on oily spots. Use only covered metal containers in disposing of oily rags.

18. Drill Press Work

18-1. Look at all of the metal objects that you can see around you; most of them contain one or more holes. Because most hole drilling in AF machine shops is done on drill presses, it is important for you to know about them.

18-2. **Types of Drill Presses.** There are many types of drill presses, but we will discuss only the three general-purpose types most commonly used in the AF. They are the sensitive, the plain, and the heavy duty.

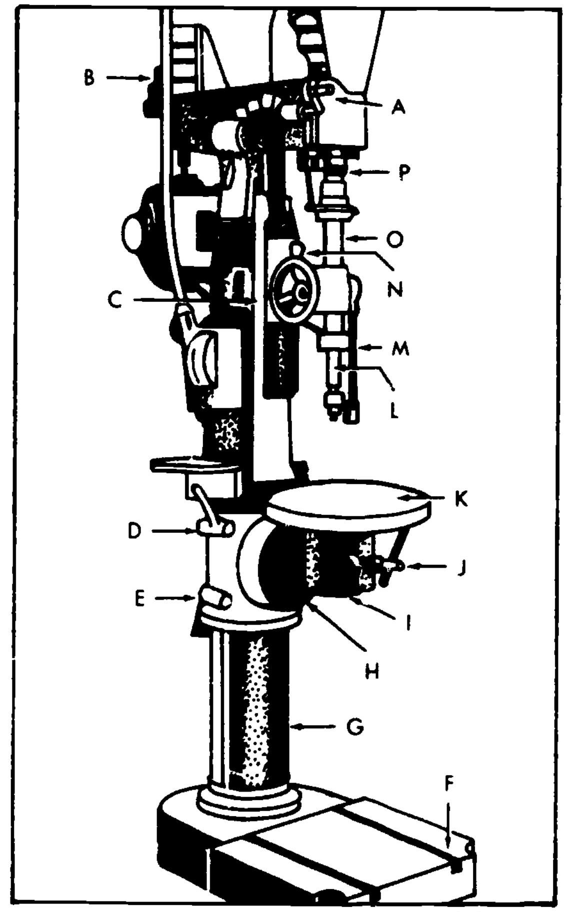
18-3. *Sensitive.* Sensitive drill presses are designed to drill very small diameter holes at very high speeds. They do not have power feeds. The operator feeds the drill into the work by hand and can "feel" the cutting action taking place. These drills may be bench or floor mounted, although the bench type is probably the more commonly used. The size of a sensitive drill is designated as the maximum diameter of work it is capable of drilling. For example, a 10-inch drill press would be capable of drilling a hole in the center of a piece of work 10 inches in diameter.

18-4. *Plain.* Plain drill presses are used for light and medium type work. The floor-mounted type, shown in figure 85, is found in many Air Force machine shops. The bench type, such as the one shown in figure 86, is also quite common. Some of these types are designed so that the drill may be fed by power. The sizes of plain drill presses are designated in the same manner as for sensitive types.

18-5. *Heavy-duty.* Heavy-duty drill presses are similar to plain drill presses except that they are more massive and rigid. They are intended for drilling large, heavy types of work. They are floor mounted, equipped with power feed, and usually operate at lower speeds to permit large diameter holes to be drilled. The same size designation used for sensitive and plain drill presses is used for heavy-duty types.

18-6. **Construction of a Drill Press.** Although they may differ in details, all drill presses are similar in construction. Most of the parts are common to all makes of machines. A knowledge of these parts and their uses will be helpful to you when you are working with your particular machine. Refer to figure 85 for illustrations of these common features as they are being discussed.

18-7. *Base.* The primary purpose of the base is to support the machine, but additional uses are sometimes made of it. Some drill presses have T-slots machined in the base so that objects may be fastened to it when the size, shape, or weight of



53-65

A. HAND CRANK
B. STEP CONE PULLEYS
C HEAD
D. TABLE CLAMP
E. TABLE CLAMP
F. BASE
G. COLUMN
H. TABLE ARM TILTING CLAMP
I. TABLE ARM
J. TABLE CLAMP
K. TABLE
L. SPINDLE
M. HAND FEED LEVER
HEAD CLAMP
O. SPINDLE SLEEVE
P. SHIFTER HANDLE

Figure 85. Floor-mounted drill press.

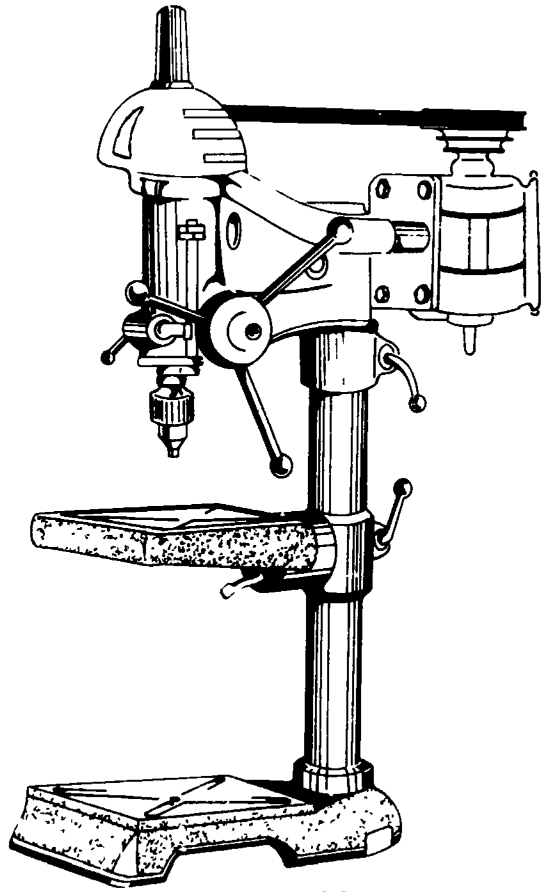


Figure 86. Bench-mounted drill press.

the item prevents using the table. Heavy-duty drill presses frequently have a coolant reservoir located in the base.

18-8. *Column.* The column is fastened to the base and supports the table arm and the head. It is usually cylindrical, but the heavy-duty type is often rectangular. Sometimes a column is made in two sections, with cylindrical bottom and rectangular top portions, such as the one shown in figure 85. Ways are sometimes machined in the column to act as a guide for the movable head.

18-9. *Head.* The head of the 'rill press carries the sleeve, spindle, and feed gears and is bolted permanently in place on many models. However, the head of some types can be adjusted vertically on machined ways located on the column. A counterweight located within the column and connected to the head with a chain is often used to balance movable heads.

18-10. *Spindle.* The spindle revolves within the spindle sleeve. The cutting tool or drill check is inserted in the lower end of the sleeve.

18-11. *Spindle sleeve.* The spindle sleeve, which contains the spindle, does not revolve but moves up and down. Pressure applied to the sleeve, either by hand or by power, causes the cutting tool to advance into the workpiece a few thousandths of an inch per revolution.

18-12. *Table.* The table of the drill press is supported on an adjustable arm on the lower section of the column. Most tables have T-slots machined in them to allow vises, clamps, or the work itself to be fastened to them. Heavy-duty types sometimes have a trough around the edges to collect cutting oil or coolant and to guide it back into the coolant reservoir.

18-13. There are four methods by which you may adjust a table for position, such as the one shown in figure 88:

a. Raise or lower the supporting arm and table. This provides for work of different heights.

b. Swing the supporting arm horizontally approximately 90° to either side of center. This allows you to swing the table out of the way when you are machining work mounted on the base.

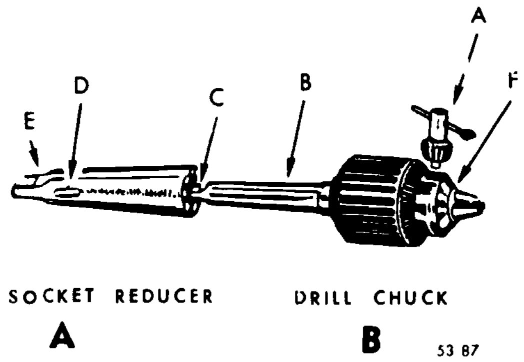
c. Pivot the table at its center and revolve it horizontally 360°. Some tables, such as the one shown in figure 89, do not have this feature.

d. Tilt the drill press table to the right or left 90° by loosening the table arm tilting clamps. This enables you to drill holes at any desired angle to the work surface.

18-14. **Holding Devices.** Satisfactory results can only be obtained when the cutting tool and the workpiece have been properly secured. Improper tooling or work setups can damage the to the work, or the machine and can result in injury to you. Thus a knowledge of tool and work holding devices will help you produce acceptable work and reduce the possibility of your being injured.

18-15. *Toolholding devices.* The spindle of a drill press has a tapered hole in the end which is used to hold a cutting tool or a drill chuck. This hole is usually reamed to one of the standard Morse tapers, and you can insert a tool having a tapered shank with the same Morse taper directly into the spindle hole. When you use a taper, it automatically alines the tool with the hole, and the wedging action of the taper holds the tool in place. To prevent taper shanked tools from slipping, a tang on the end of the shank, such as those shown in figure 87, fits into a slot in the spindle. When the tool shank is smaller than the spindle hole, socket reducers, shown in figure 87, which are also called drill sleeves, can be used to provide the proper fit. You may need more than one reducer, however, to fit very small shanks to larger holes.

18-16. Tools having straight shanks are held by a drill chuck, shown in figure 87. Drill chunks have three self-centering, movable jaws that grip the tool shank. You tighten and release the jaws



A. KEY
B. TAPERED SHANK
C. TANG
D. KEYWAY
E. TANG
F. DRILL CHECK

Figure 87. Socket reducer and drill chuck.

by means of a drill chuck key. CAUTION: Never leave the chuck key in the chuck. The spindle hole and the mating parts must be free of chips or burrs to insure that the cutting tools run true.

18-17. Tapered shank tools are removed from a drill sleeve or drill spindle with a drill drift. A regular drift, figure 88, part A, is a narrow, flat strip of steel with one tapered end. Insert it into the keyway with the taper against the tang of the tool and gently tap it with a hammer to remove the drill or drill sleeve. A safety drill drift, figure 88, part B, has a movable handle which is slid forcibly against the end of the drift blade, eliminating the need for using a hammer. CAUTION: To prevent possible damage, never allow the tool being removed to drop against the drill table or workpiece.

18-18. *Workholding devices.* To properly mount the work, you may use various holding and clamping devices. Work may be held in drill press vises, or clamped and bolted to the drill press table.

18-19. The drill press vise is the most commonly used work-holding device. Figure 89 shows two typical drill press vises. The work is usually supported on parallels to prevent holes from being drilled in the vise. After tightening the vise jaws, seat the work firmly on the parallels with a lead mallet. When you do rough work, you may substitute pieces of wood for the parallels. A drill press vise is normally used whenever the size and shape of the object to be held will allow it.

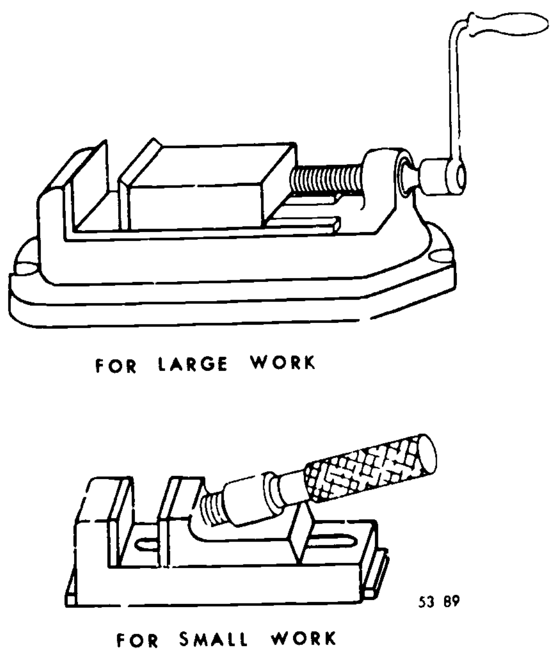


Figure 89. Drill press vises.

18-20. When the workpiece cannot be held in a vise, you will have to clamp it to the drill press table. Only a few of the many sizes and shapes of clamps are shown in figure 90. Figures 91 and 92 show some typical work setups using clamps. Note the use of blocks placed under the ends of the clamps. These blocks should be the same height as

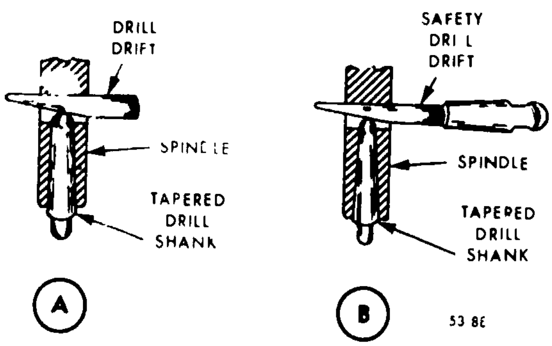


Figure 88. Using drill drifts.

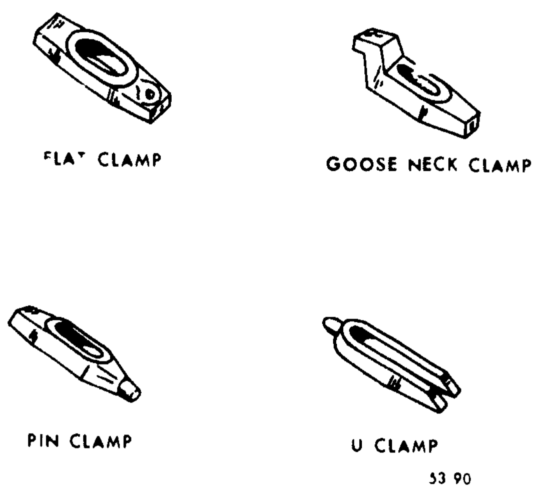


Figure 90. Common clamps.

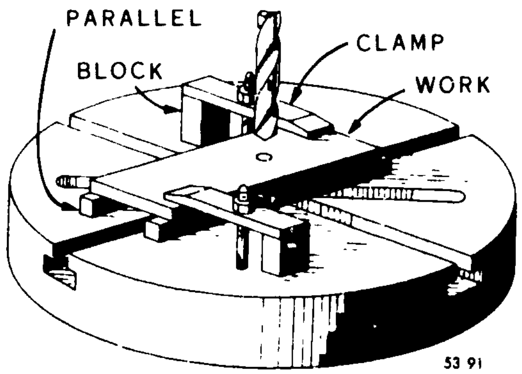


Figure 91. Work clamped to table.

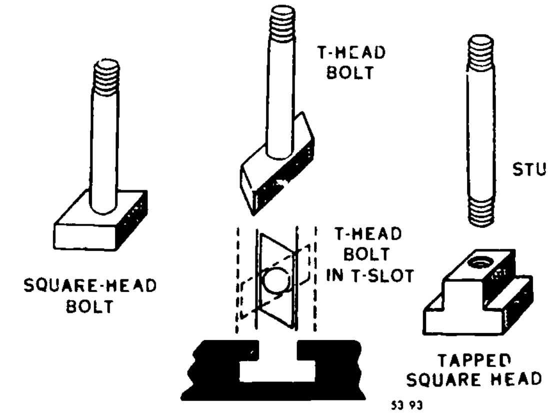


Figure 93. Holddown bolts.

the portion of the workpiece that the clamp contacts to insure that the clamping pressure is properly applied.

18-21. Three types of holddown bolts are commonly used to fasten objects to the drill press table: the cutaway T-bolt, the square head, and the tapped block and stud. The cutaway T-bolt, shown in figure 93, has an advantage over the square-head type in that it does not have to be slid in place all the way from the end of the T-slot. It can be dropped into the slot at any location and then turned 90° for use. Advantages of the tapped block and stud over the other types are its increased strength and variety of uses. It can be adapted to any size of job by using studs of any desired length. Bolts should be located approximately one-third of the distance from the work to the clamp leveling block.

18-22. **Feed and Speed.** Correct feed and speed are necessary because incorrect feed and speed result in poorly drilled holes, wasted time and material, and damage to the cutting tools or equipment. However, no hard-and-fast rules can be given regarding the correct feed and speed to use. You must take many factors into consideration, such as: (1) the hardness of the metal; (2) the depth of the hole; (3) the size, type, and condition of the cutting tool; (4) the operation being done; (5) the condition of the machine; (6) the work setup; and (7) the type of cutting lubricant being used. The actual feed and speed that you use will be your decision, after all the factors have been considered.

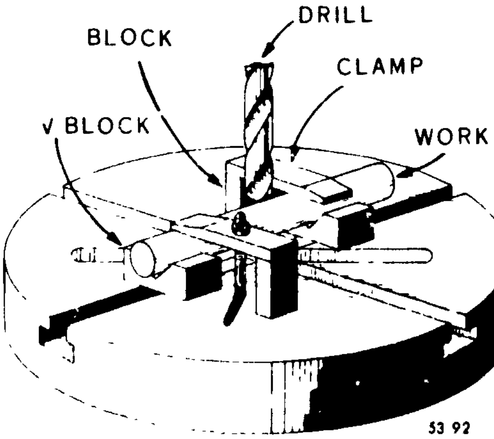


Figure 92. Work clamped to table.

18-23. *Feed.* Feed is the distance the cutting tool advances into the work per spindle revolution. When you are using hand feed, you must apply enough pressure to maintain a cutting action without forcing the drill. You should drill the hole, and not "punch" it out. Applying too much pressure chips the cutting edges and may even split the drill along the web. When the drill begins to emerge, you should reduce the pressure to prevent the work from "climbing up" the drill; this is especially true when you are drilling very thin pieces. Chart 11 gives the recommended feeds when power feed is used. The desired rate of feed is usually obtained by positioning levers according to a chart fastened to the machine. Most machines must be turned off before the feed selection levers are moved to avoid damaging the gearing.

18-24. *Speed.* The speed of a drill press refers to the number of revolutions per minute (RPM) of the spindle. It is necessary that you know how

CHART 11
DRILL FEEDS

TABLE OF FEEDS

DRILL SIZE (INCHES)	FEED (PER REV)
1/8 AND LESS	001 TO 002
1/8 TO 1/4	002 TO 004
1/4 TO 1/2	004 TO 007
1/2 TO 1	.007 TO 015
1 AND LARGER	015 TO 025

53-133

CHART 12
CUTTING FOOT SPEEDS

MATERIAL	CUTTING FOOT SPEED
LOW CARBON STEEL	80 TO 110
MEDIUM CARBON STEEL	60 TO 80
HIGH-CARBON TOOL STEEL	50 TO 60
STEEL FORGINGS	50 TO 60
STAINLESS STEEL	30 TO 40
SOFT CAST IRON	100 TO 150
HARD DRILLED CAST IRON	70 TO 100
MALLEABLE IRON	80 TO 90
ORDINARY BRASS AND BRONZE	200 TO 300
HIGH-TENSILE BRONZE	70 TO 150
MONEL	40 TO 150
ALUMINUM AND ITS ALLOYS	200 TO 300
MAGNESIUM AND ITS ALLOYS	250 TO 400
BAKELITE	100 TO 150
WOOD	300 TO 400

NOTE: CARBON STEEL DRILLS SHOULD BE RUN AT SPEEDS OF FROM 40 TO 50 PERCENT SLOWER THAN THOSE GIVEN ABOVE.

53—134

to determine the speed to use for drilling and then how to set the machine to obtain the desired RPM.

18-25. As a drill rotates, a point located on its outer surface (periphery) travels a certain distance in 1 minute of time. The exact distance traveled depends upon the distance around the drill (circumference) and the speed of its rotation (RPM). When this distance is changed from inches to feet, it is called the surface foot speed (SFS). It has been determined through experience and experiment that various metals machine best when a specific SFS is maintained. This desired SFS is known as the cutting foot speed (CFS). Chart 12 gives the CFS for various materials. The lower CFS is generally used for drilling and rough machining operations. The higher CFS is ordinarily used for finishing operations.

18-26. Since a change of drill diameter results in a change of circumference, the RPM must be changed in order to obtain a desired CFS. For example, a drill 1/2 inch in diameter has a circumference one-half as great as a drill 1 inch in diameter and revolves twice as fast as the latter to obtain the same CFS. You can see that you must take the diameter of the drill into consideration when you are calculating the spindle RPM to use. The most practical formula for determining spindle speed is:

$$\text{RPM} = \frac{4 \times \text{CFS}}{\text{drill diameter}}$$

For example, if you were to drill a 1/2-inch hole in low carbon steel, the formula would be as follows:

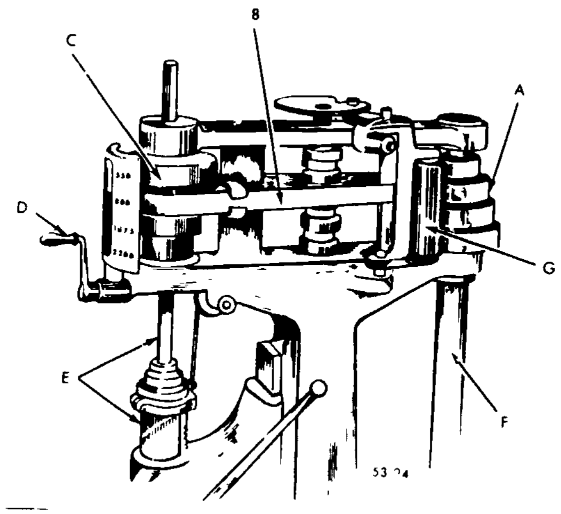
$$\text{RPM} = \frac{4 \times 80 \text{ (the CFS for low carbon steel)}}{0.500 \text{ (the decimal equivalent of 1/2 inch)}}$$

$$\text{RPM} = \frac{320}{0.500}$$

$$\text{RPM} = 640$$

18-27. The method that you use to obtain the desired RPM depends upon the type of drill press you are using. Some machines use a gear train located in the drill head to provide the various spindle speeds. You change the speed by positioning speed-change levers in accordance with a speed chart which is located on the drill head. You must turn off most machines before moving the levers.

18-28. Many machines use a step cone pulley and belt system to provide various speeds. Figure 94 shows a typical pulley and belt system. On this type of machine, by moving the belt from one pair of pulley steps to another with the spindle-speed change crank, you can select four spindle speeds. This type of machine must be operating when you



A. STEP CONE DRIVE PULLEY
B. DRIVE BELT
C. STEP CONE PULLEY ON SPINDLE
D. SPINDLE SPEED CHANGE CRANK
E. DRILL PRESS SPINDLE
F. SHAFT TO DRIVE MOTOR
G. BELT TIGHTENING PULLEY

Figure 94. Step cone pulleys.

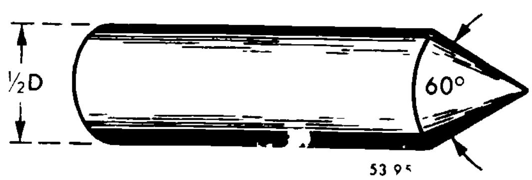


Figure 95. Center finder.

move the belt. Some machines, such as the bench drill press in figure 86, must be turned off. On the latter type of machine, tilt the motor to relieve the tension on the belt and move the belt by hand. Many drill presses have high- and low-speed ranges, which provide four additional speeds. On these drill presses you select the desired speed range either by moving a shift lever to the high- or low-range position or, on some machines, by positioning an electrical switch to the desired range setting.

18-29. **Drill Press Operations.** A variety of jobs can be done with a drill press, such as drilling, countersinking, counterboring, spot facing, reaming, and tapping. These operations are done quite frequently in any machine shop, and you will find the following information of great value to you throughout your career as a machinist.

18-30. *Drilling.* Drilling is the process of originating a hole by means of a twist drill. You can obtain good results only when you have followed the proper drilling practices.

18-31. The accurate locating of a drilled hole depends upon the accuracy of the layout and the proper positioning of the work under the drill. We will cover layout work, including locating the center of holes, in Section 20. Use a center finder to help locate the work under the spindle. Figure 95 shows one type of center finder that can be locally manufactured. Place the center finder in the drill chuck and position it so that the point barely clears the surface of the work. Adjust the drill press table until the intersection of the layout lines or the punchmark is directly below the point of the center finder. Check the alinement by viewing the work and center finder from at least two positions approximately 90° apart.

18-32. After you have correctly positioned the work under the spindle, center drill it. Do this with a combination drill and countersink, which is often called a center drill, as shown in figure 96. The tapered portion of the hole guides the drill and keeps it centered. If the final drill is very large, you may find it necessary to use one or more *lead* drills. A lead drill, sometimes called a pilot drill, is any drill that is slightly larger than the thickness of the web of the next drill to be used. In addition to keeping the final drill centered, a lead drill also reduces the power that you need to drill the hole. You should use cutting oil and remove the drill from the hole frequently, expecially when you are drilling a small diameter hole or a very deep hole. This "backing out" of the drill clears the chips from the hole and prevents drill breakage. It also allows the cutting oil to reach the cutting edge of the drill, where it does the most good.

18-33. Suppose you have to drill a hole in a piece of metal. First, lay out and center punch the location of the hole. Depending upon the size and shape of the work, mount it in a vise or clamp it to the table. Next, install a center finder in the drill

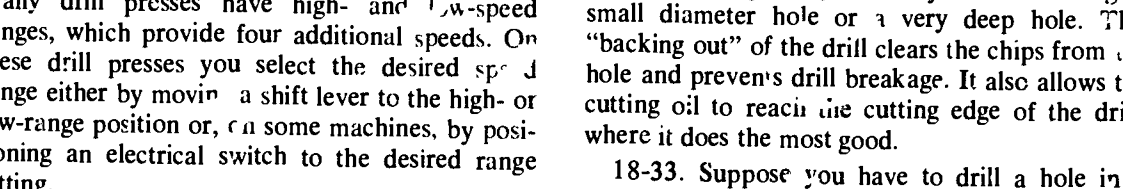


Figure 96. Combination drill and countersink (center drill).

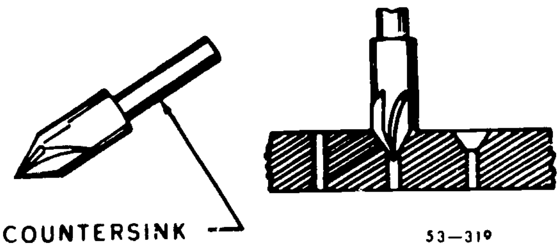


Figure 97. Countersink and countersunk hole.

chuck, and aline the punchmark under it. Now replace the center finder with a center drill. Set the machine for the correct spindle RPM, which is based on the center drill diameter and the metal's cutting foot speed. You can now start center drilling.

> NOTE: Stop the machine and check to insure that the location of the hole is correct. Reposition the work if necessary, then resume center drilling, using some cutting lubricant.

Center drill deep enough to insure that the drill will start correctly, but do not allow the countersink portion to exceed the diameter of the drill you are going to use. Replace the center drill with the drill you are going to use, making sure that it is sharp and correctly ground for the metal being drilled. Start drilling the hole, using hand feed, while you apply cutting lubricant. Remember to back the drill out occasionally to prevent chips from clogging the drill flutes. This may cause the drill to jam in the hole. If the hole is to be drilled completely through the metal, reduce the feed pressure as the drill begins to break through, to prevent damaging the drill.

18-34. *Countersinking.* A countersink is used to enlarge the end of a drilled hole with a large enough chamfer to enable a countersunk (shallow cone shape) bolt or screwhead to lie flush with, or just below, the surface of the work. It can also be used to chamfer a hole prior to tapping threads.

18-35. The countersink has teeth milled on its cone-shaped end at standard included angles of 82° and 100°, as shown in figure 97. If you don't

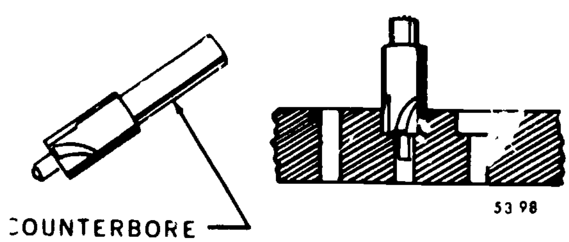


Figure 98. Counterbore and counterbored hole.

have the correct size countersink, you can grind a drill to the required angle to serve as a substitute.

18-36. Assume that you need to drill and countersink a hole in a cover plate that is to be secured with a countersunk flathead machine screw. You would first lay out the location of the hole that is to be drilled and countersunk. Select a drill that is a few thousandths larger than the diameter of the screw shank, aline it over the work, and drill the hole. Replace the drill with the proper size countersink. The countersink should be larger in diameter than the head of the screw. For best results, the RPM of the countersink should be slower than that used to drill the hole. You should also use a cutting lubricant. Begin the countersinking operation, using hand feed. If chatter occurs, reduce the speed. Raise the countersink clear of the hole and stop the drill press. Check the size of the counter-

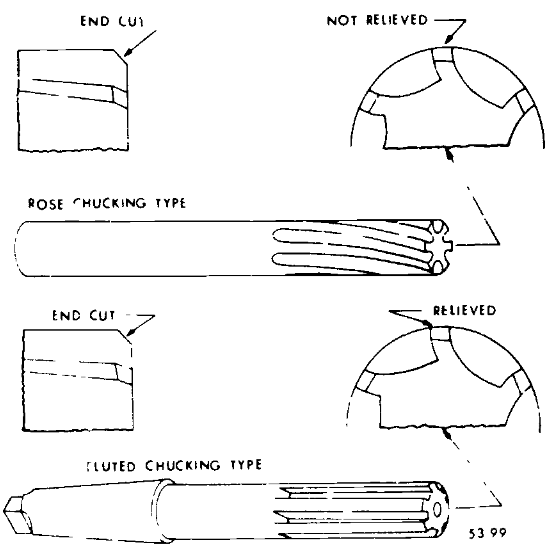


Figure 99. Machine reamers.

sunk portion by inserting the correct size screw into the hole. Continue countersinking the hole until the screwhead is flush with the surface of the work.

18-37. *Counterboring.* It is often necessary to enlarge the end of a drilled hole to receive the head of a bolt or screw. Use a counterbore, shown in figure 98, when it is necessary to make a cylindrical-shaped enlargement at the surface of a drilled hole to receive a fillister-head screw. Counterbores have interchangeable pilots to fit drilled holes and have a large flat bottom cutting section to make the cylindrical enlargement.

18-38. The counterbore is an end-cutting tool with three or more straight or spiral teeth relieved at the end to form cutting edges. The sizes of counterbores are designated by the diameter of

CHART 13
DRILL SIZES

Size	Decimal Equivalent	Size	Decimal Equivalent	Size	Decimal Equivalent
1/2	0.500	3	0.213	3/32	0.0938
31/64	0.4844	4	0.209	42	0.0935
15/32	0.4688	5	0.2055	43	0.089
29/64	0.4531	6	0.204	44	0.086
7/16	0.4375	13/64	0.2031	45	0.082
27/64	0.4219	7	0.201	46	0.081
Z	0.413	8	0.199	47	0.0785
13/32	0.4062	9	0.196	5/64	0.0781
Y	0.404	10	0.1935	48	0.076
X	0.397	11	0.191	49	0.073
25/64	0.3906	12	0.189	50	0.070
W	0.386	3/16	0.1875	51	0.067
V	0.377	13	0.185	52	0.0635
3/8	0.375	14	0.182	1/16	0.0625
U	0.368	15	0.180	53	0.0595
23/64	0.3594	16	0.177	54	0.055
T	0.358	17	0.173	55	0.052
S	0.348	11/64	0.1719	3/64	0.0469
11/32	0.3438	18	0.1695	56	0.0465
R	0.339	19	0.166	57	0.043
Q	0.332	20	0.161	58	0.042
21/64	0.3281	21	0.159	59	0.041
P	0.323	22	0.157	60	0.040
O	0.316	5/32	0.1562	61	0.039
5/16	0.3125	23	0.154	62	0.038
N	0.302	24	0.152	63	0.037
19/64	0.2969	25	0.1495	64	0.036
M	0.295	26	0.147	65	0.035
L	0.290	27	0.144	66	0.033
9/32	0.2812	9/64	0.1406	67	0.032
K	0.281	28	0.1405	1/32	0.0312
J	0.277	29	0.136	68	0.031
I	0.272	30	0.1285	69	0.0292
H	0.266	1/8	0.125	70	0.028
17/64	0.2656	31	0.120	71	0.026
G	0.261	32	0.116	72	0.025
F	0.257	33	0.113	73	0.024
E-1/4	0.250	34	0.111	74	0.0225
D	0.246	35	0.110	75	0.021
C	0.242	7/64	0.1094	76	0.020
B	0.238	36	0.1065	77	0.018
15/64	0.2344	37	0.104	1/64	0.0156
A	0.234	38	0.1015	78	0.016
1	0.228	39	0.0995	79	0.0145
2	0.221	40	0.098	80	0.0135
7/32	0.2188	41	0.096	--	------

53—135

the body and the pilot. For example, a 3/8–16 N.C fillister-head capscrew would have a body size of 3/8 inch, with a head size of 9/16 inch. In order to counterbore a hole to accommodate this screwhead, the pilot of the counterbore would be 0.372 inch in diameter and the body would measure 0.565 inch. The diameter of the body is usually 0.003 to 0.005 inch larger than the standard size of the hole to be counterbored. For example, a counterbore for a ½-inch hole may be from 0.503 to 0.505 inch in diameter. The bodies of counterbores fall into three general types: solid, removable pilot, and inserted blade. The pilot acts as a guide in centering the body of the counterbore over the hole to be worked. The diameter of the pilot is usually from 0.001 to 0.002 inch undersize to prevent it from enlarging the drilled hole If the pilot is too large, it will bind in the hole. If it is too small, it will cause the tool to cut oversized or irregularly shaped holes.

18-39. Assume that you have to drill and counterbore a hole to a specified diameter and depth. First, lay out the work, and drill the hole to the required diameter. Replace the drill with a counterbore of the correct diameter that has the correct size pilot. The counterbore RPM should be one-half to two-thirds of that for a drill of the same diameter. A cutting lubricant should be used. Using hand feed, begin counterboring. Check the depth of the counterbore frequently with a depth scale or depth micrometer, depending upon the accuracy required, until you reach the desired depth.

18-40. *Spot facing.* Holes in work are sometimes "spot faced," to remove surface irregularities and to provide a smooth flat surface that is square with the hole—and on which to seat a washer, screw, or bolthead. You spot face with a counterbore.

18-41. *Reaming.* Reaming a drilled hole is another operation that you may do on a drill press. Reaming is necessary because of the difficulty of drilling a hole to an exact diameter. Consequently, when you require great accuracy, you should first drill the hole slightly undersize, then ream it. Machine reamers can have either a straight shank, so that it can be held in a drill chuck, or a tapered shank with a tang, so that it can be held directly in the spindle of a drill press. The speed used for machine reaming is approximately one-half the normal drilling speed. A cutting lubricant should always be used. Machine reaming is not as accurate as hand reaming and the finished product is not as smooth. CAUTION: Hand reamers can be ruined if used in a machine.

18-42. A reamer is a fluted cylindrical side-cutting tool which is used to size drilled holes to precise diameters. They are also used to produce holes that are round, smooth, and straight. The teeth are unequally spaced around the body of the reamer to prevent chatter. Reamers with spirally cut teeth are more desirable than those with straight-cut teeth. They produce a slightly smoother and more accurate hole. This is because of the extra shearing action that the spiral flutes lend to the teeth. There are two types of machine reamers commonly used for reaming on the drill press. They are the rose reamer and the fluted chucking reamer.

18-43. The rose reamer has no clearance on the lands, since it does all its cutting with end teeth that are beveled at an angle of 45° and backed off as shown in figure 99. Since the rose reamer cuts only on the end, it has a slight back taper of about 0.001 inch per inch of length to reduce friction. The rose reamer does not cut a particularly smooth hole, as it has a tendency to gather stock on the cylindrical surface of the teeth. Intended mainly as a roughing reamer, it is sometimes made a few thousandths of an inch undersize. It will then be followed by a fluted chucking reamer or a hand reamer to finish the hole to size. The amount of metal that may be removed with a rose reamer ranges from 0.005 to 0.015 inch and sometimes to as much as 0.030 inch for roughing purposes.

18-44. The fluted chucking reamer, on the other hand, has clearance or relief along the entire length of its side-cutting edges or lands. It is used for finishing holes that are smooth and true to size. Since fluted chucking reamers have a greater number of teeth and thinner lands than rose reamers, as you can see in figure 99, they are intended for removing only small amounts of metal in order to accurately size and finish a hole. The amount of metal removed by this type of reamer usually varies from 0.003 to 0.005 inch, with 0.010 inch as a maximum.

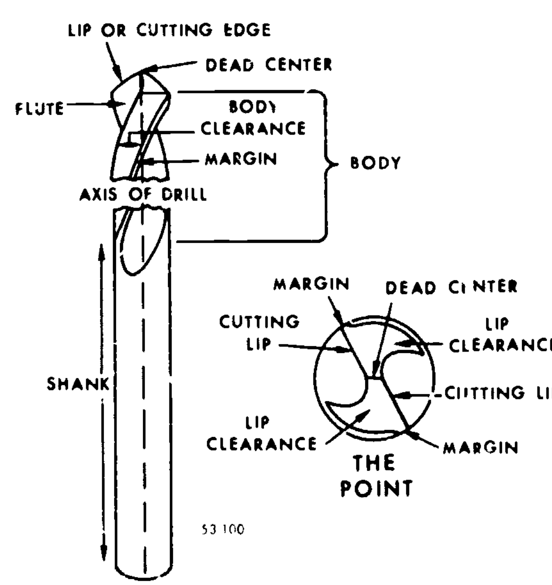


Figure 100. Twist drill.

18-45. *Tapping.* To save time and insure accuracy, many holes are tapped immediately after drilling, while the work is mounted on the drill press. One method of tapping in a drill press is to use a hand tap and wrench. Holes to be threaded must be drilled smaller than the tap size. Select the proper size drill by referring to a tap drill chart, which can be found in machinist's publications, such as *Machinist's Handbook*. If no chart is available, you can determine the correct size by applying the following formula:

$$\text{Tap drill size} = \text{O.D.} - \frac{1}{N}$$

O.D. is the diameter of the tap, and N represents the number of threads per inch of the tap. Then use the closest standard drill size. After drilling the hole, chamfer it with a countersink to a diameter no smaller than the diameter of the tap. Place the tap in the drilled hole, with a center finder inserted in the center hole on the shank end of the tap. Then apply downward pressure through the drill press spindle as you turn the tap with a tap wrench. Be sure to use the proper cutting lubricant. As you turn the wrench, apply pressure with the hand feed lever, but don't apply too much pressure, or you will break the tap. Occasionally, turn the tap backward to break the chips being formed. Release the spindle pressure and back out the tap. Once the tap begins feeding itself into the hole, eliminate the spindle pressure entirely. Then continue tapping the hole until the tap has cut threads to the required depth.

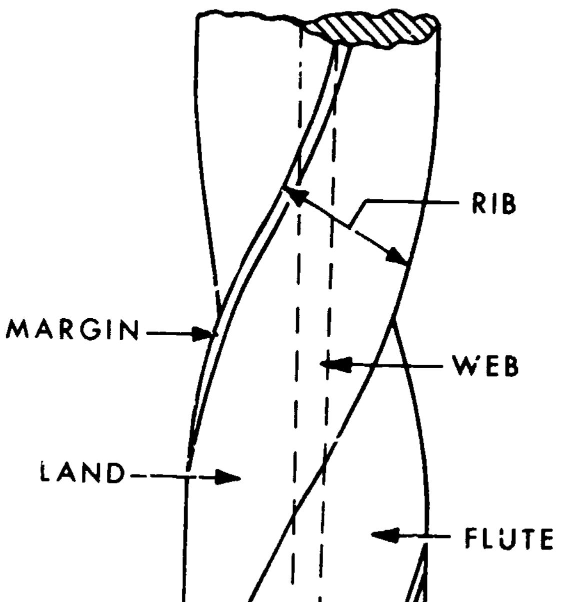


Figure 100. Body parts of a twist drill.

19. Drill Grinding

19-1. Drill grinding or sharpening is done on a pedestal grinder. Before you can sharpen drills you should have a basic knowledge of drills.

19-2. **Drills.** The drill is one of the most used and abused cutting tools in a machine shop. They frequently become dull and require sharpening. To be able to sharpen a drill correctly, you must be familiar with its parts, and know what the various clearance angles are and what the shape of the point should be.

19-3. *Series.* The three common drill series are the number, letter, and fractional series. Only the fractional series is larger than half inch. The number series starts with number 1 (0.228 inch) and ends with number 80 (0.0135 inch). The letter series starts with A (0.234 inch) and goes through Z (0.413 inch). The fractional series starts at $\frac{1}{64}$ inch and goes to $3\frac{1}{2}$ inches. Drills in this series are available by sixty-fourths up to $1\frac{3}{4}$ inches, by thirty-seconds up to $2\frac{1}{4}$ inches, and by sixteenths the rest of the way. Chart 13 lists the sizes of the fractional, number, and letter series drills.

19-4. *Parts.* A drill consists of three main parts: the shank, the body, and the point. Figure 100 shows the various parts of a drill. Refer to it as we discuss the various terms.

19-5. The shank is that portion of the drill that is held in a chuck or spindle. Several types of shanks are available; however, the straight and taper shanks are the more common. Most drills larger than ½ inch in diameter are made with Morse taper shanks. Sizes below ½ inch are usually of the straight-shank type. The tang (sometimes called the tongue) is the flat portion on the end of the taper shank. It fits the keyway in the holder or spindle and assists in driving and removing the drill. The tang is not intended to carry the entire torque load.

19-6. The body is that part of the drill between the shank and the point and consists of the following:

a. The axis is the longitudinal centerline about which the drill revolves.

b. The flutes of a twist drill are the two helical grooves cut in the body of the drill. They produce free-cutting lips and a space for chip removal and

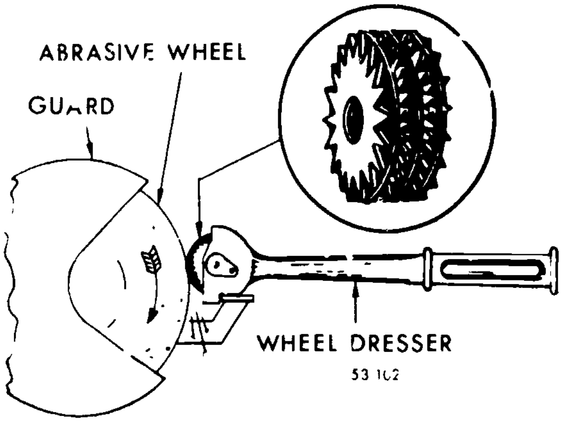


Figure 102. Huntington wheel dresser.

provide a means for the cutting lubricant to reach the cutting lips.

c. The web is the portion of the drill body between the flutes. It provides strength and rigidity to the drill, as shown in figure 101.

d. The body diameter is the diameter of the drill. It is measured across the margins just behind the drill point.

e. The body clearance is the portion of the rib that has been c away behind each margin to reduce the friction between the drill and the walls of the hole.

19-7. The drill point is the entire cone-shaped portion at the cutting end of the drill. The surfaces are ground to form the cutting edges.

a. The dead center, or chisel edge, is the sharp edge connecting the bottom of the flutes at the extreme end of the drill. It is formed by the intersection of the cone-shaped surfaces of the point and should always be in exact alinement with the drill axis.

b. The cutting lips are the cutting edges of the drill and extend from the chisel edge to the periphery.

c. The heel of the drill is the portion of the point back of the lips or cutting edges.

d. The lip clearance is the relief which is ground on the point of the drill extending from the cutting lip back to the heel. It allows the cutting edges of the drill to enter the metal without interference.

19-8. **Sharpening.** The sharpening of a drill is a three-step operation. First the abrasive wheel must be prepared, then the angles to be ground must be determined, and finally the angles are ground and checked.

19-9. *Preparing the abrasive wheel.* Before grinding a drill, you should dress the abrasive wheel, and, if necessary, true it. The terms "dressing" and "truing" are frequently confused. Dressing is the reconditioning of the abrasive surface of

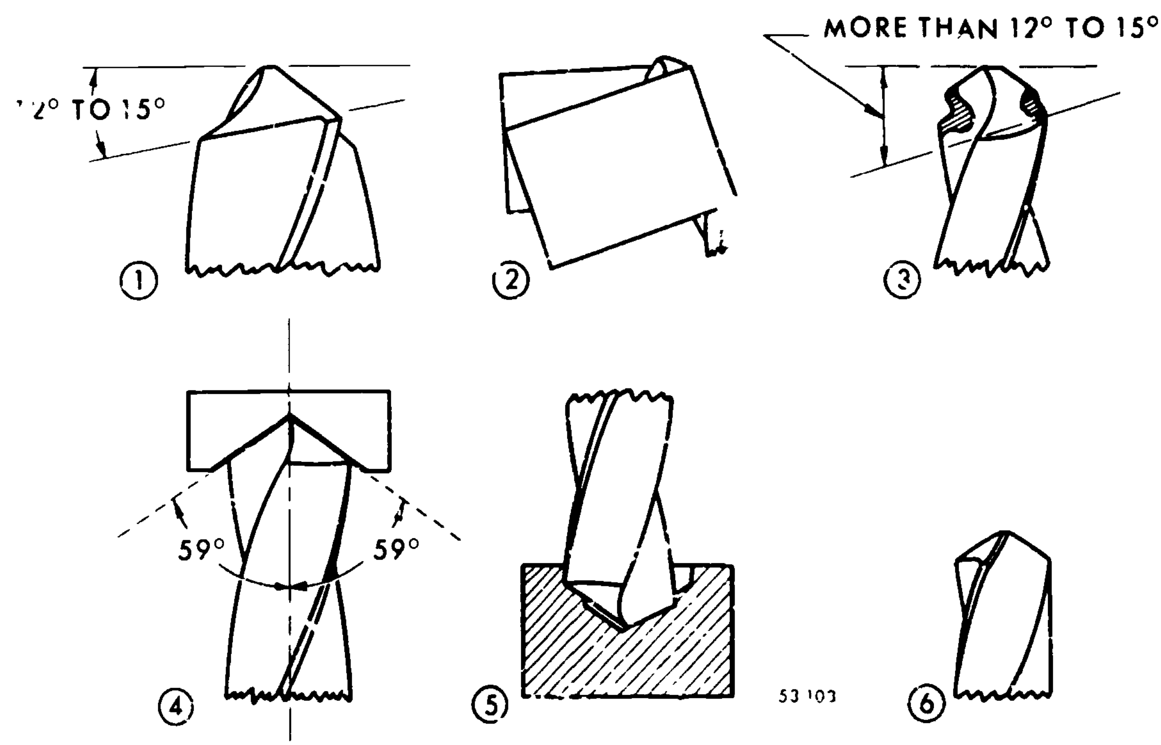


Figure 103 Drill point.

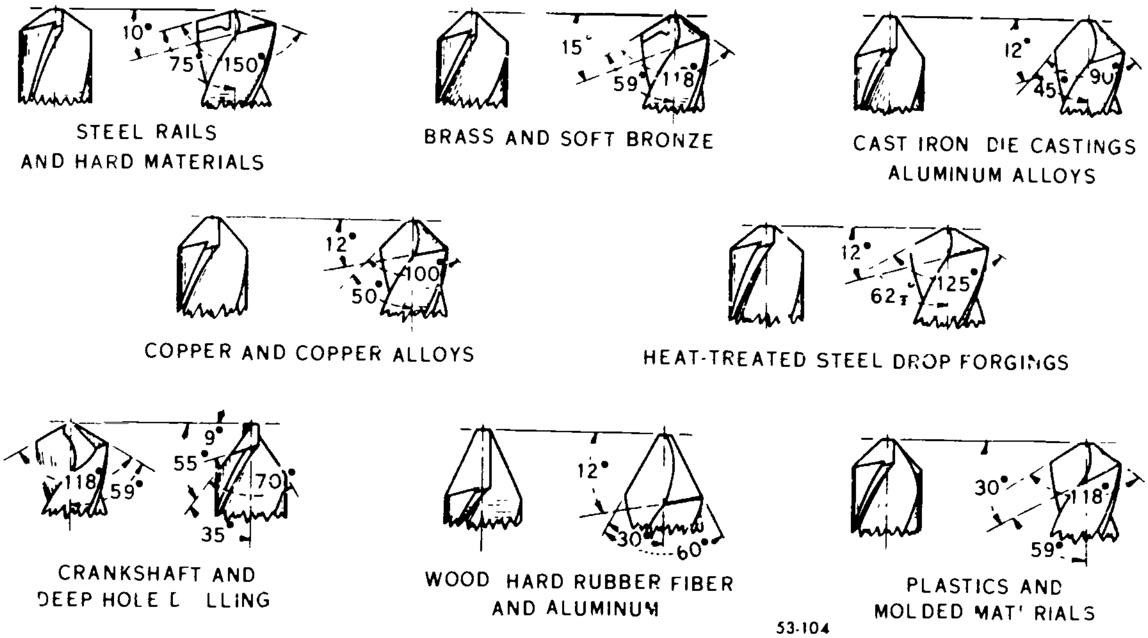


Figure 104. Drill point shapes.

a wheel that has lost some of its cutting ability. This is caused by glazing or loading up (filling the spaces between abrasive particles) or dulling the abrasive particles. Truing is restoring the abrasive wheel to its correct geometrical shape in relation to its axis. Truing is not required as frequently as dressing. The Huntington type dresser, which consists essentially of a number of circular metal cutters mounted on a spindle in a holder, is the most commonly used type of offhand dressing tool. Figure 102 shows this tool in use. Here the dulled abrasive grains and any loading of metal or foreign material are being removed so that sharp grains are being presented to the work. Before using the wheel dresser, position the tool rest so that the legs of the dresser may be hooked over it, as shown in figure 102. CAUTION: Be sure that the grinder has been turned off before you attempt to loosen the tool rest. After positioning the tool rest, turn the grinder on and bring the dresser into contact with the wheel. Never stand in front of a grinding wheel until after it has been running for several minutes. It may possibly disintegrate when it is first turned on. Also, never operate a grinder without wearing approved goggles or a face shield.

19-10. Pass the wheel dresser back and forth across the face of the abrasive wheel until it has been properly dressed and trued. Too little pressure will cause excessive sparking and rapid wearing of the dresser cutters and should be avoided. After completing the dressing, turn the grinder off and position the tool rest not more than 1/8 inch away from the wheel surface.

19-11. *Determining drill angles.* Drills with a lip clearance angle of 12° to 15°, figure 103, part 1, and an included angle of 118° (59° + 59° = 118°), figure 103, part 4, are used for most general drilling operations on carbon and soft alloy steel. Harder materials require less clearance than the softer ones. Too much clearance for the hardness of the material causes a rapid breakdown of the cutting edge, as shown in figure 103, part 3. Sharper included angles give faster, smoother production with soft materials. Flatter points have longer wear on drills used on hard, tough steels. Drills have a tendency to "hog-in" (grab) when drilling brass. Grinding the face of the cutting edge parallel to the drill axis, as shown in the illustration for brass and soft bronze in figure 104, helps

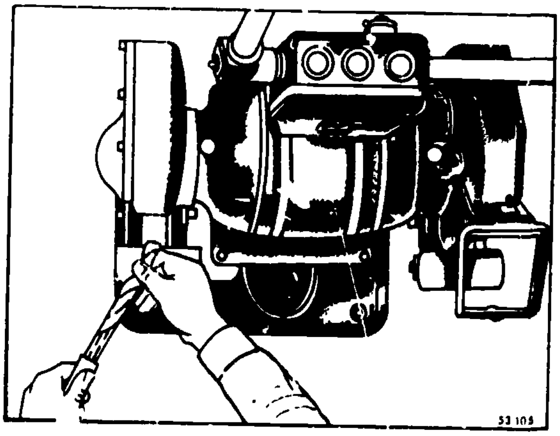


Figure 105. Positioning a drill for grinding.

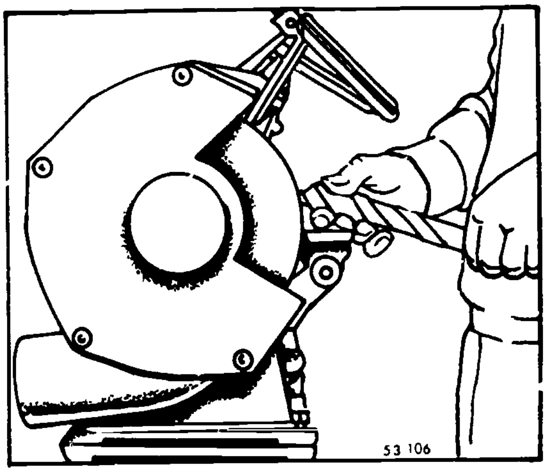


Figure 106. Grinding a drill.

reduce this problem. Drills used to drill any kind of thin metal should also be ground this way. As the drill is shortened by repeated sharpening, the web grows thicker. Maximum ease of penetration and wear resistance can be obtained by thinning the web to its original thickness. A "notched point," as shown in figure 104, is helpful when you drill deep holes. Figure 104 shows the recommended angles for various materials.

19-12. *Grinding and checking angles.* The actual grinding operation is relatively simple. Position the drill for the desired cutting edge angle. Using your fingers below the drill as a pivot, push the shank down, as shown in figures 105 and 106. This grinds the cutting edge angle and the clearance angle at the same time. You need not rotate the drill except for very large sizes. First grind one cutting edge, then the other, as many times as necessary, until the grinding is completed. Make frequent checks to insure that the proper angles and lengths of cutting edges are being maintained. NOTE: Never raise the shank of the drill higher than the cutting edge while you are grinding, or a negative angle will be produced.

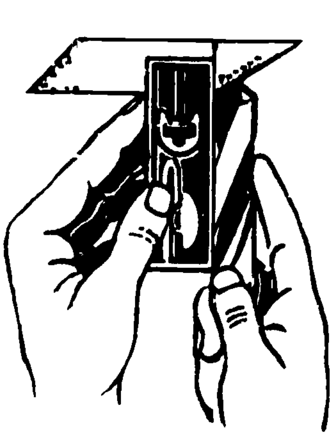


Figure 107. Drill grinding gage.

19-13. To sharpen a drill accurately, you must know how to check angles. You can check the cutting edge angle with a drill grinding gage, as shown in figure 107, or with the protractor head and blade, as shown in figure 108. You can also check the length of the cutting edge with these same tools. Both lips must be of equal length or oversize holes will be produced, or the drill may break. Check the clearance angle by using a clearance gage made of paper. Use a strip of paper 3 inches wide and 8½ inches long. Place a mark on the margin 1¾ inches from the lower side and bring the upper right-hand corner into contact with it, as in figure 103, part 2. Insert the drill in the gage and compare the clearance angle to the angle of the paper strip.

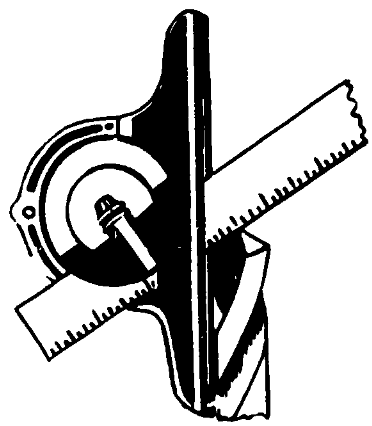
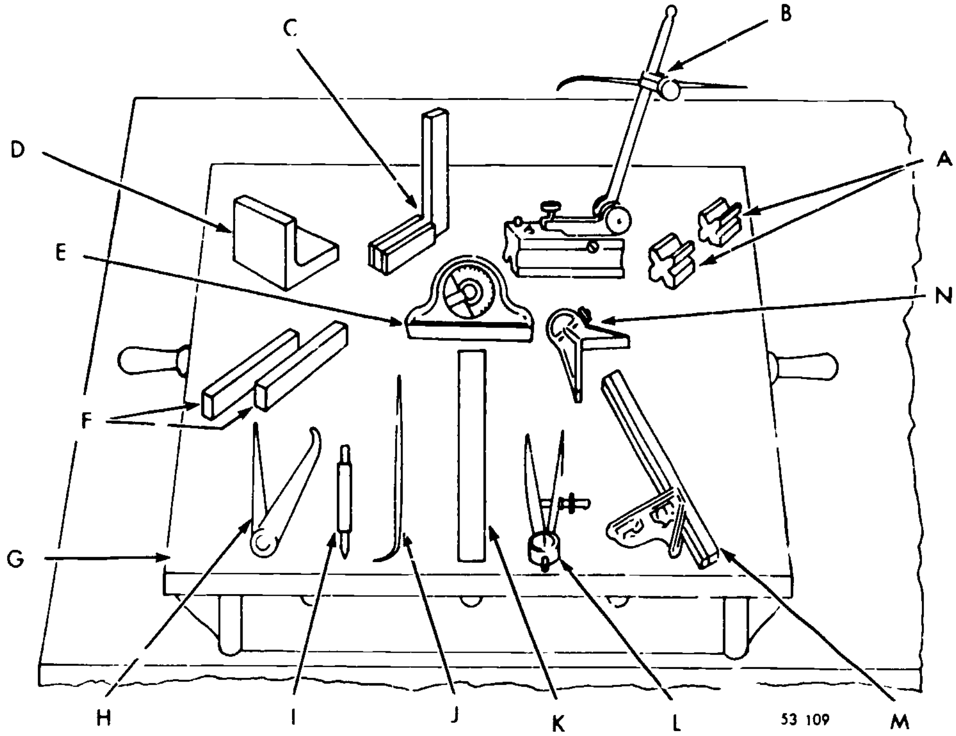


Figure 108. Protractor head and blade.

20. Layout and Bench Work

20-1. **Work Layout.** You will be required at times to lay out work prior to machining it. Laying out the work is planning the work on the surface of the material. It is the scribing (marking) of lines which indicates the boundaries, centers, and other locations on the object so that you are able to machine it to the desired size and shape. The care with which you do layout will determine the accuracy of the finished work.

20-2. **Layout Tools and Uses.** No matter what type of layout you do or what the object looks like, you will have to use layout tools. The shape of the object and the accuracy required will determine which tools to select. By knowing what the various



A. V-BLOCKS
B. SURFACE GAGE
C. TOOLMAKERS TRY SQUARE
D. ANGLE PLATE
E. PROTRACTOR HEAD
F. PARALLELS
G. SURFACE PLATE
H. HERMAPHRODITE CALIPER
I. CENTER PUNCH
J. SCRIBE
K. RULE
L. DIVIDERS
M. SQUARE
N. CENTER HEAD

Figure 109. Common layout tools.

tools are and what they are used for, you will be able to select the proper tools for laying out any given object. Figure 109 shows some common layout tools. The following tools are the ones most often used to lay out work.

20-3. *Layout compound.* To lay out work, you scribe lines on a layout compound which has been applied to the surface of the work. A commerical layout blue dye is most often used. This is a liquid that dries rapidly, leaving a glare-resistant dark blue film on the work. Lines scribed through this film show up distinctly. Since the fluid evaporates rapidly, you should keep the container tightly closed when you are not using it. Apply a thin coating because the compound tends to flake or produce ragged lines when it is applied too heavily. Common chalk is often used to lay out rough finished surfaces. Regardless of the type of compound you use, keep the surface clean and free of oil and remove all burrs with a file or oilstone to prevent inaccurate measurements and possible injury.

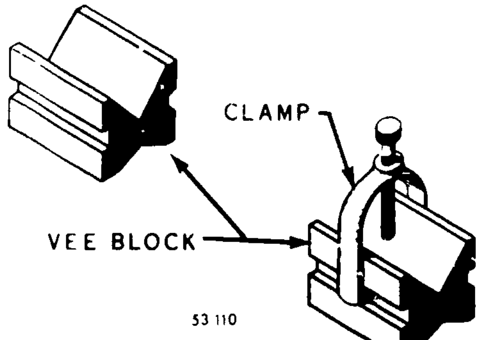


Figure 110. V-blocks.

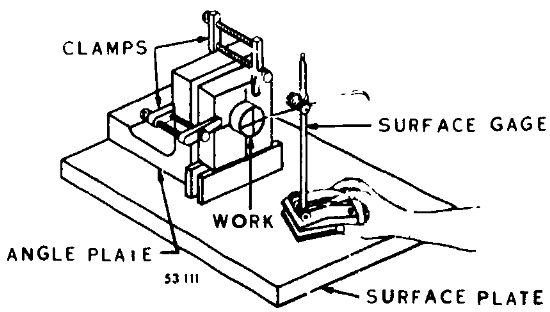


Figure 111. Use of angle plate in layout.

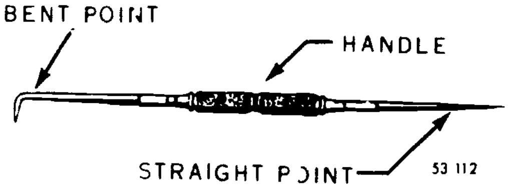


Figure 112. Scriber.

20-4. *Surface plate.* A surface plate, shown in figure 109 is usually a cast iron plate with a carefully machined top surface. It is used primarily for layout work. Better grades have the top surface hand scraped to produce an absolutely flat, smooth surface. They are extensively ribbed to prevent warping. Marble is sometimes used as surface plate material. Surface plates are expensive; therefore, be careful to prevent damage to them. Here are some of the precautions to be taken:

a. Do not allow items to drop on the plate; lower them gently until contact is made.

b. Work placed on the plate should be free of burrs.

c. Place only required items on the plate. Never place wrenches, hammers, etc., on it.

d. Do not perform hammering or punching operations while the work is on the plate.

e. When a surface plate is not in use, apply a light film of oil on it to prevent rust; and cover it with a heavy felt pad, followed by a sheet metal or wooden cover to prevent accidental damage to the plate.

20-5. *Parallels.* Parallels, shown in figure 109, are bars of steel or cast iron that are machined so that their opposite surfaces are parrallel to each other. Their adjacent surfaces are usually at a right angle to each other. Parallels are available with adjacent surfaces at other angles. Parallels are used when projections on the work prevent setting it directly on the surface plate. They are also

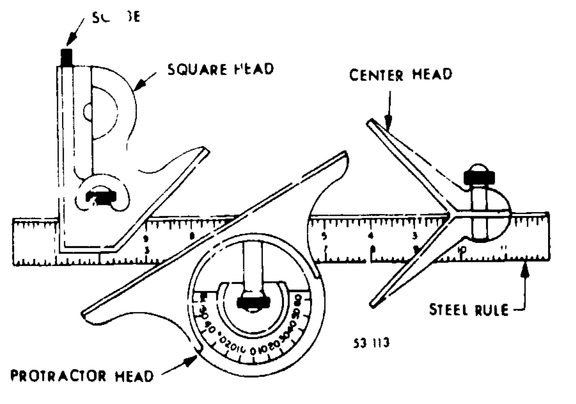


Figure 113. Combination set.

used when it is desirable to raise items above the surface and still maintain parallelism.

20-6. *V-blocks.* V-blocks are used to prevent the rolling or moving of round stock. They have 90° V-shaped slots on the top and bottom, usually of different depths, as shown in figure 110. The material is placed lengthwise in the slot and may be secured in place with a clamp, if this is desired.

20-7. *Angle plates.* Angle plates, shown in figures 109 and 111, have surfaces accurately machined at a right angle to each other. When it is desirable to mount the work at a right angle to the surface plate, it may be clamped against the angle plate, as in figure 111.

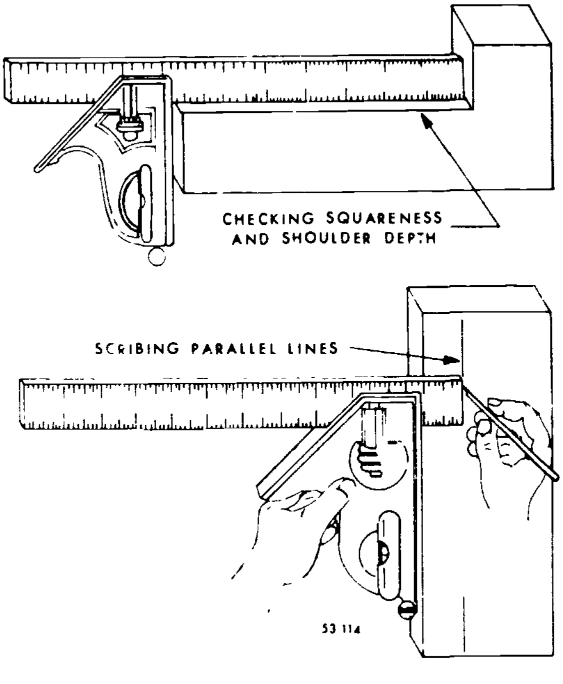


Figure 114. Use of square head.

20-8. *Scribers.* Scribers, shown in figure 112, are used to mark layout lines through bluing. Keep the point of the scriber sharp by means of an oilstone so that thin, accurate lines may be drawn. One accurate line should be made. Avoid going back annd forth with the scriber. This action usually results in a series of closely spaced lines which are inaccurate. Too much pressure will cause the scriber to scratch the surface of the work.

20-9. *Combination set.* The combination set, shown in figure 113, consists of four units: square head, protractor head, center head, and scale, or "blade." With the attachments removed, the blade has a 12-inch scale and may be used for measuring or as a straightedge. All three attachments are designed to slide along the blade and

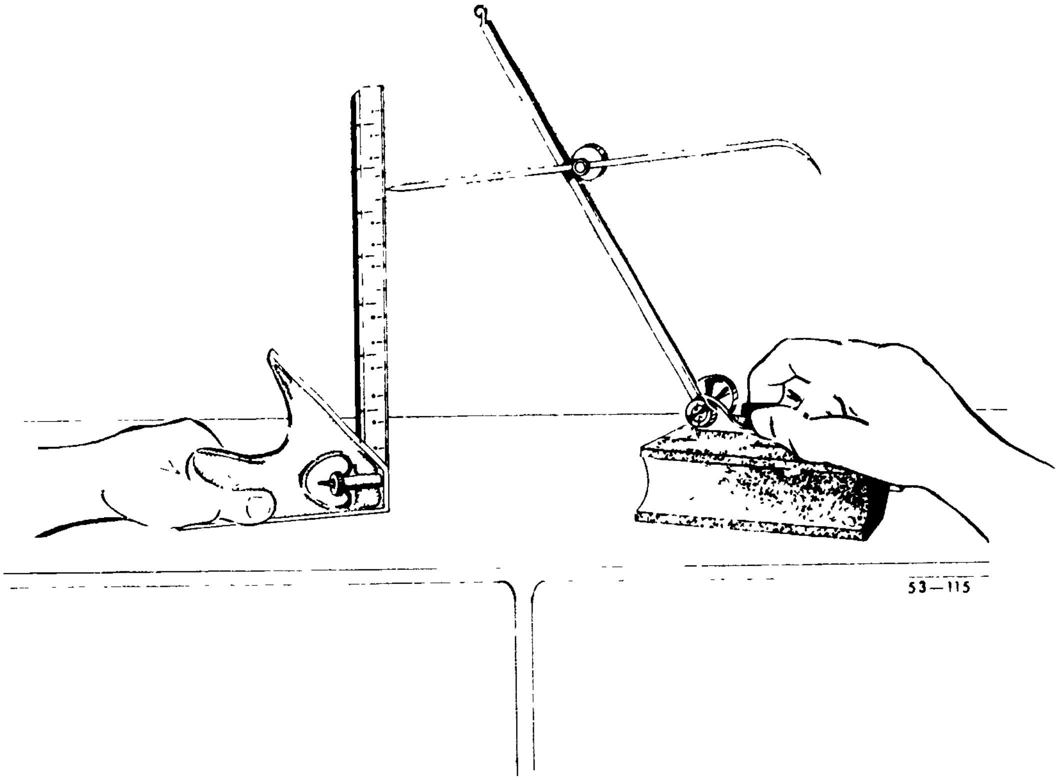


Figure 115. Setting surface gage to square head.

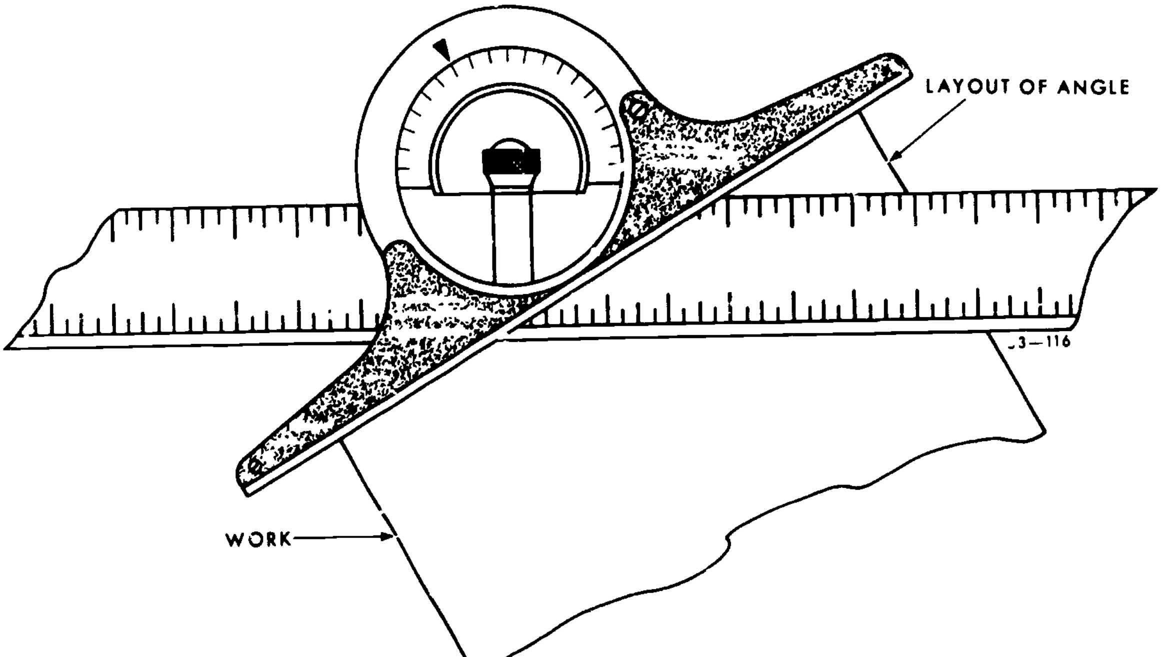


Figure 116. Use of protractor head.

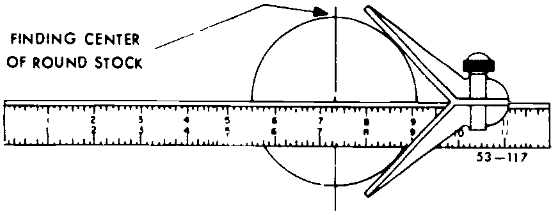


Figure 117. Use of center head.

clamp rigidly in any desired location. By clamping the square head to the blade, you can perform the operations shown in figure 114: scribing lines parallel to an outside edge and scribing perpendiculars to such lines or edges.

20-10. Figure 115 illustrates how to check the height adjustment of a surface gage. The other edge of the square head forms a 45° angle with the blade for use when you need to measure or check such an angle; a small spirit level contained in the head aids in leveling.

20-11. You should use the protractor head when you are measuring or laying out angles other than 45° or 90°, as shown in figure 116.

20-12. The center head provides a rapid method of locating the center of cylindrical and square objects. Figure 117 shows how, by clamping the center head to the blade and placing the V-shaped opening snugly against the work, you can scribe a line along the inner edge of the blade which will pass through the center of the work. By rotating the work about one-quarter turn and scribing another line in the same manner, you form an intersection which marks the exact center of the cylindrical piece.

20-13. *Machinist's square.* The machinist's square is used to lay out or check perpendiculars when more accuracy is required than the combination square head and blade provide. In the manufacture of this tool, the edges of both the body and the blade are carefully machined and then lapped to extreme trueness. Figure 118 illustrates how you can use the machinist's try square, as it is sometimes called, to test adjacent faces of a piece of work for squareness. You should realize that such a precision type tool will remain accurate only if you take special care of it. Do not allow the square to come into contact with other tools that might damage its lapped edges or distort its 90° setting. Clean and polish it frequently to keep it from corroding. Store it in a safe place when you are not using it.

20-14. *Surface gage.* Figure 111 shows one of the uses of a surface gage—the scribing of horizontal layout lines. You can also use it for locating centers in rough work or as a height gage for leveling work on a machinist vise or plate. Since both the vertical spindle and the scriber can be adjusted to and locked in any angular position, you can scribe layout lines on the work at any given height and from almost any position. Set the scriber point of the surface gage to the desired height on the blade of the combination set, using the square head to hold it in the vertical position, as shown in figure 115. Use the small thumbscrew located near the rear of the surface gage base to obtain fine movement of the point for accurate setting. To obtain the best performance at all times from the surface gage, you should oil all moving parts so that they operate freely. When you are not using it, store the surface gage carefully in a safe place to avoid damaging it.

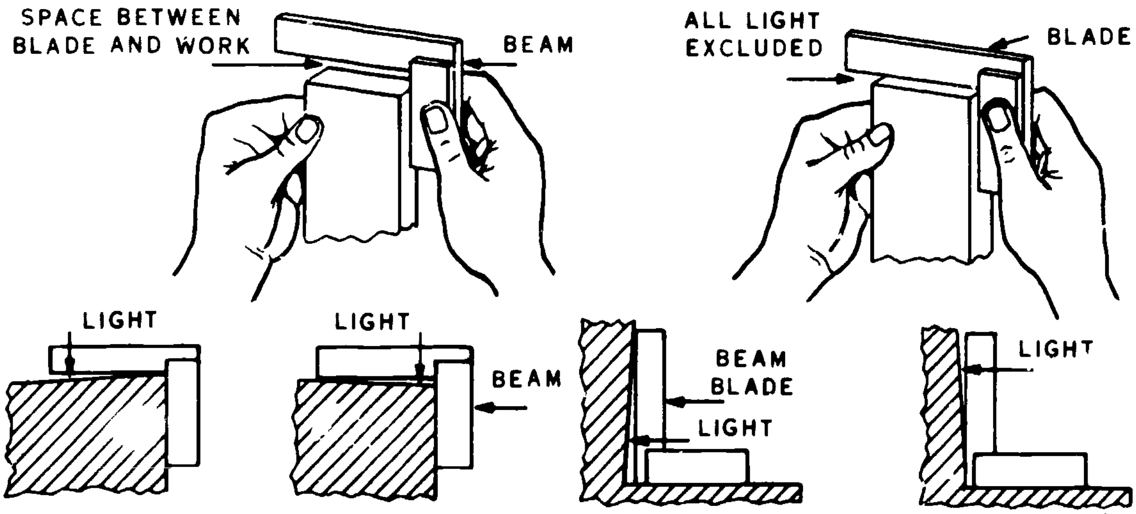


Figure 118. Machinist square.

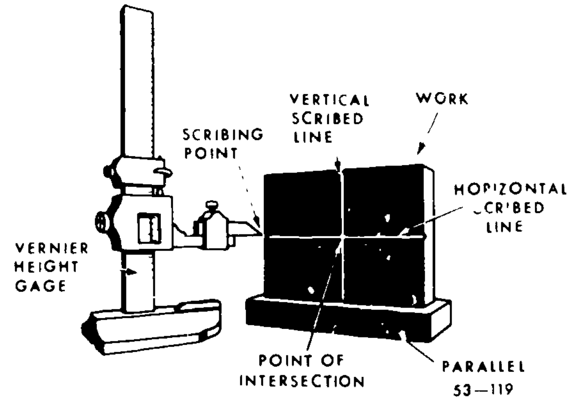


Figure 119. Vernier height gage.

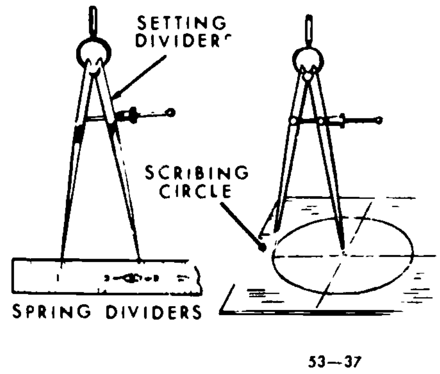


Figure 121. Using dividers.

20-15. *Vernier height gage.* When a high degree of accuracy is desired, you may use the vernier height gage, shown in figure 119, in place of the surface gage. Read it in the same manner as the vernier caliper; it allows you to set the scriber point accurately to 0.001 inch.

20-16. *Ball peen hammer.* You may use a lightweight (8 oz.) ball peen hammer for layout work: Remember, never hammer work on the surface plate.

20-17. *Layout punches.* Although your toolbox usually contains a variety of punches, only the prick punch and the center punch are considered as layout tools.

20-18. The prick punch, shown in figure 120, is ground to an included angle of 30°. The long, slender point allows it to be accurately positioned on the layout lines. Use it to make fine punchmarks along scribed lines so that their location can be found even though the line may be worn off. It is also used to locate centers of holes and to provide a pivot point for the leg of the divider in laying out holes or radii.

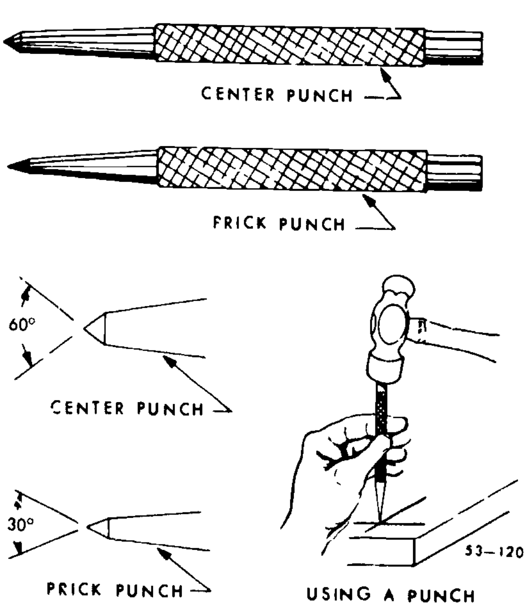


Figure 120. Center punch and prick punch.

20-19. The center punch, shown in figure 120, differs from the prick punch in that its included angle is 60°. Use it mainly to deepen prick punchmarks prior to drilling. You will have to grind the points of these punches frequently to keep them sharp. Check the driving end of the punch at the same time to see that it is flat and perpendicular to the centerline of the punch. Remove any dangerous burrs that form around the edges.

20-20. *Dividers.* In layout work you use spring-jointed dividers, as shown in figure 121, to transfer dimensions and scribe arcs and circles on the work surfaces. You should inspect the points of the dividers to make sure that they are sharp enough to make a clean, precise scratch on a metal surface. If the points are dull, sharpen them on an oilstone by rotating the tool between your forefinger and thumb while you are rubbing the points back and forth against the stone.

20-21. *Trammel points.* Use trammel points, shown in figure 122, when you need a larger than normal pair of dividers. You may position the points along any suitable metal or wooden bar or

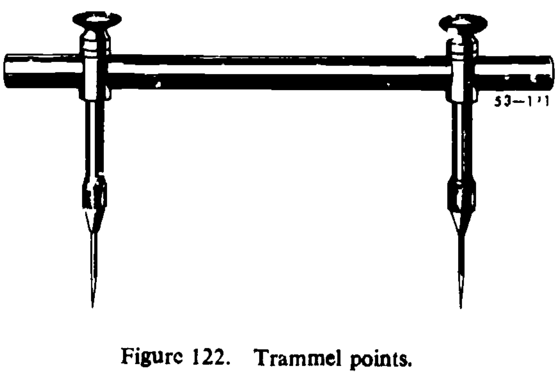


Figure 122. Trammel points.

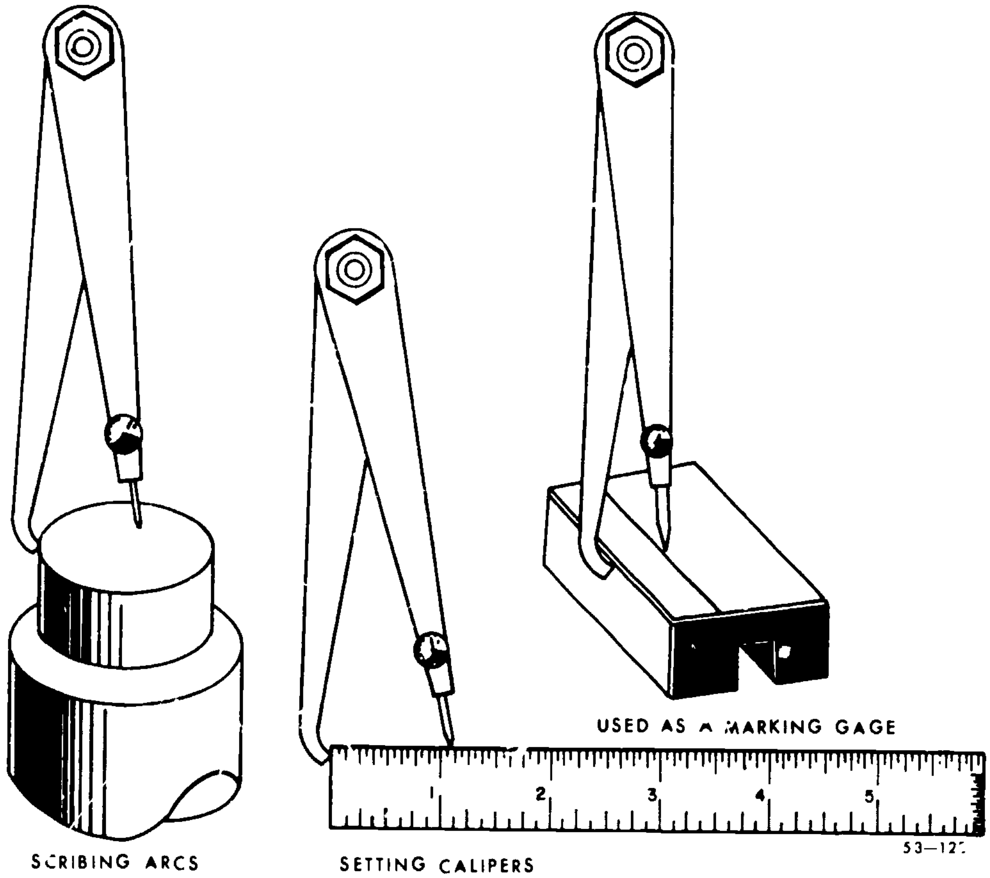


Figure 123. Using hermaphrodite calipers.

rod and use this tool as an ordinary pair of dividers.

20-22. *Hermaphrodite calipers.* Although most calipers are considered measuring instruments, the hermaphrodite caliper is closer to a layout tool. It draws its name from the fact that it has one straight sharp-pointed divider leg and one curved (or bent) dull-edged caliper leg. You can adjust these legs for either outside or inside measurements, as you can see in figure 123, and then quickly scribe lines parallel to edges. You can also use this caliper to locate the center of a boss or other round projection on the piece of work by setting the caliper to a rough estimate of half the diameter of the boss and scribing four arcs spaced approximately 90° apart. A small square is formed around the true center of the boss. This method is not recommended when great accuracy is required.

Bibliography

Books

ARMSTRONG, W. H. *Machine Tools for Metal Cutting.* New York: McGraw-Hill Book Co., Inc., 1957.

BASTION, E.L.H. *Metalworking Lubricants.* New York: McGraw-Hill Book Co., Inc., 1951.

DOYLE, Lawrence E. *Metal Machining.* New York: Princeton-Hall, Inc., 1953.

GRAHAM, Frank D. *Audels Answers on Blueprint Reading for Mechanics and Builders.* New York: Theodore Audel and Co., 1962.

HARRISON, Oval S. *Creative Mechanical Drawing: Basic.* Chicago: American Technical Society, 1954.

HOELSCHER, Randolph P.; Springer, Clifford H.; and Dobovolny, Jerry S. *Basic Drawing for Engineering Technology.* New York: John Wiley and Sons, Inc., 1964.

LE GRAND, Rupert (Editor). *The New American Machinist's Handbook.* New York: McGraw-Hill Book Co., Inc., 1955.

OPERG, Erik, and Jones, Franklin D. *Machinery's Handbook, Eighteenth Edition.* New York: The Industrial Press, 1968.

Engineering Publications

SAE, *Aerospace Material Specifications Index,* New York: Society of Automotive Engineers, Inc., July, 1963.

Department of the Air Force Publications

AFM 39–1, *Airman Classification Manual.*

AFM 127–101, *Accident Prevention Handbook.*

TO 00–5–1, *AF Technical Ord r System,* 15 January 1968.

TO 00–5–2, *Distribution and Storage of AF Technical Order System Publications,* 15 March 1968.

TO 1–1A–8, *Engineering Manual Series—Structural Hardware,* 1 January 1959; change 10 March 1970.

TO 1–1A–9, *Engineering Series for Aircraft Repair—Aerospace Metals—General Data and Usage Factors,* 22 November 1967; change 26 July 1968.

TO 32–1–101, *Maintenance and Care of Hand Tools,* 19 April 1945.

TO 32–1–151, *Hand, Measuring, and Power Tools,* 26 May 1941.

TO 42D–1–3, *Method of Identifying Steel, Aluminum, and Copper Alloys in AF Stock,* 1 May 1958.

Handbooks

MIL–Hdbk–1B, *Cross Index of Chemically Equivalent Specifications and Indentification Code,* 15 May 1958.

Military Standards

Mil–Std–1B, *General Drawing Practices for Engineering Drawings,* 3 May 1958.

Mil–Std–2C, *Engineering Drawings Sizes and Format,* 1 October 1963.

Mil–Std–7, *Types and Definitions of Engineering Drawings,* 18 February 1959.

Mil–Std–8C, *Dimensioning and Tolerancing*, 16 October 1963.

Mil–Std–9A, *Screw Thread Conventions and Methods of Specifying*, 26 May 1960.

Resident Course

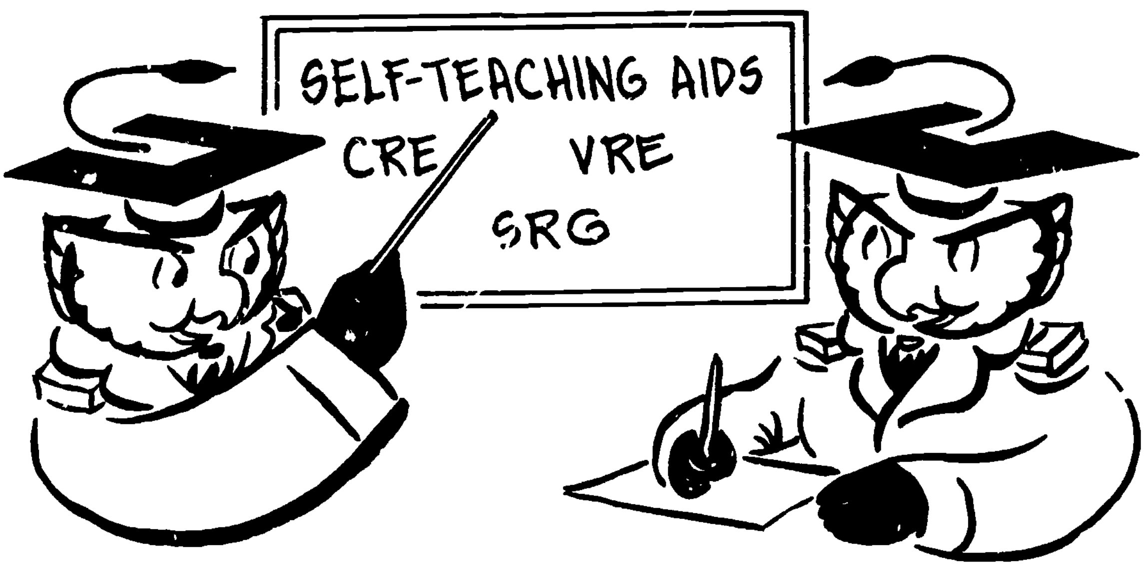
Training literature from Block I, ABR 53130, *Machinist.*

Training literature from Block VII, ABR 53130, *Machinist.*

NOTE: None of the items listed in the bibliography above are available through ECI If you cannot borrow them al sources, such as your base library or local library, you may request one item at a a loan basis from the AU Library, Maxwell AFB. Alabama, ATTN· ECI Bibliograp Assistant. However, the AU Library generally lends only *books* and a limited number of *AFM's.* TO's, classified publications, and other types of publications are *not* available. (For complete procedures and restrictions on borrowing materials from the AU Library, see the latest edition of the *ECI Catalog.*)

53130 01 21
WORKBOOK

Machine Shop Fundamentals



This workbook places the materials you need *where* you need them while you are studying. In it, you will find the Study Reference Guide, the Chapter Review Exercises and their answers, and the Volume Review Exercise. You can easily compare textual references with chapter exercise items without flipping pages back and forth in your text. You will not misplace any one of these essential study materials. You will have a single reference pamphlet in the proper sequence for learning.

These devices in your workbook are autoinstructional aids. They take the place of the teacher who would be directing your progress if you were in a classroom. The workbook puts these self-teachers into one booklet. If you will follow the study plan given in "Your Key to Career Development," which is in your course packet, you will be leading yourself by easily learned steps to mastery of your text.

If you have any questions which you cannot answer by referring to "Your Key to Career Development" or your course material, use ECI Form 17, "Student Request for Assistance," identify yourself and your inquiry fully and send it to ECI.

Keep the rest of this workbook in your files. Do not return any other part of it to ECI.

EXTENSION COURSE INSTITUTE

Air University

TABLE OF CONTENTS

Study Reference Guide

Chapter Review Exercises

Answers For Chapter Review Exercises

Volume Review Exercise

ECI Form No 17

STUDY REFERENCE GUIDE

1. *Use this Guide as a Study Aid.* It emphasizes all important study areas of this volume.

2. *Use the Guide as you complete the Volume Review Exercise and for Review after Feedback on the Results.* After each item number on your VRE is a three digit number in parenthesis. That number corresponds to the Guide Number in this Study Reference Guide which shows you where the answer to that VRE item can be found in the text. When answering the items in your VRE, refer to the areas in the text indicated by these Guide Numbers. The VRE results will be sent to you on a postcard which will list the *actual VRE items you missed.* Go to your VRE booklet and locate the Guide Number for each item missed. List these Guide Numbers. Then go back to your textbook and carefully review the areas covered by these Guide Numbers. Review the entire VRE again before you take the closed-book Course Examination.

3. *Use the Guide for Follow-up after you complete the Course Examination.* The CE results will be sent to you on a postcard, which will indicate "Satisfactory" or "Unsatisfactory" completion. The card will list *Guide Numbers* relating to the questions missed. Locate these numbers in the Guide and draw a line under the Guide Number, topic, and reference. Review these areas to insure your mastery of the course.

Guide Numbers 100 through 120

Guide Number	
100	General Machine Shop Information: The Metalworking Career Field, pages 1-4
101	On-the-Job Training, pages 4-6
102	Technical Publications, pages 6-8
103	Supply Discipline, pages 8-11
104	Communications Security (TRANSEC), pages 11-12
105	Shop and Flight Line Safety, pages 12-18
106	Metals and Heat Treatment: Classification of Metals, Identification and Selection of Metals, pages 19-24
107	Heat Treatment, pages 24-27
108	Inspection of Metals, pages 27-29
109	Preparatory Shop Work: Review of Shop Mathematics, pages 30-35
110	Blueprint Reading and Mechanical Drawings, pages 35-44
111	Geometric Construction, pages 44-47
112	Handtools: Care of Tools—Common Handtools, pages 47-52
113	Handtools: Metal-Cutting Handtools, pages 52-60
114	Measuring Tools, pages 60-66
115	Introduction: Machine and Bench Work, Machinability of Metals and Cutting Lubricants, pages 67-68
116	Power Hacksaw, pages 68-72
117	Drill Press Work: Types of Drill Presses—Feed and Speed, pages 72-77
118	Drill Press Work: Drill Press Operations, pages [illegible]
119	Drill Grinding, pages 81-84
120	Layout and Bench Work, pages 84-90

CHAPTER REVIEW EXERCISES

The following exercises are study aids. Write your answers in pencil in the space provided after each exercise. Immediately after completing each set of exercises, check your responses against the answers for that set. Do not submit your answers to ECI for grading.

CHAPTER 1

Objectives: To show ability to relate the duties of the apprentice machinist to the metalworking career field. To explain on-the-job training procedures in the machinist career ladder. To demonstrate knowledge of Air Force technical publications and supply discipline. To identify the various types of classified information and to determine the appropriate methods of transmission. To demonstrate an awareness of shop and flight-line safety hazards.

1. What is an Air Force career field? (1-2)

2. How many ladders are in the metalworking career field? (1-3)

3. Explain the meaning of the numbers in AFSC 53130. (1-4)

4. What are the main differences between the duties and responsibilities of the machinist and machine shop technician? (1-10, 11)

5. Compare the duties of the machinist and the machine shop technician with regard to work and tool setups on a lathe. (1-10, 11)

6. What is the dual channel concept of on-the-job training? (2-2)

7. What do the letters "CDC" and "OJT" mean? (2-1, 2)

8. What documents are kept with the airman's Consolidated Training Record, AF Form 623? (2-7)

9. What are techn[illegible] [illegible]ders? (2-3)

10 Which type of Air Force publication would you consult for

a A directive with regard to change in Air Force reenlistment policy?

b An explanation of management principles?

c Instructions on repairing J-57 fuel tanks?

(3-2*a*, *b*, 3)

11 Of what value is the *Index of Indexes* TO 0-1-01, to a machinist? (3-4, 5, 6)

12 If a technical order number begins with 00-20, 00-25, or 00-35, what is the general subject matter of the TO? (3-6)

13 Who has pecuniary liability in the Air Force? (4-2)

14 *Situation* The chief of the fabrication section has signed the tool and equipment account for the machine shop What is the shop foreman's obligation with respect to property accountability and responsibility? (4-2)

15 List the two functions in the Air Force Supply System (4-3)

16 Identify each of the three parts of the Federal stock number 5310214287 (4-5)

17 Why would you consult a supply classification index before researching a stocklist? (4-9)

18 What is the classification of defense information the unauthorized disclosure of which could result in

a A situation prejudicial to the defense of the nation?

b Serious damage to the nation?

c. Exceptionally grave damage to the nation?

(5-2)

19 Which of the types of classified materials can be transmitted by registered mail? (5-7)

20 What type of fire extinguisher should be used for an electrical fire in a confined area? (6-11, 19)

21 Why is it safer to use a sharp chisel than a dull one? (6-20,c)

22 List four sources of danger when you work near an operating jet engine (6-22, 23, 25)

23 Why does the Air Force use the AFTO Form 9 radiation sign series rather than a single form? (6-29*a-g*)

CHAPTER 2

Objectives To demonstrate the ability to classify metals To identify metals and to exhibit knowledge of metal selection To explain the effects on metals of various heat-treating processes To exhibit a knowledge of hardness testing, corrosion control, and nondestructive inspection methods

1 What are the two main classifications of metals and how do they differ? (7-1)

2 What is the difference between iron and steel? (7-2)

3 What are the main qualities that are given steel when alloys are added? (7-3)

4 What alloy is added to steel to increase corrosion resistance (7-4*e*)

5 What is meant by the term "stainless steel"? (7-4*f*)

6 Why are aluminum and magnesium used so extensively in aircraft construction (7-5*a*, *b*)

7 What extinguishing agent is used for magnesium fires? (7-5*b*)

8 How are metals identified in the Air Force? (8-1)

9 What do the letters "SAE" mean in referring to metal identification?

10 What do the first two numbers of a four-digit SAE number represent? (8-6)

11 Why is an AMS number more helpful to a machinist than a SAE number? (8-8)

12 What is the Aluminum Association number for aluminum that is 99 60 percent pure? (8-13)

13 What is meant by temper in aluminum? (8-14)

14 Why is the DOD master code for both ferrous and nonferrous metals helpful to the machinist? (8-15)

15 How is the color code on each end of a piece of stock read? (8-16)

16 What technical order describes the Air Force color code system? (8-16)

17 What are the mechanical properties of a metal? (8-18)

18 How are heat-treating furnaces classified? (9-2)

19 What type of furnace is most used in the AF, and why? (9-3

20 What are the five basic heat-treating processes? (9-5)

21 What steps are used in each heat-treating process? (9-6)

22 What two heat treating processes can be used on both ferrous and nonferrous materials? (9-6)

23 Why should tool and chatter marks be removed from a part that is to be heat treated? (9-9)

24 What two publications contain information on heat-treating temperatures and controls? (9-10 11)

25 Which hardness tester is used by the Air Force, and for what reason(s)? (10-3)

26 What is oxidation? (10-6)

27 What factors can increase the rate of oxidation? (10-6)

28. Where does corrosion generally start on a metal? (10-7)

29 What are some of the methods of combating corrosion? (10-8, 9)

30 What are the five methods presently used to perform nondestructive inspection in the Air Force? (10-13)

CHAPTER 3

Objectives To demonstrate a working knowledge of shop mathematics To read blueprints and to explain the types and uses of mechanical drawings To explain the application of geometric construction to mechanical drawing and layout work To explain the proper use and care of hand tools and measuring tools.

1 What are some of the tasks that require the use of mathematics? (11-1)

2 What do the symbols ⊥, ∟, and π represent? (11-3)

3 What is the difference between a proper and improper fraction? (11-6e, f)

4 Reduce the following fractions to their lowest terms $\frac{48}{64}$, $\frac{6}{16}$, $1\frac{28}{64}$, $\frac{50}{32}$ (11-7, 8)

5 Add the following fractions $1\frac{1}{8} + \frac{3}{64}$, $\frac{29}{32} + \frac{33}{64}$, $\frac{7}{8} + \frac{1}{2}$, $5\frac{1}{4} + \frac{7}{64}$ (11-9-12)

6 Subtract the following fractions $1\frac{1}{8} - \frac{3}{64}$, $\frac{29}{32} - \frac{33}{64}$, $\frac{7}{8} - \frac{1}{2}$, $5\frac{1}{4} - \frac{7}{64}$ (11-14)

7 Multiply the following fractions $\frac{3}{8} \times \frac{1}{2}$, $\frac{3}{16} \times \frac{3}{4}$, $1\frac{1}{2} \times \frac{1}{4}$, $\frac{22}{7} \times 1\frac{1}{4}$ (11-15-17)

8 Divide the following numbers $\frac{3}{16} \div 4$, $\frac{1}{2} \div \frac{3}{8}$, $1\frac{1}{2} \div \frac{3}{4}$, $5\frac{1}{4} \div \frac{1}{8}$ (11-19)

9 How can a decimal be defined as a fraction? (11-22)

10 How would you write the following dimensions in decimal form? Thirty-seven thousandths, six tenths, and fifty-one hundredths (11-23)

11 In adding decimal measurements, what is the most important step? (11-25)

12 The overall dimension of a stepped shaft on a blueprint is blurred, while the dimensions for the lengths of each step are 2.25, 0.380, 0.185 and 3.437. What is the overall length? (11-25, 26)

13 The overall dimension of a crankshaft is shown on a drawing as 9.980, while the length of the bearing surfaces and crank total 7.357. The remainder of the shaft is threaded. What is the length of the threaded protion of the shaft? (11-28)

14 If 2.68 is multiplied by 8.62, how many places to the right of the decimal point will there be? (11-29)

15 If the decimal point in the divisor is moved to the right five places, how many places should it be moved in the dividend? (11-30)

16 A drawing arrives in the shop, with no overall length dimension given The seven segments have the following dimensions 0 312, $\frac{1}{8}$, 1 25, $4\frac{7}{16}$, 1 755, $4\frac{7}{16}$, 2 100 What is the overall length? (11-26, 32)

17 What is the purpose of a blueprint? (12-1)

18 In what ways are mechanical drawings different from artist's drawings? (12-4)

19 What is the preferred way of drawing objects for machine shop purposes, and why? (12-5)

20 What determines the number of views necessary in a shop drawing? (12-9, 10)

21 How is the front (or principal) view of an object determined? (12-11*f*)

22 What is the main difference between center and datum lines? (12-12*a*, *k*)

23 What are the three general groupings of dimensions? (1` 14)

24 In what position are numerals and letters placed on drawings? (12-15*h*)

25 What do the abbreviations DIA and R refer to on a shop drawing? (12-16, 17)

26 What is meant by "limits"? (12-20)

27 How is tolerance calculated? (12-21)

28 What is the intentional difference between mating parts? (12-22)

29 What do the abbreviations C'Drill, C'Bore, and C'Sink mean? (12-24c)

30 How many ways can threads be illustrated on a drawing, and what are they? (12-25)

31 What does the abbreviation LH mean, and where is it used? (12-25)

32 In what way is geometric construction useful to a machinist? (13-1)

33. What is the only angle that can be trisected? (13-5)

34 When would geometric construction be useful in dividing a line into equal parts? (13-8)

35 What is the relationship between the sides of a regular hexagon and the radius of the circle that incloses it? (13-13)

36 Why should hand tools be gi · coating of oi before being put away? (14-2)

37 Why is a checklist necessary while you are working around aircraft and other equipment? (14-2)

38 After receiving new standard screwdrivers from supply, what can be done to prevent them from slipping out of the screw slot? (14-5, 6)

39 What are some suggestions concerning the use of hammers? (14-7, 8)

40 If you are going to remove a 9/16 nut from a 3/8 diameter stud with an open-end wrench, what size wrench would you use, and why? (14-11)

41 In a standard set of 10 open-end wrenches, which wrench is the longest? (14-11)

42 In what degrees of offset are open-end wrenches available? (14-12, 13)

43 What are the advantages of box-end wrenches? (14-14)

44. What is the size of an adjustable jaw wrench whose jaw opening is $2\frac{1}{4}$ inches and whose handle length is 20 inches? (14-15)

45 If you suspect a bolt will be difficult to loosen, and you have two sets of sockets, one with 1/4-inch drive and the other with 1/2-inch drive, which size drive should you use, and why? (14-16)

46 What are the types of torque wrenches, and which is in common use in the Air Force? (14-17)

47. Why should an Allen wrench not be used when high torque is required? (14-18)

48 You have to use a chisel on a piece of high carbon steel and a piece of aluminum What angles should be ground on the cutting edge of the chisel? (14-23)

49. A chisel with a mushroomed head has been issued to you What should do about it? (14-24)

50. What length of file is coarser, a 10-inch or 4-inch, if both are of the same cut? (14-25)

51 What safety precautions should you take in using files? (14-26, 27)

52 What are the advantages and disadvantages of a solid frame hand hacksaw? (14-30)

53 When should an all-hard blade be used in a hacksaw? (14-31)

54 What is the minimum number of teeth that should always be in contact with the material at any time? (14-32)

55 What is the best cutting speed for a hand hacksaw? (14-33)

56 How do the three common types of hand taps differ? (14-34)

57 Why should a 1/2-inch diameter hole not be drilled before using a 1/2 inch 13 tap? (14-35)

58 Why should a tap be started as straight as possible? (14-36)

59 Why should the tap be occasionally turned backward as you are tapping a hole? (14-38)

60 What are several precautions that can be used to obtain longer life between reamer sharpenings? (14-40)

61 Why are expansion reamers so useful to a machinist? (14-42)

62 When is a line reamer used? (14-43)

63 In what denominations are scales graduated? (15-2)

64. What is the smallest dimension that can be measured with a micrometer? (15-4)

65 What is the smallest dimension that can be accurately measured with a 4-inch capacity outside micrometer? (15-5)

66 Explain how turning a micrometer thimble one revolution moves the spindle 0 025 inch? (15 6)

67 What is the range of most inside micrometers? (15-8)

68 What is the maximum dimension that can be measured with an inside micrometer? (15-8)

69 Is there any difference between reading 0.000 on an outside micrometer and a depth micrometer? (15-9)

70 What are some precautions you should take to keep your micrometer accurate? (15-10, 11)

71 What are two advantages of vernier calipers over an outside micrometer? (15-12)

72 If you were required to check a shaft for being out of alinement, what tool should you use? (15-13)

73 What do center and screw pitch gages have in common? (15-15, 18)

74 You have found a drill whose size is unknown The shank fits into one size hole in a drill gage, and the fluted end fits into another Which hole is giving the correct size for this drill? (15-16)

75 A very narrow slot is milled in the damaged part you are using as a sample to make a new part What could be used to accurately determine the width of the slot? (15-19)

CHAPTER 4

Objectives To show ability to relate the machinability of metals to cutting lubricants To explain the use of the power hacksaw To demonstrate a knowledge of drill press work and drill grinding To explain layout and bench work

1 What is meant by the machinability of a metal? (16-1)

2 Name three factors that affect the machinability of a metal (16-2)

3 How does a machinist compensate for differences in the machinability of metals? (16-3)

4 How does a cutting lubricant amke it easier to machine metal? (16-4, 5)

5 Why are chlorinated oils generally better cutting lubricants than sulphurized oils? (16-7, 8)

6 Since water is an excellent coolant, why is it not used alone as a cutting lubricant? (16-9)

7 What is the purpose of the baffles in a power hacksaw reservoir? (17-3)

8 Explain the cutting action of the power hacksaw (17-5)

9 What are some of the problems that would cause a power hacksaw clutch to slip? (17-7)

10 What rule of thumb is used to govern the strokes per minute of a power hacksaw? (17-10)

11 What range of blade thicknesses and pitches do power hacksaw blades come in? (17-12)

12 If a power hacksaw blade breaks after cutting only half way through the material, what should you do to prevent breaking the replacement blade? (17-12)

13 Why should extra length be left on material being cut off in a power hacksaw? (17-13)

14 If you had to saw a piece of material that measured 1/2-inch x 3-inches, how would you place it in the vise, and why? (17-14)

15 When you are lubricating machines, what method should be used, and why? (17-17)

16 What are the three general-purpose types of drill presses, and what are their features? (18-2-5)

17 What are the three main parts of the head of a power-driven drill press? (18-9)

18 What are the four ways that most drill press tables can be adjusted? (18-13)

19 What is the shape of the spindle hole of a drill press? (18-15)

20 What tool holder is used when a tool shank taper is too small for the spindle hole? (18-15)

21 How are tapered snank tools removed from drill sleeve or spindle? (18-17)

22 How is work held on a drill press table? (18-19, 20)

23 What are the three types of bolts used to clamp work, and what are some of their advantages? (18-21)

24 Where should a holddown belt be placed in use? (18-21)

25 What are some steps that should be taken in feeding a drill into the work? (18-23)

26 The cutting foot speed range for a material is 100 to 150. What CFS would you use to determine the drill RPM? (18-25)

27 Using the formula RPM $= \frac{(4 \times CFS)}{\text{Drill dia}}$ calculate the RPM for a 3/16-inch drill for material with a CFS of 175-225. (18-26)

28 How should you check the alinement of a center finder over the intersection of layout lines? (18-31)

29 Why are lead drills used? (18-32)

30 Why should a drill be backed out of the hole? (18-32, 33)

31 What are the standard included angles of countersinks? (18-35)

32 While you are using a countersink, chatter occurs. What should you do to stop it? (18-36)

33 Why is the body of a counterbore a few thousandths larger than the standard size of hole to be counterbored and the pilot a few thousandths smaller than drilled hole? (18-38)

34 Why is a hole spot faced? (18-40)

35 When is a hole reamed? (18-41, 42)

36 What type of reamer is used for roughing? For finishing? (18-43, 44)

37 What are some of the advantages of tapping a hole while work is mounted in a drill press? (18-45)

38 Using the formula, tap drill size = O D $-\frac{1}{N}$, what size tap drill would be used for a 1/4 - 28 thread? (18-45)

39 What are the three common drill series, and what are their ranges? (19-3)

40 If you find a drill with a tapered shank on it and the drill size worn off, what do you know about its size? (19-5)

41 What are the three steps in sharpening a drill? (19-8)

42 What is the difference between dressing an abrasive wheel and truing it? (19-9)

43 What is the included angle and clearance angle for a drill used for general drilling operations? (19-11)

44 How should you grind a drill to be used for brass or thin metal? (19-11)

45 What could happen if you drill a hole with a drill whose lips are ground unevenly? (19-13)

46 What factor helps to determine the accuracy of your finished work? (20-1)

47 Why is a thin coat of layout bluing better than a thick coat? (20-3)

48 What are some precautions to observe when you use surface plates? (20-4)

49 What is the proper technique in using a scriber? (20-8)

50 What layout tools would you use to measure or lay out angles other than 45° or 90°? (20-11)

51 What can a surface gage be used for? (20-14)

52 How are prick punches different from center punches? (20-18, 19)

53 What are dividers used for? (20-20)

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 1

1 The grouping of related jobs into common work areas

2 Six

3 53 metalworking career field, XXI machinist ladder, XXX30 apprentice machinist

4 The machine shop technician is able to perform machinist skills with a greater degree of proficiency than the machinist He has more resonsibility for supervision and training

5 The machinist must know how to make routine work and tool setups on a lathe, while a machine shop technician must be able to supervise work and tool setups on a lathe and the design and manufacture of holding devices

6 One phase of the dual channel concept is upgrading the trainee's knowledge through career development courses, the other phase is upgrading the trainee's skills while he works on the job

7 Career development course and on-the-job training

8 The JPG, STS, and STS continuation sheet

9 Official Air Force publications that provide technical information, instructions, and safety procedures for AF equipment

10 *a* Regulation, *b* Manual *c* Technical order

11 Using TO 0-1-01 he can determine the particular general, aircraft specific aircraft, or equipment index which will enable him to locate the technical order he needs

12 Maintenance management system, miscellaneous technical orders, and administrative TOs

13 Persons having command, supervisory, and custodial responsibility

14 The shop foreman is not accountable for the tools and equipment, he shares with the people in his shop the responsibility for custody, care, and safekeeping

15 Equipment management office and base supply

16 53 is the FSC group, 10 is the FSC class, and 2142877 is the identification number

17 To find the four-digit code number for the item and to determine the stocklist for the item

18 *a* CONFIDENTIAL, *b* SECRET, *c* TOP SECRET

19 CONFIDENTIAL and SECRET

20 Carbon dioxide

21 You do not have to strike the head of a sharp chisel with as much force and the hammer is less likely to glance off

22 Temperature and velocity of exhaust gages, suction at the engine intake, area in line with turbine wheel rotation, and noise

23 Each warning sign in the AFTO Form 9 series warns against radiation hazards under specific circumstances

CHAPTER 2

1 Ferrous and nonferrous A ferrous metal contains more iron than any other element the main element in a nonferrous metal is some element other than iron

2 Cast iron contains more than 2 percent carbon, and steel contains less than 2 percent

3 Increase in hardness and toughness

4 Chromium

5 "Stainless steel" is the trade name commonly used in referring to corrosion-resistant steel

6 Because of their light weight

7 Dry sand

8 By numerical and color codes

9 Society of Automotive Engineers

10 The first number indicates the type of steel, and the second indicates the percentage of the main alloying element

11 Because it is a complete procurement specification

12 AA-1060

13 Hardness and physical condition

14 Because it groups materials of similar chemical composition

15 From the end toward the center of the peice of metal

16 TO 42D-1-43

17 The internal reactions of a material to external forces

18 By the type of fuel used, condition of the internal air, or type of atmosphere in the furnace

19 The electrically heated, still-air type, because they are clean efficient, automatic, and require very little maintenance

20 Hardening, case hardening, annealing, normalizing, and tempering

21 Heating, soaking, and cooling

22 Hardening and annealing

23 Because they lead to stress concentrations during heat-treating operations

24 The *Machinery's Handbook* and TO 1-1A-9

25 The Rockwell hardness tester because of its simple operation, ability to test many different metals of different hardnesses, and it does not depend on judgment of operator for accuracy

26 The reaction of a metal to the oxygen in the air

27 An increase in moisture content and air temperature

28 On the surface

29 Scheduled inspections, preventive maintenance, and covering surfaces with paint or corrosion-resistant metal

30 Magnetic particle, eddy current, fluorescent and dye penetrant, ultrasonic, and X-ray

CHAPTER 3

1 Measuring, calculating cutting speeds, machining tapers, and manufacturing gears

2 Perpendicular, right angle (90°), and pi (3 1416), which is the ratio of diameter of circle to its circumference

3 A proper fraction's numerator is less than its denominator, while an improper fraction's numerator is larger than its denominator

4 $\frac{3}{4}$, $\frac{3}{8}$, $1\frac{7}{16}$, $1\frac{9}{16}$

5 $1\frac{11}{64}$, $1\frac{27}{64}$, $1\frac{3}{8}$, $5\frac{23}{64}$

6 $1\frac{5}{64}$, $\frac{25}{64}$, $\frac{3}{8}$, $5\frac{9}{64}$

7 $\frac{3}{16}$, $\frac{9}{64}$, $\frac{3}{8}$, $3\frac{13}{14}$

8 $\frac{3}{64}$, $1\frac{1}{3}$, 2, 42

9 A decimal is a fraction having a denominator of 10, or a power of 10

10 0 037, 0 6, 0 51

11 Be sure that each decimal point is directly below the one above

12 6 252

13 2 623

14 Four

15 Five

16 14 416

17 It indicates the form and size of an object in such a way that the user can understand what the designer wants

18 A mechanical drawing is made with aid of instruments and may appear to distor the object, while an artist's drawing is done freehand and generally shows the object as it is seen by the eye

19 The orthographic projection method because it shows several views of the same object at the same time

20 The nature and shape of the object

21 It is the view that shows the characteristic contour of the object

22. A center line consists of alternate long and short dashes, while a datum line consists of a long and two short dashes

23 Detail or size, position or location, and overall

24 They are positioned so that they can be read from the bottom of the drawings

25 DIA means diameter and R means radius.

26 Limits are the extreme dimensions to which an object may be made and still be acceptable

27 Add the amount a part can be made larger to the amount that it can be made smaller

28 Allowance

29 Counterdrill, counterbore nd countersink.

30 Three As they actually appear, conventional, and simplified.

31 LH means left hand, and it appears in thread designations

32 It enables him to accurately construct drawings and to lay out work prior to machining operations

33 A right angle (90°)

34 When the divisions are not easily measured, such as 6 or 7

35 They are of equal length

36 To help prevent corrosion (rust) from forming

37 To avoid leaving tools in the wrong places, which could then lead to foreign object damage

38 Grind the faces of the blade until the surfaces are nearly parallel

39 Don't hold hammer handle too close to the head, check security of wedge used to secure head to har e, and keep face of head flat

40. A 9/16 wrench because that is distance across the flats of the nut

41 The one with the largest opening

42 15°, 22½°, 75°, and 90°

43 They can be used in close quarters and without slipping off the nut or bolt.

44 20-inch size

45 The 1/2-i h drive, because it is stronger

46 The indicating and the breakaway, with the breakaway being the more common in the Air Force

47 Because damage to the wrench or screw recess could result.

48 Grind an angle of 70° for work on steel and 60° for aluminum

49 Grind the mushroom head off and bevel the edge, to help prevent particles from flying off

50 The 10-inch is coarser

51 Never use a file without a handle or as a pry bar or hammer

52 A solid frame hacksaw is stronger, but is restricted to only one length of blade

53 To saw brass, tool steel, cast iron or any heavy cross-sectional material

54 Two

55 40-50 strokes a minute.

56 A taper tap has a taper of 7 to 9 threads, a plug tap of 2½ to 5 threads, and a bottoming tap of 1 thread

57 Because there would be no material left for the threads.

58 To make sure that the threads are square with the surface of the work and to prevent possible tap breakage.

59 This action prevents long chips which may clog the flutes, causing thread damage, or jam the tap, causing tap breakage

60 Never leave more than 0 015 to be reamed, never turn the reamer in the wrong direction, and always use a cutting lubricant

61 Not only can they ream holes to basic sizes, but they can produce enough variations in sizes to compensate for various fits

62 When the same diameter of reamed holes must be alined

63 Fractions, decimals, and metric

64 Ten-thousandths of an inch

65 3 inches

66 The threads on the spindle screw have a pitch of 40 threads per inch One inch divided by 40 equals 0 025

67 0 500 inch

68 The maximum dimension possible depends on the longest extension rod available.

69 When 0 000 is read on an outside micrometer, no graduations are visible on the barrel, while on a depth micrometer, all graduations are visible.

70. Never overtighten the spindle against the anvil or the work, never swing it by the thimble, don't store it with spindle and anvil in contact, and keep it clean and oiled.

71 The capacity is greater, and it can be used for internal and external measurements.

72. A dial indicator

73 Both can be used to check thread pitches of 7, 10, 12, 14, 16, 20, 24, 28, and 32.

74 The hole in which the fluted end fits.

75 A thickness gage.

CHAPTER 4

1 The ease with which a metal can be cut

2. Chemical composition, grain structure, and heat treatment

3 By adjusting the rate of speed and the rate of feed

4 By reducing friction and carrying away the heat

5 They are lighter bodied and they wet the tool and the work better

6 Water used alone could cause rust to form

7 To cause the chips to settle to the bottom of the tank.

8 The cutting action occurs only on the back stroke The saw blade does not contact the work on the forward stroke

9 Too much down pressure, a dull blade, too large a cut, or use of too coarse a blade

10 The harder the material, the slower the speed

11 Thickness ranges from 0 030 to 0 100 inch, and pitch ranges from 2½ to 18 teeth per inch

12. Turn the work over and start the cut on the opposite side of the material.

13 To allow for the saw not sawing straight down, and to leave enough to finish the ends of the work square.

14 Place it so that the 3-inch-wide side is parallel to the saw blade because this permits more material to be remove per stroke and spreads the teeth wear.

15 Use the progressive method to avoid missing any lubrication points.

16 A sensitive drill press is a very high speed drill press without power feed which is used to drill small holes A plan drill press is used for light and medium duty work, may have power feed, and can be floor or bench mounted A heavy-duty press is floor mounted and has power feed and low speeds for large drills

17 The sleeve, spindle, and feed gears

18 Raising and lowering, swinging to either side of center, revolving around the center, and tilting.

19 It is shaped to fit a standard Morse taper

20 Use a socket reducer (drill sleeve)

21 With a drill drift.

22 In a drill press vise or clamped directly to the table.

23 Cutaway T-bolt can be inserted into T-slot anywhere, tapped block and stud is strongest, and the square head.

24 About a third of the distance from the work to the clamp leveling block.

25. Apply enough pressure to maintain cutting action, do not force or "punch" it through, and reduce pressure as the drill begins to come through the bottom of the work.

26 The lowest, 100

27 3733 RPM.

28 By viewing the work and the center finder from at least two positions about 90° apart

29. To help keep the final drill centered and to reduce the power required to drill the hole.

30 To clear the chips from the hole and drill flutes, to allow the cutting oil to reach the cutting edge of the drill, and to prevent drill breakage.

31 82° and 100°.

32 Reduce the RPM of the countersink

33. To provide clearance for the head of the screw and to prevent the pilot from binding in the drilled hole.

34 To remove irregularities from around hole so that a washer, screw, or bolt has a smooth, square surface to rest on.

35 When a round smoothly finished hole of accurate size is required

36 A rose reamer is used for roughing, while a fluted chucking reamer is used for finishing.

37 You save setup time, insure accurate alinement of the tap with the hole to be tapped, and you can use the spindle to keep pressure on the tap to start it.

38 $0.250 - \frac{1}{28} = 0.213$, the tap drill for $\frac{1}{4} - 28$

39 Number series, 1-80 (0 228 to 0 0135), letter series, A-Z (0 234 to 0 413), and fractional series, 1/64 to 3½ inches

40 It is probably larger than 1/2 inch diameter, since most drills up to and including 1/2 inch have straight shanks.

41 Preparing the abrasive wheel, determining the angles to be ground, and grinding and checking the angles.

42 Dressing reconditions the abrasive surface of the wheel, while truing restores the geometric shape in relation to the wheel axis.

43 Included angle of 118° and clearance angle of 12°-15°

44 Grind the face of the cutting edge parallel to the drill axis

45 Oversize holes will be produced, or the drill may break

46 The care with which you lay out your work

47 A thick coat will flae and produce ragged, inaccurate lines

48 Do not drop items on the plate, place only required items on it, do not perform hammering operations, keep the work free of burrs, and keep light oil film on it when it is not in use

49 Scribe only one thin line just heavy enough to penetrate the bluing but not enough to scratch the work surface

50 Two parts of a combination set the protractor head and 12-inch scale (blade).

51 Scribing horizontal layout lines, locating centers on rough work, and as a height gage for leveling work in a vise

52 The point on a prick punch has an included angle of 30° while on a center punch it is 60°

53 To transfer dimensions and to scribe arcs and circles on the work surfaces

STOP-

1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER.

2. USE NUMBER 2 PENCIL ONLY.

53130 01 21

EXTENSION COURSE INSTITUTE
VOLUME REVIEW EXERCISE

Carefully read the following:

DO'S:

1. Check the "course," "volume," and "form" numbers from the answer sheet address tab against the "VRE answer sheet identification number" in the righthand column of the shipping list. If numbers do not match, take action to return the answer sheet and the shipping list to ECI immediately with a note of explanation.
2. Note that item numbers on answer sheet are sequential in each column.
3. Use a medium sharp #2 black lead pencil for marking answer sheet.
4. Write the correct answer in the margin at the left of the item. (When you review for the course examination, you can cover your answers with a strip of paper and then check your review answers against your original choices.) After you are sure of your answers, transfer them to the answer sheet. If you *have* to change an answer on the answer sheet, be sure that the erasure is complete. Use a clean eraser. But try to avoid any erasure on the answer sheet if at all possible.
5. Take action to return entire answer sheet to ECI.
6. Keep Volume Review Exercise booklet for review and reference.
7. If *mandatorily* enrolled student, process questions or comments through your unit trainer or OJT supervisor.
   If *voluntarily* enrolled student, send questions or comments to ECI on ECI Form 17.

DONT'S:

1. Don't use answer sheets other than one furnished specifically for each review exercise.
2. Don't mark on the answer sheet except to fill in marking blocks. Double marks or excessive markings which overflow marking blocks will register as errors.
3. Don't fold, spindle, staple, tape, or mutilate the answer sheet.
4. Don't use ink or any marking other than a #2 black lead pencil.

NOTE: TEXT PAGE REFERENCES ARE USED ON THE VOLUME REVIEW EXERCISE. In parenthesis after each item number on the VRE is the *Text Page Number* where the answer to that item can be located. When answering the items on the VRE, refer to the *Text Pages* indicated by these *Numbers.* The VRE results will be sent to you on a postcard which will list the *actual VRE items you missed.* Go to the VRE booklet and locate the *Text Page Numbers* for the items missed. Go to the text and carefully review the areas covered by these references. Review the entire VRE again before you take the closed-book Course Examination.

Multiple Choice

Note The first three items in this exercise are based on instructions that were included with your course materials The correctness or incorrectness of your answers to these items will be reflected in your total score There are no Study Reference Guide subject-area numbers for these first three items

1. The form number of the VRE must match

   a my course number
   b the number of the Shipping List
   c the form number on the answer sheet
   d my course volume number

2 So that the electronic scanner can properly score my answer sheet, I must mark my answers with a

   a pen with blue ink.
   b. number 1 black lead pencil
   c ball point or liquid-lead pen
   d. pen with black ink.

3 If I tape, staple or mutilate my answer sheet, or if I do not cleanly erase when I make changes on the sheet, or if I write over the numbers and symbols along the top margin of the sheet,

   a I will receive a new answer sheet.
   b my answer sheet will be hand-graded.
   c I will be required to retake the VRE.
   d my answer sheet will be unscored or scored incorrectly

Chapter 1

4 (100) The Air Force career field to which you are assigned is

   a field maintenance
   b machinist
   c metalworking
   d metals processing

5 (100) The third digit in AFSC 53150 identifies a

   a specific job specialty
   b ladder of a career field
   c skill level
   d career field

6 (100) The AFSC in which you work in a machine shop is your

   a duty AFSC
   b primary AFSC
   c control AFSC
   d secondary AFSC

7 (100) The feature that AFSC 53130 and AFSC 53150 have in common is similar

   a specialty descriptions
   b knowledge requirements
   c proficiency requirements
   d supervisory requirements

8 (100) The machining, fitting, and assembling of metal parts are primarily the responsibility of the

   a metals processing specialist
   b machinist
   c machine shop technician
   d machine shop supervisor

9 (101) The two parts of the dual channel concept of OJT are development of career and

a knowledge.
b supervision
c. production.
d. job proficiency.

10. (101) For certain AFSC's mandatory enrollment in a CDC is required as soon as the airmen

a. are notified of a Direct Duty Assignment.
b begin upgrade training for the next skill level
c graduate from a formal ATC course.
d are given a bypass qulaification test.

11 (101) The document which keeps track of your career field training is

a. AF Form 752, General Military Training Record
b. AF Form 623, Consolidated Training Record
c. Machinist Specialty Training Standard 53130/50/70
d. Machinist Job Proficiency Guide.

12 (102) The technical order file maintained in a machine shop is usually

a a complete file of all AFTOs.
b a single index file.
c. limited to the shops' assigned tasks.
d. a training file.

13. (102) The TO number for the *Index of Indexes* is

a. TO 0-1-01.
b TO 0-1-1.
c. TO 00-1-01.
d. TO 00-1-1.

14 (103) The group of numbers in a Federal stock number which identify the group and class are the

a first two digits
b first three digits.
c. first four digits.
d. last four digits.

15. (103) In which of the following indexes are the manufacturers' names, addresses, and Federal code numbers found?

a The individual stocklist volume index.
b Numeric index (cataloging handbook H2-2).
c. Alphabetical index (cataloging handbook H2-3).
d. Classification structure (cataloging handbook H2-1).

16 (103) Which of the following forms is attached to serviceable items?

A DD Form 1577.
b. DD Form 1577-1.
c. DD Form 1574.
d. DD Form 1574-2

17 (104) The type of documents that require registration and receipting for whenever they are removed from a safe are termed

a classified
b SECRET
c. CONFIDENTIAL
d For Official Use Only.

130

18 (104) The safest method of transmitting TOP SECRET information is by

a registered mail
b coded radio message
c coded telegraph message
d messenger

19 (105) When should spilled liquids be removed?

a At the end of the shift
b Immediately after they are spilled
c When they do not evaporate quickly
d When they are spilled in an area of heavy traffic

20 (105) In addition to adequate lighting, one of the more important factors in producing good work is

a keeping your toolbox in order
b a large work area
c keeping dust to a minimum
d good ventilation

21 (105) According to regulations, the minimum safe distance between a parked aircraft and a person smoking is

a 25 feet
b 50 feet
c 100 feet
d 200 feet

22 (105) Before using an electrically powered tool, you should always verify that it has been

a tested
b certified
c grounded
d calibrated

23 (105) The line pressure of an air-driven tool should *not* exceed a maximum of

a 75 PSI
b 150 PSI
c 200 PSI
d 225 PSI

24 (105) The type of fire extinguisher that should *never* be used on an electrical fire is

a CO_2
b CBM
c dry chemical
d water

25 (105) The most dangerous type of radiation you can be exposed to is

a gamma
b thermal
c beta
d alpha

26 (105) A crate in base supply has a yellow three-bladed magenta colored insignia on it The placard indicates that the contents of the crate are

a serviceable
b radioactive
c reparable
d explosive

27 (106) A ferrous metal is any metal whose chemical composition must contain

a more than 50 percent iron
b less than 2 percent carbon
c. more iron than any other element.
d more than 2 percent carbon

28 (106) The best alloy steel for manufacturing ultra high-speed cutting tools is

a nickel steel
b chromium steel
c tungsten steel.
d silicon-manganese steel

29 (106) An alloying element that makes steel corrosion resistant is

a molybdenum
b chromium
c tungsten.
d vanadium

30 (106) Magnesuim and titanium are both desirable for aircraft construction because of their

a corrosion resistance
b ease of machining.
c. light weight
d heat resistance.

31. (106) The numbers or colors painted on a piece of metal are for

a dimensions
b identification
c protection.
d. shape

32 (106) The best known numerical code used to identify steel by chemical composition is the

a AISI (American Iron and Steel Institute) system
b ASTM (American Society for Testing Materials) system.
c MIL (Military Specificiations) system.
c SAE (Society of Automotive Engineers) system

33 (106) The percentage of carbon content in SAE 1030 coded steel is

a 0.10
b 0 30
c 1 00
d. 3.00

34. (106) Essentially the same numerical codes are used by the

a AISI and SAE
b AISI and ASTM.
c. SAE and ASTM
d. AMS and AA.

35 (106) Complete procurement specifications are covered in the numerical codes of the

a Society of Automotive Engineers system.
b Aluminum Association system
c joint Army-Navy system.
d American Iron and Steel Institute system.

36 (106) The AF color code system is used to

a indicate the size of metal
b identify stored metal
c help prevent accidents.
d represent the temper of steel

37 (106) In a metal, hardness is the resistance that it offers to

a being pulled apart by an applied load.
b deformation or penetration
c cutting or abrasive action
d sudden shock without breaking

38 (107) The heat-treating steps that can be performed on *ferrous and nonferrous* metals are

a annealing and tempering
b tempering and case hardening
c. hardening and annealing
d normalizing and annealing

39 (107) Following the hardening operation tempering increases the hardness of

a. nickel steel
b chromium steel
c high speed steel
d. high carbon steel

40 (107) To reduce brittleness in a piece of high carbon steel that has just been hardened, the next operation would be

a. soaking
b. tempering.
c. annealing.
d normalizing.

41 (107) A hole that is close to the edge of a piece of metal that is to be heat-treated is undesirable because

a uneven heating will develop internal stresses.
b. the material between the h[illegible]d the edge would not develop sufficient hardness.
c it will lead to uneven coolin[illegible]nich causes internal stresses.
d too little surface would exposed to the furnace atmosphere.

42 (107) Metal heat-treating is the responsibility of the

a. heat-treating technician
b metals processing technician
c. machinist.
d. machine shop technician.

43. (108) Testing with a Rockwell hardness tester is performed by a

a. machinist.
b. machine shop technician
c nondestructive inspector
d metals processing technician.

44 (108) The type of corrosion which occurs when dissimilar metals are in contact is

a. pitting.
b. intergranular
c. stress.
d. galvanic.

45. (108) In addition to helping resist corrosion, which of the following surface treatments also increases wear resistance?

a Spraying.
b Chrome plating.
c Hot dipping.
d. Painting

46 (108) The person who operates such equipment as an ultrsonic, X-ray, or magnetic particle machine is in what ladder of the metalworking career field?

a Machinist.
b. Sheet metal
c Metals processing.
d. Nondestructive inspection

47 (109) What value does the symbol π represent?

a 3 1216.
c 3.1612.
b 3 1416
d 3.1614.

48 (109) What is the least common denominator of the fractions 3/8, 9/16, 1½ and $2^1/_8$?

a 16
c. 4
b 8
d. 2.

49 (109) If the fraction 33/64 is subracted from 29/32, the result is

a. 32/64.
c. 25/64.
b. 29/64.
d. 21/64.

50 (109) For a repair job you cut 3 pieces of metal $4^5/_{16}$, $3^3/_4$, and $2^5/_8$ inches long from a 20-inch piece of metal. Allowing 1/8 inch for each saw cut, what is the length of the remaining piece of metal?

a $8^{13}/_{16}$ inches.
c. $9^1/_{16}$ inches.
b $8^{15}/_{16}$ inches.
d. $9^3/_{16}$ inches.

51. (109) A work order for a jack handle specifies that 3/4 of the overall length of 11½ inches be knurled. How long will this knurled portion be?

a. 8 inches.
c. $8^5/_8$ inches
b $8^1/_4$ inches
d. $8^7/_8$ inches.

52. (109) Using the formula C = π X diameter, what is the circumference (C) of a circle 14 inches in diameter?

a 40.0 inches
c 42.5 inches.
b. 42 0 inches.
d. 44.0 inches.

53 (109) Disregarding the width of the saw cut, how many 1¾-inch pieces can be sawed from a bar of material 6½ feet long?

a 30
c 44.
b 41
d 82.

54 (109) If 3 21 is divided by 0 963, the result is

a 3 333
c 0 333
b 3 000
d 0.300

55 (109) With a feed of 0 006 and a speed of 125 strokes per minute, how many minutes are required to saw through a metal rod $1^1/_8$ inches in diameter?

a 0 8
c. 2 25
b 1 5
d 4 00.

134

56 (109) On a blueprint of a shaft which arrives in your shop, the overall length dimension is indistinct If the lengths of the segments are 7/16, 3/32, 1 760, $5^1/_4$, 1 760, 3/32, and 7/16 inches, what is the overall length of the shaft?

a 9 812 inches
b 9 832 inches
c 9 850 inches
d 10 832 inches

57 (110) A drawing which presents top and side views in line with the front view is an

a oblique drawing
b isometric drawing
c anisometric drawing
d orthographic drawing

58 (110) Which of the following shapes of objects would be most likely to require a three-view working drawing?

a Square
b Spherical
c Cylindrical
d Rectangular

59 (110) In mechanical drawing, the basic principle which should be followed in drawing an object is to use the

a natural front as the principal view
b top, front, and right side views
c number of views needed to establish the dimensions and shap
d views which include as many invisible lines as possible

60 (110) The three general groupings of dimensions are

a diameter, width, and length
b size, diameter, and detail
c detail, position, and overall
d surface, center, and height

61 (110) Determine the tolerance of a shaft that has a specified diameter of 190 $^{+\,005}_{-\,010}$ inch

a 005 inch
b 010 inch
c 015 inch
d 020 inch

62 (110) Dimensions for a countersunk hole state the

a depth and diameter of the countersink
b included angle and diameter of the countersink
c included angle and depth of the countersink
d depth of the countersink

63 (111) What is the only angle that can be trisected by using geometric construction?

a. Acute
b Obtuse
c Right
d Oblique

135

64 (112) You are using a new screwdriver of the correct size but have trouble keeping it in the screw slot. You could remedy this situation by

a magnetizing the blade
b grinding the blade tip
c applying less torque to the screw
d grinding the sides of the tip almost parallel.

65 (112) If you are using a center punch and hammer to lay out the locations of holes to be drilled, you should use

a an 8-ounce tack hammer.
b. an 8-ounce ball peen hammer.
c. an 8-ounce slag hammer.
d. 16-ounce ball peen hammer.

66. (112) Most accidents involving hammers are caused by a

a wrong size hammer.
b mushroomed face.
c. cracked handle.
d. loose head

67. (112) The heads and openings of wrenches are offset to

a assist in working in close quarters.
b. shorten the wrench
c. help overcome torque.
d. prevent your hand from slipping.

68. (112) Other than its normal use, the 6-point socket wrench can be used to good advantage when

a. the nut is octagonal in shape.
b access to the nut is difficult.
c. greater holding action is needed.
d there is little space in which to swing the wrench.

69 (113) The tendency of a flat chisel to "dig in" can be reduced by grinding

a. a slightly concave cutting edge.
b. a slight convex cutting edge.
c. the corners at a 45° angle.
d. radii on the corners.

70. (113) The type of file you should use to repair damaged threads on a bolt is a

a three-square file.
b square file
c. mill file.
d. knife file.

71 (113) Which type of hacksaw blade does *not* break easily under bending stresses?

a Set off blade.
b. All hard blade.
c. Raker blade
d. Flexible blade.

72. (113) Which of the following hacksaw blades is recommended for sawing electrical conduit?

a A 14-pitch blade.
b. A 18-pitch blade.
c. A 24-pitch blade.
d. A 32-pitch blade.

73. (113) What is the minimum number of hacksaw teeth that should be in contact with the work at any one time?

a. One.
b Two
c. Three.
d Four.

136

74 (113) If a taper tap of the correct size is *not* available, the type of tap you could use to start tapping a hole is a

a bottoming tap.
b. pipe tap.
c. modified tap
d. plug tap.

75 (113) The proper action to take regarding feed pressure after die teeth take hold and begin to cut is to

a release pressure
b. gradually increase pressure
c. gradually decrease pressure
d. maintain constant pressure.

76. (113) For hand reaming, what type of expansion reamer is the most efficient?

a Solid
b Spiral flute
c. Adjustable blade.
d. Straight flute

77. (114) What measurement operations can be made with spring-joint outside inside calipers?

a Transferring measurements from the work to a rule.
b. Determining the depth of worn areas on a shaft.
c. Setting the flutes on an expansion reamer
d. Checking the concentricity of a pulley.

78 (114) A 5-inch micrometer has a range of

a. 0 to 5 inches.
b. 4 to 5 inches.
c 5 to 6 inches.
d. 5 to 10 inches

79 (114) How many revolutions do you turn a micrometer thimble to move the spindle 1 inch?

a. 4.
b 20
c. 40
d. 400.

80. (114) The range of graduations on the barrel of an inside micrometer is usually

a 0.500 inch
b. 1.000 inch.
c. 1 500 inches.
d 5.000 inches.

81. (114) What is the minimum an inside micrometer can measure?

a. 0.500 inch
b. 1 000 inch.
c 1 500 inches
d. 5 000 inches.

82. (114) You are using a 5-inch extension on an inside "mike" with the standard 1½-inch body and have obtained a reading of .222. What is the diameter of the hole?

a. 5.222 inches.
b. 6.222 inches
c. 6.722 inches.
d 6.822 inches

137

83 (114) On some "mikes," the knurled ring located in the frame near the spindle is designed to act as a

a. ratchet
b. spindle hook
c. calibration ring.
d. spindle lock.

84. (114) If you had several pulleys mounted on the same shaft and wanted to check their concentricity, you would use

a. a dial indicator
b. a vernier caliper.
c. an outside micrometer
d. an outside caliper

85 (114) Which of the following drill sizes could not be checked with a drill gage?

a. 3/32"
b. .1935 (No 10).
c. .234 (A)
d. 9/16"

86 (114) Telescope gages are used in combination with

a. a micrometer
b. a scale.
c. an outside caliper.
d. a center gage

87 (115) One of the main factors which compensates for differences in the machinability of metals is the

a. coolant flow
b. type of operation
c. depth of cut
d. cutting speed

88 (115) The primary purpose of cutting lubricants is to

a. improve work finish
b. reduce friction
c. wash chips away from the tool
d. prevent work warpage.

89 (115) If a machining operation requires high cutting pressure, the proper cutting lubricant to use is

a. chlorinated oil
b. lard oil
c. sulphurized oil
d. soluble oil

90 (116) If a power hacksaw size is 8 inches, you know that this dimension refers to the

a. usable length of blade
b. opening of the frame above the blade.
c. largest piece of metal that can be held in the vise and sawed
d. largest piece of metal that can be held in the vise

91 (116) If the feed clutch of a power hacksaw slips the *least* probable cause is

a. too much feed pressure
b. a dull blade
c. too long a cut has been attempted
d. a loose blade

92 (116) Power hacksaw blades fail most often because of improper

a speed.
b. feed.
c. blade installation
d clamping of the work in the vise

93. (116) The main advantage a horizontal bandsaw has over a power hacksaw is that

a the endless blade distributes coolant more evenly.
b less stock is wasted because of the narrower kerf.
c replacement saw bands are cheaper than saw blades
d. it is easier to set up and use.

94 (117) A 15-inch designation for a drill perss means that the

a total height of the base, column, and head is 15 inches square
b. table or machined surface on the base is 15 inches square.
c. diameter of the column is 15 inches.
d largest piece that can be drilled in the center has a 15-inch diameter

95. (117) The drill perss that can operate at the lowest speeds is the

a plain type.
b sensitive type.
c heavy-duty type
d. bench type

96. (117) A taper shanked tool is prevented from slipping in the spindle by the

a wedging action of the taper.
b. tang on the end of the tapered shank.
c. twisting action of the drill.
d. key and keyway between the sleeve and the tool

97. (117) The main purpose of supporting work on parallels in a drill press vise is to

a prevent holes from being drilled in the table
b. aline the work parallel to the spindle.
c. prevent holes from being drilled in the vise
d aline the work parallel to the vise jaws

98. (117) Using the formula $RPM = \frac{4 \times CFS}{\text{Drill Diameter}}$, determine the desired RPM to drill a 3/4-inch hole in a piece of metal with a CFS of 75

a. 22 5 RPM.
b. 40 0 RPM
c. 225 RPM
d 400 RPM

99 (118) In addition to the proper positioning of the work under the drill, the most important factor for accurate location of a drilled hole depends on the

a sharpness of the drill
b speed of the drill.
c. accuracy of the layout
d rigidity of the work set up.

139

100 (118) The minimum diameter of a lead (pilot) drill should be

a the same diameter as the center drill
b slightly larger than the web thickness of the next drill to be used
c one third the diameter of the final hole size
d one half the diameter of the final hole size

101 (118) Spot facing on a drill press should be done by using a

a counterbore
b countersink
c modified drill
d large diameter reamer

102 (118) The reamer that produces the smoothest and most accurate hole is the

a rose reamer
b hand reamer
c chucking reamer
d taper reamer

103 (118) A proper procedure when tapping threads in tough metal is to

a turn the tap backward occasionally to break up chips
b use a smaller tap when binding occurs
c reverse the tap when it begins to bend
d use a 50 percent bigger tap drill size

104 (118) When the tap takes hold and begins to cut and pull itself into the hole, the proper action is to

a maintain constant spindle pressure
b gradually decrease spindle pressure
c gradually increase spindle pressure
d release the spindle pressure

105 (119) What series of drills would contain a drill suitable for drilling a $625 {}^{+003}_{-002}$-inch-diameter hole?

a Number series
b Letter series
c Fractional series
d Tap drill series

106 (119) In comparing the shank of a 7/16 diameter drill and the shank of a 9/16 drill, you find the

a 7/16 drill shank uses a smaller Morse taper
b two shanks use the same size of Morse taper
c 7/16 drill shank is straight
d two drills will fit in the same size spindle hole

107 (119) Repeated sharpening of a drill will cause the

a flutes to become deeper
b body clearance to decrease
c dead center to get smaller
d web to grow thicker

108 (119) The purpose of dressing a grinding wheel is to

a restore the original concentricity
b reduce drill discoloration
c restore the original abrasiveness
d improve heat dissipation

109 (119) If a drill produces oversize holes, the most probable cause is

a. an improper cutting edge angle
b an excessive clearance angle
c an excessive feed pressure.
d unequal length cutting lips

110 (120) Laying out work is the process of

a planning a job on p ,er
b laying out the necessary tools
c laying out parts prior to assembly.
d marking lines on the surface of material

111 (120) In addition to using sharp layout tools, to insure accurate layout work, you should use a

a machinist square
b. thin coat of layout dye
c combination set
d straight edge

112 (120) The best way to mount work so that it is 90° to the surface plate is to use

a parallels
b an angle plate
c a pair ov V-blocks
d angle parallels

113 (120) The most accurate layout tool a machinist can use to check whether adjacent surfaces are 90° to one another is the

a square head
b surface gage
c machinist's square
d angle plate.

114 (120) The tool that should be used for the most accurate location of horizontal layout lines is the

a surface gage
b vernier height gage
c combination gage
d machinist's square

141

STUDENT REQUEST FOR ASSISTANCE

PRIVACY ACT STATEMENT

AUTHORITY: 44 USC 3101. PRINCIPAL PURPOSE(S): To provide student assistance as requested by individual students. ROUTINE USES: This form is shipped with every ECI course package. It is utilized by the student, as needed, to place an inquiry with ECI. DISCLOSURE: Voluntary. The information requested on this form is needed for expeditious handling of the student's need. Failure to provide all information would result in slower action or inability to provide assistance.

SECTION I: CORRECTED OR L[illegible] ROLLMENT DATA: MAIL TO ECI, GUNTER AFS, ALA 36118

1. THIS REQUEST CONCERNS COURS.
2. TODAY'S DATE
3. ENROLLMENT DATE
4. PREVIOUS SERIAL NUMBER
5. SOCIAL SECURITY NUMBER ___ - __ - ____
6. GRADE/RANK
7. INITIALS
LAST NAME
8. OTHER ECI COURSES NOW ENROLLED IN
9. ADDRESS: (OJT ENROLLEES - ADDRESS OF UNIT TRAINING OFFICE/ALL OTHERS - CURRENT MAILING ADDRESS)
ZIP CODE
10. NAME OF BASE OR INSTALLATION IF NOT SHOWN ABOVE:
11. AUTOVON NUMBER
12. TEST CONTROL OFFICE ZIP CODE/SHRED

SECTION II: Old or INCORRECT ENROLLMENT DATA

1. NAME: 2. GRADE/RANK: 3. SSAN:
4. ADDRESS: 5. TEST OFFICE ZIP/SHRED:

SECTION III: REQUEST FOR MATERIALS, RECORDS, OR SERVICE

(Place an "X" through number in box to left of service requested)

ADDITIONAL FORMS 17 available from trainers, OJT and Education Offices, and ECI. The latest course workbooks have a Form 17 printed on the last page.

1	EXTEND COURSE COMPLETION DATE. (Justify in Remarks)
2	SEND VRE ANSWER SHEETS FOR VOL(s): 1 2 3 4 5 6 7 8 9 - ORIGINALS WERE: NOT RECEIVED, LOST, MISUSED
3	SEND COURSE MATERIALS (Specify in remarks) - ORIGINALS WERE: NOT RECEIVED, LOST, DAMAGED.
4	COURSE EXAM NOT YET RECEIVED. FINAL VRE SUBMITTED FOR GRADING ON (Date):
5	RESULTS FOR VRE VOL(s): 1 2 3 4 5 6 7 8 9 NOT YET RECEIVED. ANSWER SHEET(s) SUBMITTED ON (Date):
6	RESULTS FOR CE NOT YET RECEIVED. ANSWER SHEET SUBMITTED TO ECI ON (Date):
7	PREVIOUS INQUIRY (ECI FORM 17, LTR, MSG) SENT TO ECI ON:
8	GIVE INSTRUCTIONAL ASSISTANCE AS REQUESTED ON REVERSE:
9	OTHER (Explain fully in remarks)

REMARKS: (Continue on Reverse)

OJT STUDENTS must have their OJT Administrator certify this request. ALL OTHER STUDENTS may certify their own requests.	I certify that the information on this form is accurate and that this request cannot be answered at this station. (Signature)

ECI FORM 17 JUN 77 PREVIOUS EDITIONS MAY BE USED

SECTION IV: REQUEST FOR INSTRUCTOR ASSISTANCE

NOTE: Questions or comments relating to the accuracy or currency of textual material should be forwarded directly to preparing agency. Name of agency can be found at the bottom of the inside cover of each text. All other inquiries concerning the course should be forwarded to ECI.

VRE ITEM QUESTIONED:

Course No. ______

Volume No. ______

VRE Form No. ______

VRE Item No. ______

Answer You Chose (Letter) ______

Has VRE Answer Sheet been submitted for grading?

☐ YES ☐ NO

REFERENCE
(Textual support for the answer I chose can be found as shown below)

In Volume No: ______

On Page No: ______

In______ (Left)_____(Right) Column

Lines_____Through______

MY QUESTION IS:

Remarks:

53130 02 0971

CDC 53130

Apprentice Machinist

(AFSC 53130)

Volume 2

Lathe Work

Extension Course Institute

Air University

PREPARED BY
DEPARTMENT OF WEAPON SYSTEMS SUPPORT TRAINING
3345TH TECHNICAL SCHOOL (ATC)
CHANUTE AFB, ILLINOIS 61868

EXTENSION COURSE INSTITUTE, GUNTER AIR FORCE BASE, ALABAMA

Preface

VOLUME 2 covers lathe work and is divided into four chapters. Note the chapter titles on the contents page. Chapter 1 covers the introduction to the lathe; Chapter 2, turning operations; Chapter 3, threading operations; and Chapter 4, special lathe operations and attachments. Now, leaf quickly through the pages of each chapter and note the numbered headings; this will help you to understand the scope of the volume.

Code numbers appearing on the figures are for preparing agency identification only.

If you have questions on the accuracy or currency of the subject matter of this text, or recommendations for its improvement, send them to Tech Tng Cen (TTOC) Chanute AFB, IL 61866.

If you have questions on course enrollment or administration, or on any of ECI's instructional aids (Your Key to Career Development, Study Reference Guides, Chapter Review Exercises, Volume Review Exercise, and Course Examination), consult your education officer, training officer, or NCO, as appropriate. If he can't answer your questions, send them to ECI, Gunter AFB. AL 36114, preferably on ECI Form 17, Student Request for Assistance.

This volume is valued at 18 hours (6 points).

Material in this volume is technically accurate, adequate, and current as of May 1971.

Contents

		Page
	Preface	*iii*
Chapter		
1	INTRODUCTION TO THE LATHE	1
2	TURNING OPERATIONS	26
3	THREADING OPERATIONS	49
4	SPECIAL LATHE OPERATIONS AND ATTACHMENTS	65
	Bibliography	73

Introduction to the Lathe

THE LATHE IS the most useful machine in the machine shop. More operations can be done on it than on any other machine tool. A machinist can do straight and taper turning, facing, thread cutting, drilling, boring, and spring winding on a lathe. By using various attachments, he can also do grinding and milling. In this chapter, we will discuss the construction and operation of the lathe, cutting tools, and work holding devices. All of this information will be used when you perform the various lathe operations.

1. Construction and Operation

1-1. We will limit our discussion in this section to basic lathe information, such as the description of a lathe; safety precautions; control levers; and speeds, feeds, and depth of cut.

1-2. **Description.** You should be able to identify the type, size, and major parts of a lathe. You will find the following information helpful.

1-3. *Lathe classification.* Lathes are divided into three classes: toolroom, engine, and turret. They are quite similar in their general construction and operation, although each is designed for a specific purpose or type of operation. The *toolroom lathe* is more accurately constructed than the *engine lathe* and usually has more attachments and accessories, making possible a greater range of precision work. The *turret lathe,* although similar in construction to both the toolroom and engine lathes, is used primarily for production work. Most lathes are mounted on a base or on legs. Small lathes are usually mounted on a bench or table and are called *bench lathes.* The addition of casters to the bench makes it possible to move them easily. Bench lathes come in various types and are classified in the same manner as the larger lathes.

1-4. The size of the lathe is designated by the maximum diameter of the work that can be swung over the ways, the distance between centers, and the overall length of the bed. The length of the bed is usually designated in inches for smaller lathes and in feet for very large ones. Figure 1 illustrates where the size measurements are made.

1-5. *Major assemblies.* A description of the major lathe assemblies will help you understand lathe operation. The major assemblies are the bed, headstock, tailstock, carriage, and the feeding and threading mechanism.

1-6. The *bed,* shown in figure 2, is the heavy framework which rests on metal legs and supports the other mechanical units of the lathe. It must be heavy, well supported, and able to resist the stress set up by heavy roughing cuts. The *ways,* which are the accurately machined and handscraped or ground surfaces on the bed, afford alinement and a bearing surface for the tailstock, carriage, and headstock. Be careful to prevent damage to the ways.

1-7. The *headstock,* shown in figure 2, is located on the left end of the lathe bed. It contains the main spindle, oil reservoir, and the necessary gearing to obtain various spindle speeds and to transmit power to the feeding and threading mechanism. The headstock mechanism is driven by an electric motor through belting and shafting. The main spindle is mounted on bearings in the headstock and has a hole through its entire length. The hole in the nose of the spindle has a taper which varies with the size of the lathe; in most cases it is a Morse taper. Centers, collets, drill chucks, and taper-shank drills may be inserted into the spindle; and chucks or faceplates may be screwed or clamped on the spindle nose.

1-8. The purposes of the *tailstock,* shown in figure 2, are to support one end of the work when it is machined between centers; to support a long piece of work held in a chuck; or to hold various cutting tools, such as drills, reamers, etc. The tailstock is mounted on the ways and is designed to be clamped at any point along them. It has a sliding spindle that is operated by a handwheel and is clamped into position with the spindle clamp. The tailstock is adjusted laterally (toward or away from you) by adjusting screws. The tailstock should be unclamped from the ways prior to making any lateral adjustments. Unclamping allows the

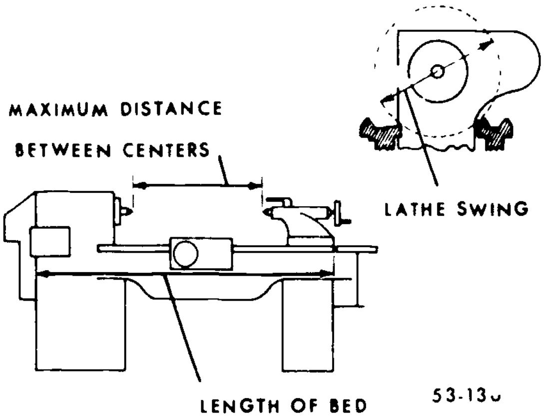


Figure 1. Size designation of a lathe.

upper portion of the tailstock to move freely and prevents damage to the lateral adjustment screws.

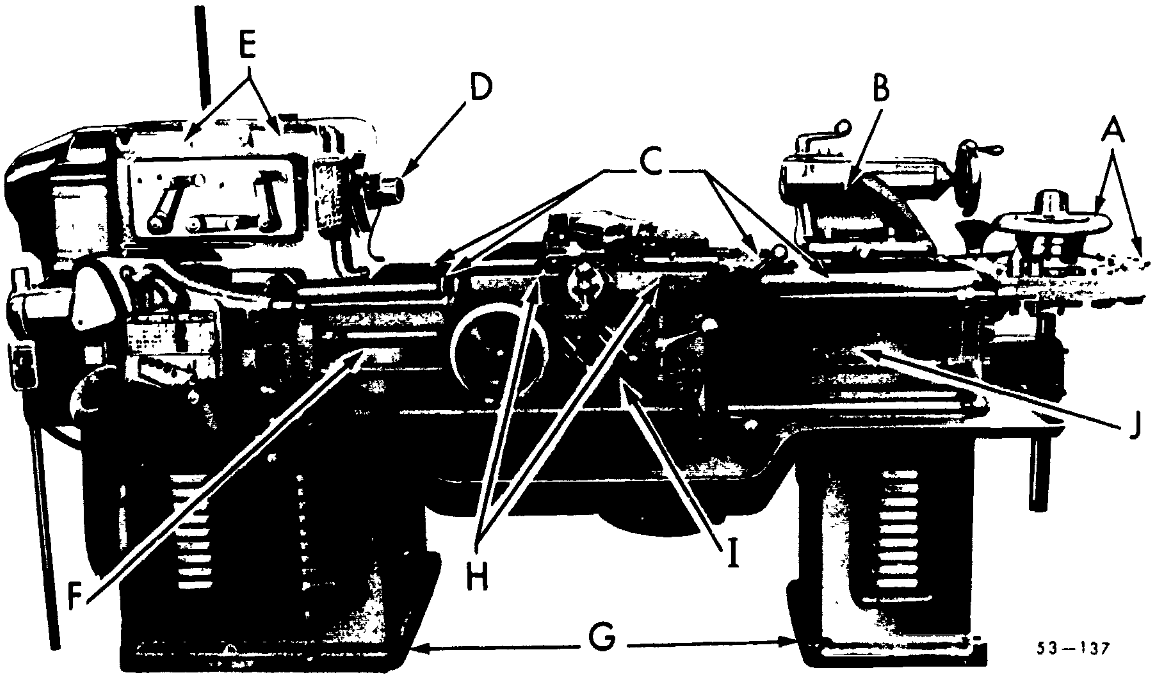
1-9. The function of the *carriage*, shown in figure 2, is to carry the cutting tool. It may be moved by hand or by power, and it can be clamped at any point along the ways by means of a clamping screw and nut. The carriage contains the saddle and the apron. The saddle carries the cross-slide and the compound rest. The cross-slide is mounted on dovetail ways on the top of the saddle; it is moved back and forth 90° to the axis of the lathe by the cross-slide screw, which may be turned by a handwheel or by power. The compound rest is mounted on the cross-slide, and it can be swiveled and clamped at any angle in a horizontal plane. It is used extensively when steep tapers and angles are being cut. The apron contains the gears and feed clutces which are used to transmit motion from the feed rod or lead screw to the carriage and cross-slide.

1-10. The *feeding and threading mechanism,* shown in figure 3, is power-driven from the headstock spindle by means of a gear train. This mechanism transmits power to the carriage for turning or thread-cutting operations. A chart showing the positions of the levers for obtaining various feeds in thousandths of an inch and the various number of threads per inch is located in the quick-change gearbox.

1-11. **Safety Precautions.** You must be constantly aware of the danger involved in operating a lathe. A lathe is not a plaything; carelessness on your part, even for a moment, can be disastrous. Most safety precautions are nothing more than common sense, and accidents usually happen to persons who "didn't think." The following precautions will help you prevent accidents. Use them and your head, and you will avoid injury.

a. Keep your sleeves rolled up to the elbow while you are operating a lathe.

b. Do not allow clothing to hang loose.



A. COLLET ASSEMBLY
B. TAILSTOCK
C. WAYS
D. MAIN SPINDLE
E. HEADSTOCK
F. BED
G. SUPPORTING LEGS
H. CARRIAGE
I. APRON
J. BED

Figure 2. Engine lathe.

c. Remove *all* jewelry and watches (including wedding rings) before you operate the lathe.

d. Do not place your hands or rags on revolving work or moving parts.

e. Do not lean on the lathe.

f. If it is necessary to leave the machine, make certain that it is turned off.

g. Turn the machine off before talking to anyone.

h. Know how to stop the machine quickly if an emergency arises.

i. Keep metal turnings and bar stock off the floor.

j. Keep oil wiped off the floor.

k. Do not handle chips with your hands.

1-12. **Control Levers.** Do not be alarmed by the number of levers on a lathe. Although they are quite numerous, you can separate them into three groups: spindle controls, lead screw and feed rod controls, and power feed controls. Figure 4 will help you identify the numerous control levers on a typical lathe; refer to it frequently as we discuss the various control levers.

1-13. *Spindle controls.* Spindle controls may be divided into two groups: those which cause the spindle to rotate and those which determine the speed of rotation.

1-14. The spindle control levers determine whether or not the spindle rotates. Most lathes have two spindle control levers: a stationary lever located near the headstock and a lever which moves with the carriage. You can use either one of these to engage and disengage the spindle clutch. The spindle clutch must be engaged before the spindle and power feed mechanism will operate. You should avoid slamming the spindle control levers in or out of engagement to prevent damage to the clutch. Exerting additional pressure on the lever after disengagement applies a braking action to the spindle, which is extremely useful in an emergency.

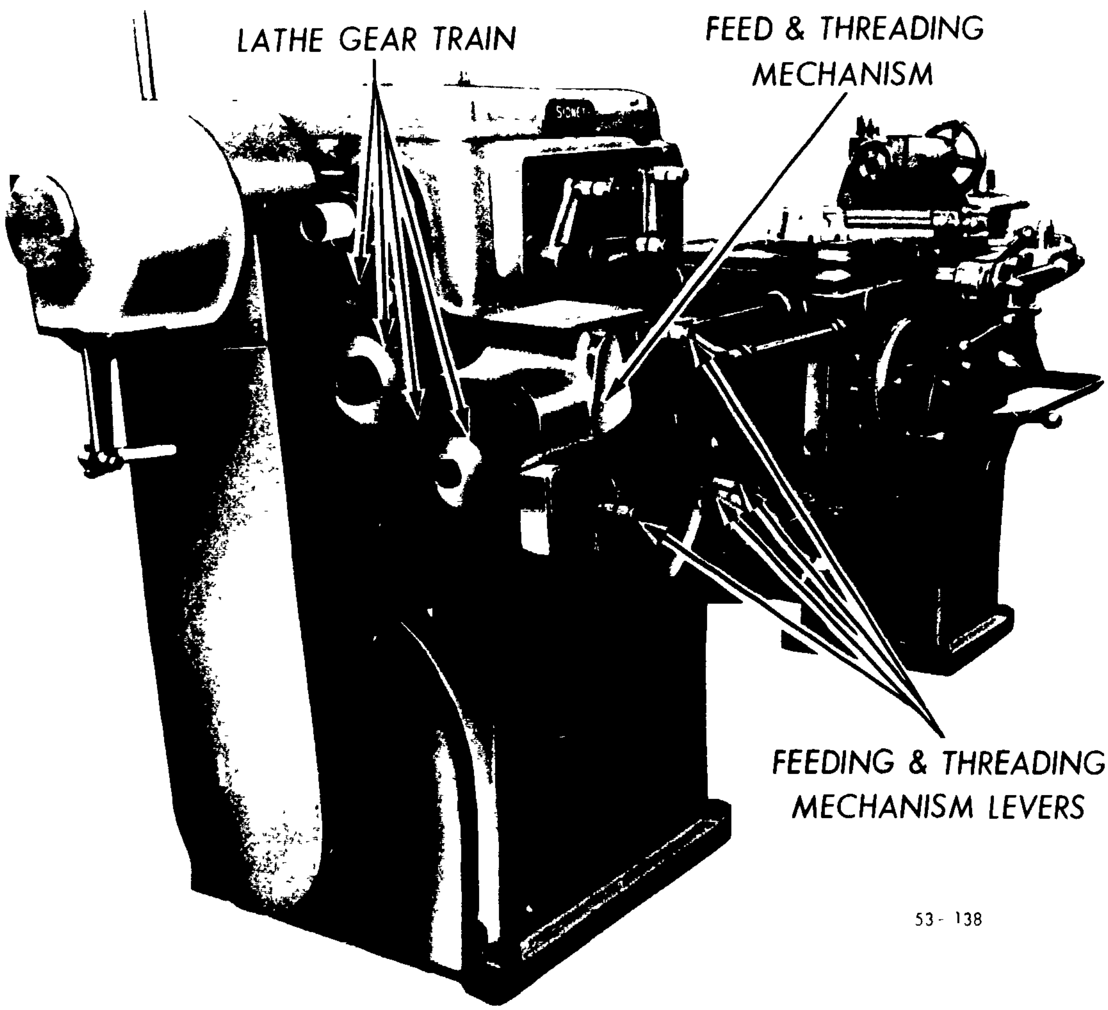
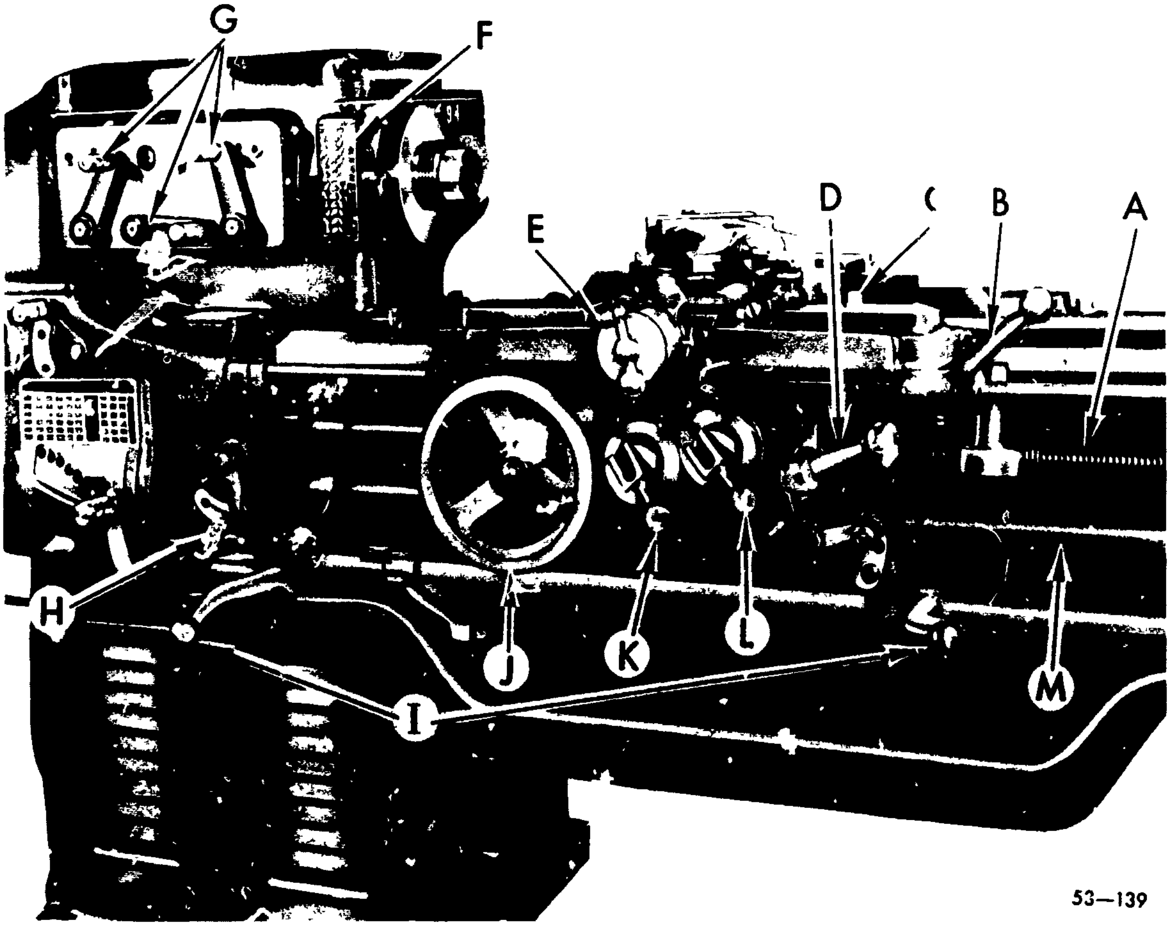


Figure 3. Feeding and threading mechanism.



A. LEAD SCREW
B. FEED REVERSE LEVER
C. CARRIAGE DOCK
D. SPLIT NUT LEVER
E. CROSSFEED SCREW HANDWHEEL
F. SPINDLE SPEED CHART
G. SPINDLE SPEED ADJUSTMENT LEVER
H. FEED OR SCREW SELECTION LEVER
I. SPINDLE CONTROL LEVERS
J. CARRIAGE HANDWHEEL
K. LONGITUDINAL FEED CONTROL LEVER
L. CROSSFEED CONTROL LEVER
M. FEED ROD

Figure 4. Operating levers.

1-15. The spindle speed adjustment levers are located on the headstock. You must position the spindle adjustment levers, as shown on the spindle speed chart, to obtain the desired spindle rpm.

CAUTION: The spindle clutch must be disengaged before you shift the spindle adjusting levers to prevent damaging the lathe.

1-16. *Lead screw and feed rod controls.* The lead screw and feed rod control levers consist of the levers located on the quick-change gearbox, the feed or screw selection lever, and the feed reverse lever.

1-17. The position of the levers located on the gearbox determines the ratio of the lead screw rpm or feed rod rpm to the spindle rpm. This ratio determines the rate of feed or the number of threads per inch. Figure 5 shows the tumbler and selector levers on the gearbox. You can obtain the desired feed or number of threads per inch by positioning these levers as indicated by the chart on the gearbox.

1-18. You will use the feed or lead screw selection lever to engage either the lead screw or the feed rod. Engage the lead screw when you machine threads and the feed rod when you use power feed. Some machines use a sliding gear to engage the lead screw instead of a lever, whereas others have a combination lead screw and feed rod and do not have a selecting device.

1-19. You will use the feed reverse lever to reverse the direction the carriage or cross-slide travels when you use power feed or when you machine threads. The feed reverse lever on bench lathes is often located on the left side of the headstock just above the quick-change gearbox.

1-20. *Power feed controls.* The power feed controls consist of the longitudinal and crossfeed control levers and the split-nut lever. They are located on the carriage apron.

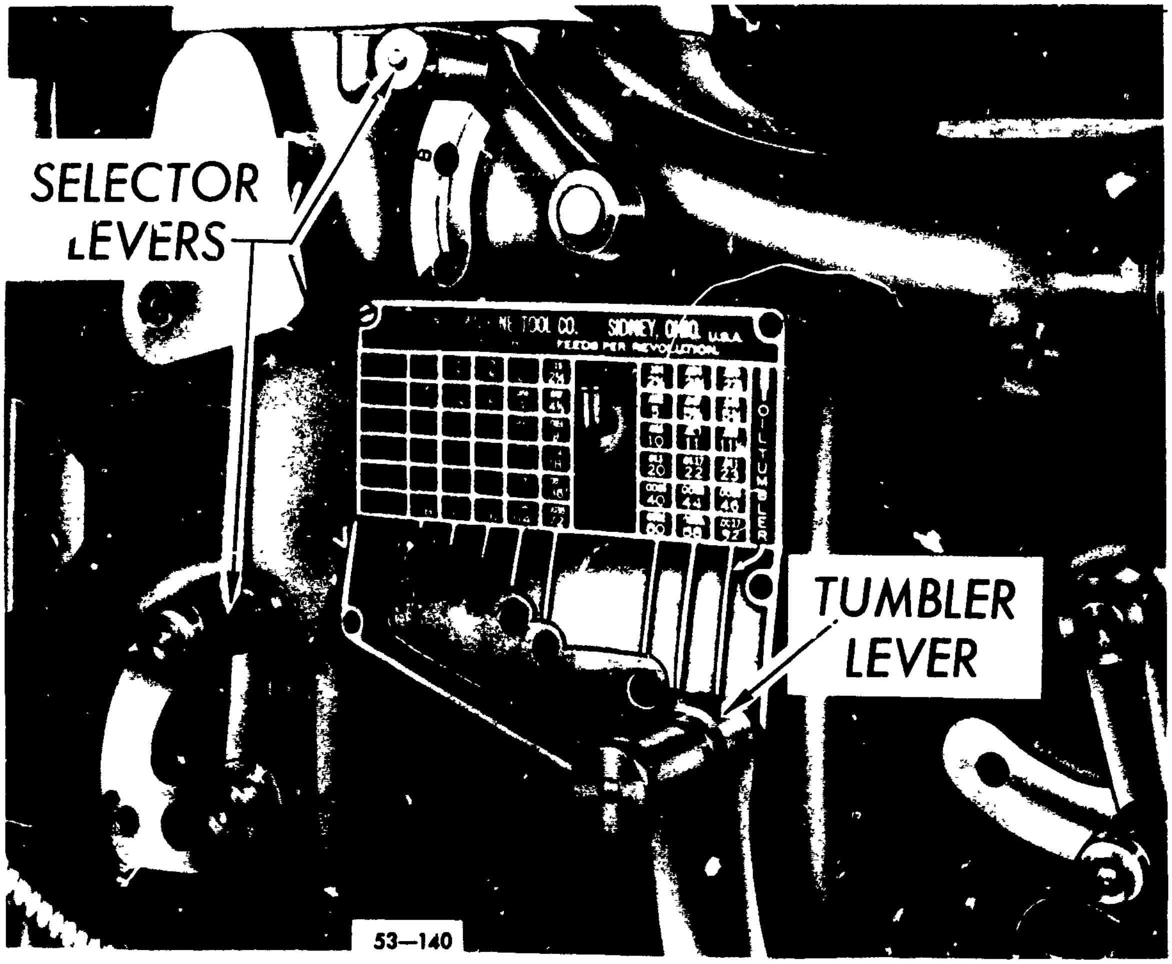


Figure 5. Quick-change gearbox.

1-21. Use the longitudinal and the crossfeed control levers to engage or disengage the power feed of the carriage or the cross-slide.

CAUTION: Do not engage both power feeds at the same time or allow the cross-slide to feed to the end of its travel or you may damage the lathe.

1-22. Use the split-nut lever to engage or disengage the split nut with the lead screw, which causes the carriage to travel when you are cutting threads. Most machines have a built-in device that prevents engaging the split-nut lever and the power feed levers at the same time.

1-23. **Speed, Feed, and Depth of Cut.** Economical and efficient machining requires proper speeds, feeds, and depth of cuts.

1-24. *Speed.* Speed means spindle rpm in lathe work. You can use the formula for selecting drill press speed for lathe work with only a slight modification. Since most lathe work involves the machining of the periphery of a piece of stock, the diameter of the workpiece must be considered. The formula is:

$$\text{Rpm} = \frac{4 \times \text{CFS}}{\text{work diameter}}$$

Work diameter is the diameter actually being machined, and not necessarily the largest diameter of the material. Table 1 gives the recommended CFS

TABLE 1
RECOMMENDED CUTTING FEED SPEED

MATERIAL	CUTTING FOOT SPEED
LOW-CARBON STEEL	80 TO 110
MEDIUM CARBON STEEL	60 TO 80
HIGH-CARBON TOOL STEEL	50 TO 60
STEEL FORGINGS	50 TO 60
STAINLESS STEEL	30 TO 40
SOFT CAST IRON	100 TO 150
HARD CHILLED CAST IRON	70 TO 100
MALLEABLE IRON	80 TO 90
ORDINARY BRASS AND BRONZE	200 TO 300
HIGH-TENSILE BRONZE	70 TO 150
MONEL	40 TO 150
ALUMINUM AND ITS ALLOYS	200 TO 300
MAGNESIUM AND ITS ALLOYS	250 TO 400
BAKELITE	100 TO 150
WOOD	300 TO 400

NOTE: CARBON STEEL DRILLS SHOULD BE RUN AT SPEEDS OF FROM 40 TO 50 PERCENT SLOWER THAN THOSE GIVEN ABOVE

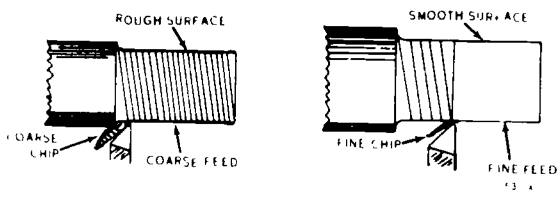


Figure 6. Roughing and finishing feeds

(cutting foot speed) for several common materials. For additional information, consult the *Machinery's Handbook* or some other machinist publication. Use the lower value for rough turning and the higher value for finish turning. However, this formula is only a guide. The speed that you actually select should be based primarily on the formula, but you must also consider the following factors:

a. The material being machined. Generally, hard materials require a slower cutting speed than soft or ductile materials.

b. Tool material. Figure 6 is based on the use of high-speed cutting tools and drills. Decrease the speed when you use carbon steel tools. Increase the speed when you use carbide and Stellite tools.

c. The shape of the tool and the operation being performed. Operations having a great amount of tool and work contact require slower cutting speeds than general turning operations.

d. Feed and depth of cut. Heavy roughing cuts require slower cutting speeds than light finishing cuts.

e. Coolant or cutting lubricant used. Materials which are machined dry require slower speeds than those which are machined with a coolant.

f. Power, design, and condition of the machine. You can use higher speeds on heavy, rigidly designed machines which are in good repair than on light-duty or worn machines.

1-25. *Feed.* Feed is the distance the tool advances per revolution of the spindle. The feed should be based on the following factors:

a. Finish desired. Coarse feeds produce a rough finish; fine feeds, a smooth finish, as shown in figure 6. A feed of 0.025 to 0.035 inch is recommended for rough turning, and a feed of 0.002 to 0.005 inch is recommended for finish turning.

b. Work setup. Work that is securely held in a four-jaw chuck or between centers can withstand heavier feeds than work held in a collet chuck.

c. Work diameter and length. Use light feeds when you machine small diameter work or very long work to prevent the material from springing away from the cutting tool.

d. Tool contact. The greater the amount of tool contact, the greater the pressure exerted on it. This requires a reduction in feed rate.

1-26. *Depth of cut.* Depth of cut is the distance the tool is fed below the surface of the work. Figure 7 shows the effect a 0.125-inch depth of cut has on the work diameter. The work diameter is reduced twice the depth of cut; e.g., a 0.100-inch depth of cut reduces the diameter by 0.200 inch. The factors affecting feed and speed selection should also be considered when you select the depth of cut.

NOTE: You must be careful not to remove so much material that the piece is made too small.

2. Cutting Tools and Holders

2-1. In order to machine materials effectively and efficiently, you must have the correct type of tool with a keen cutting edge of the proper clearance and rake angles. The cutting tool must also be properly held and supported. A cutting tool may be considered as a wedge which provides a means of separating or parting metal. Figure 8 illustrates the cutting action of a tool. In this section we will discuss tool materials, cutting tool shape, cutting tool geometry, grinding cutting tools, and tool holders.

2-2. **Cutting Tool Materials.** A cutting tool must be harder than the material that is to be machined and must retain its hardness and toughness at the high temperatures developed in machining operations. Before the development of high-speed steel and cemented carbide cutting tools, high-carbon steel tools were used extensively.

2-3. *High-carbon steel tools* are generally hand forged, hardened, tempered, and ground. There has been a decline in the use of high-carbon steel tools, and they are very seldom used in modern machine shops. The chief objection to high-carbon cutting tools is that they cannot be operated at high cutting speeds, because the heat which is generated breaks down the cutting edge. Consequently, they require reforging and heat treatment after numerous sharpenings.

2-4. *High-speed cutting tools* are used extensively in lathe work. They are available in many sizes and shapes. The basic alloying elements of

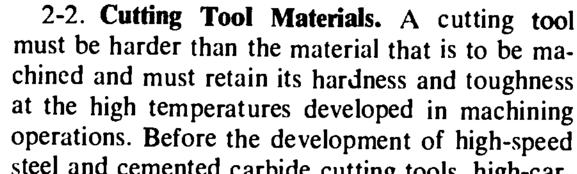


Figure 7 Depth of cut.

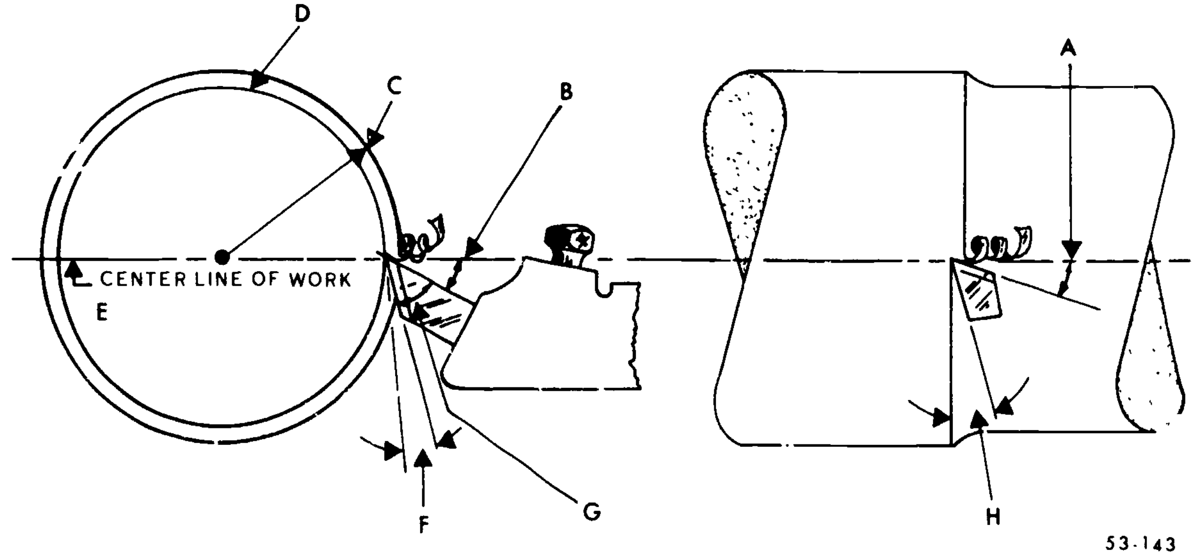


Figure 8. Cutting action of a tool.

high-speed steel are tungsten, chromium, and vanadium. Tungsten gives the steel the property of red hardness; that is, it retains its hardness at red heat and can be used at cutting speeds approximately double that of carbon steel tools. Chromium and vanadium give the steel deep hardening qualities. This means that every part of the tool hardens during heat treatment. The standard high-speed tool blank is a rectangular bar with a square cross-section. Sometimes, the larger tools are forged and held directly in the toolpost, but it is more convenient and economical to use the smaller tools in a toolholder, which allows them to be readily removed for replacement with tools of different shapes.

2-5. *Cemented-carbide tools* are used mostly in production work involving long continuous cuts at high speeds. Usually, just the tip of the tool is made of tungsten-carbide or tantalum-carbide. The tip is silver-soldered or brazed to a steel shank, as shown in figure 9. The high cost of cemented carbide prohibits making the entire tool of this material. Carbide tools may also be used to cut materials that are hard and have an abrasive action, since they do not dull easily. This tool material is extremely hard, ranking next to diamond in hardness and retaining its hardness at high temperatures; however, it is brittle and chips easily if subjected to shock. A special silicon carbide grinding wheel is necessary for sharpening carbide-tipped tools. The ordinary shop grinding wheel cannot grind these tools because of their extreme hardness.

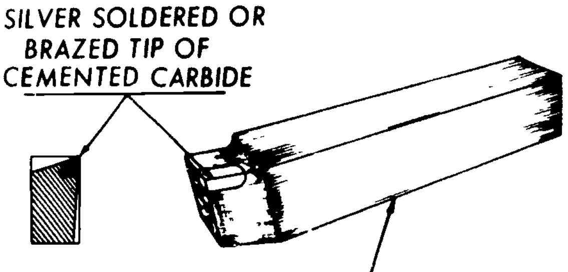


Figure 9 Carbide-tipped tool.

2-6. **Cutting Tool Shape.** The shape or contour of a cutting tool has a decided effect on its cutting efficiency. Most tools are hand-ground to shape on a *bench* or *pedestal grinder*. Portions of the tool are ground away to leave sharp and strong cutting edges. Except for the rules for grinding the proper rake and relief angles, there are no definite rules to govern the shape of lathe cutting tools. They may be square, pointed, small or large in radius, or irregular in shape. For certain classes of work, the cutting edge may be ground to fit gages of various shapes. Some of the more common lathe tool shapes, along with their application, are shown in figure 10. Lathe cutting tools may be either right-hand (right-cut) or left-hand (left-cut). The cutting edge of a right-hand tool is on the right-hand side when it is viewed from the point end of the tool with the top surface up. Figure 11 shows how to determine whether or not the tool is a right- or a left-hand tool.

2-7. **Cutting Tool Geometry.** Tool geometry pertains to the various angles and radii that shape

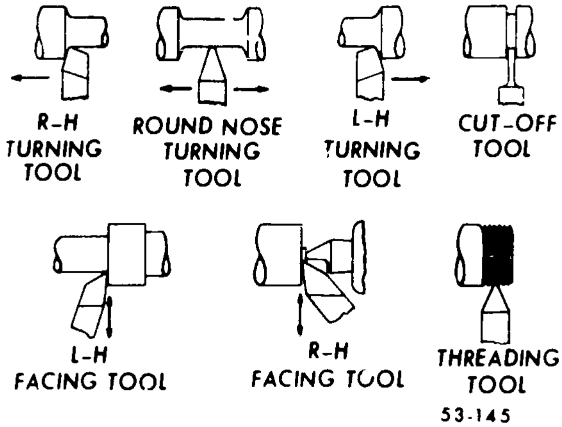


Figure 10. Cutting tool shapes.

the tool and enable it to cut. Tools having the correct geometry cut efficiently, produce good finishes, and maintain their sharpness. Conversely, improperly ground tools do not perform satisfactorily.

2-8. *Relief angles*. Relief angles are the angles formed by the intersection of the surfaces below and adjacent to the cutting edge with a plane perpendicular to the base of the tool. There are two types of relief angles, side and end, as shown in figure 12. Relief angles provide clearance to prevent the tool from rubbing on the work. They are often referred to as *clearance angles*. The tool is held in the toolholder at an angle of approximately 14½°. You must take this into consideration when you grind the end relief. For example, if the effective end relief is to be 8°, the actual end relief angle must be 22½°. The toolholder has no effect on the side relief. Table 2 gives the recommended relief angles for various metals.

2-9. *Rake angles*. Rake angle pertains to the top surface of the tool bit. There are two types of rake angles, back rake and side rake, as shown in

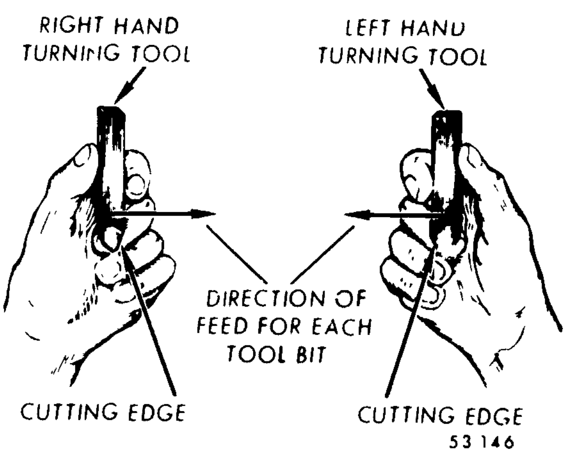


Figure 11. Left- and right-hand tools

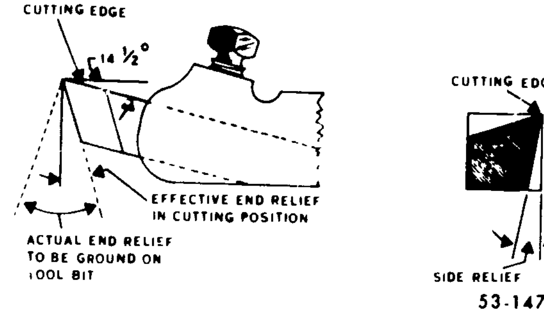


Figure 12. Relief angles.

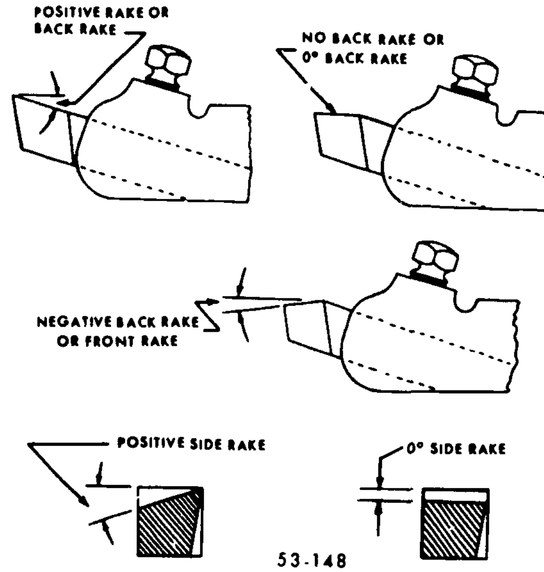


Figure 13. Rake angles.

figure 13. These angles may be positive or negative, or they may have no rake. The toolholder automatically gives a tool a 14½° effective back rake angle, and you must take this angle into consideration when you grind the tool. For example, to obtain an effective back rake of 16½°, you grind the tool with an actual angle of 2°. To obtain an effective angle of 0°, you must grind an angle of 14½° off the top of the tool, as shown in figure 13. The toolholder does not affect the side

TABLE 2
RECOMMENDED RELIEF AND RAKE ANGLES

Material	Side Relief Angle	End Relief Angle	Back Rake Angle	Side Rake Angle
Low carbon steel	12°	3°	16 1/2°	18°
Medium carbon steel	10°	8°	12°	14°
High-carbon steel	10°	8°	8°	12°
Cast iron	10°	8°	5°	12°
Stainless steel	12°	10°	16 1/2°	10°
Copper	14°	12°	16 1/2°	20°
Bronze	10°	8°	0°	0°
Brass	10°	8°	0°	0°
Aluminum	[illegible]	8°	35°	15°
Monel	15°	13°	8°	14°
Silicon bronze	10°	10°	10°	6°

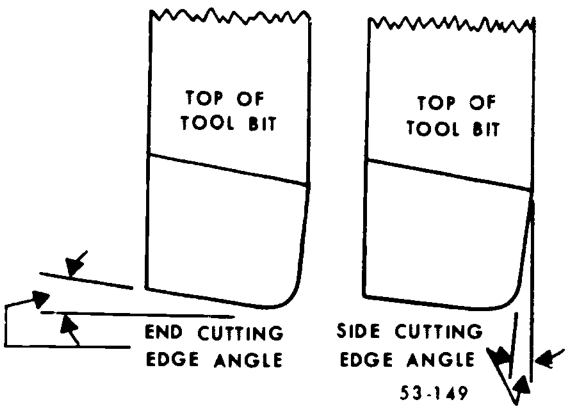


Figure 14. Cutting edge angles.

rake angle. You measure rake angles from a plane parallel to the base of the tool. Table 2 gives the recommended back rake and side rake angles for various metals.

2-10. *Cutting edge angles.* Cutting edge angles are the angles formed by the cutting edge with the end of the tool (the end cutting edge angle) or with the side of the tool (the side cutting edge angle), as shown in figure 14. The end cutting edge angle permits the nose of the tool to make contact with the work and aids in feeding the tool into the work. This angle is usually 8° to 15°. The side cutting edge angle reduces the pressure on the tool when it begins to cut. A side cutting edge angle of 15° is recommended for rough turning operations.

2-11. *Wedge angles.* Wedge angles are the angles formed by the front relief and back rake or by the side relief and side rake, as shown in figure 15. These angles are usually 60° to 65°. However, when you grind the tool to the recommended relief and rake angles, the wedge angles may vary.

2-12. *Nose radius.* The nose radius, as shown in figure 16, strengthens the tip of the tool, helps to radiate the heat generated by the cutting action, and helps to obtain a good finish. You will normally grind a $\frac{1}{64}$-inch nose radius on tools $\frac{3}{8}$ inch square or smaller, and a $\frac{1}{32}$-inch radius on tools from $\frac{3}{8}$ inch to $1\frac{1}{4}$ inches square. Some tools, such as the side-finishing tool, require very

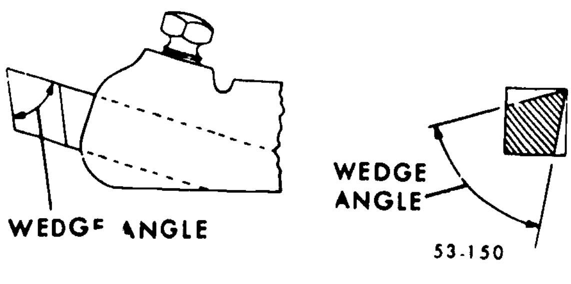


Figure 15. Wedge angles.

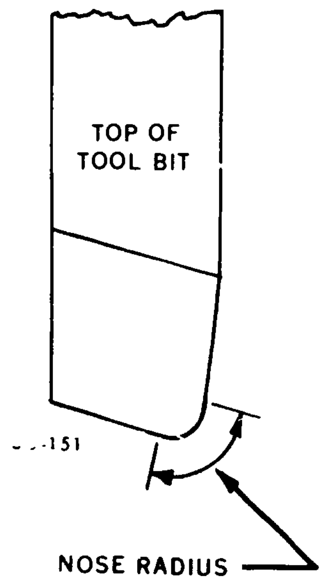


Figure 16. Nose radius.

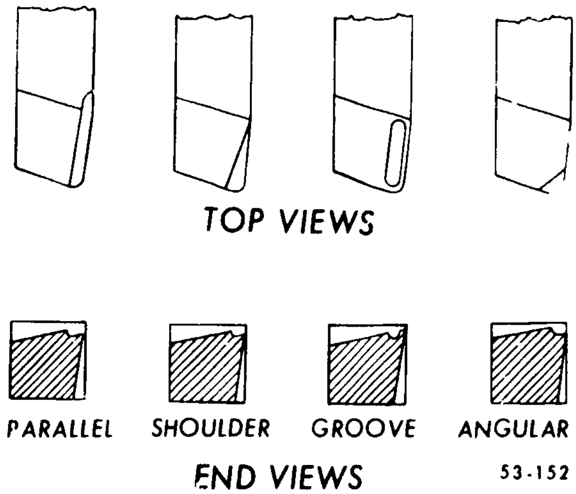


Figure 17. Chip breakers.

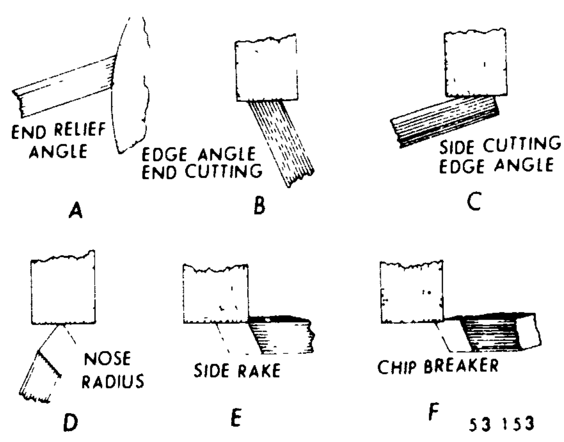


Figure 18. Grinding a tool.

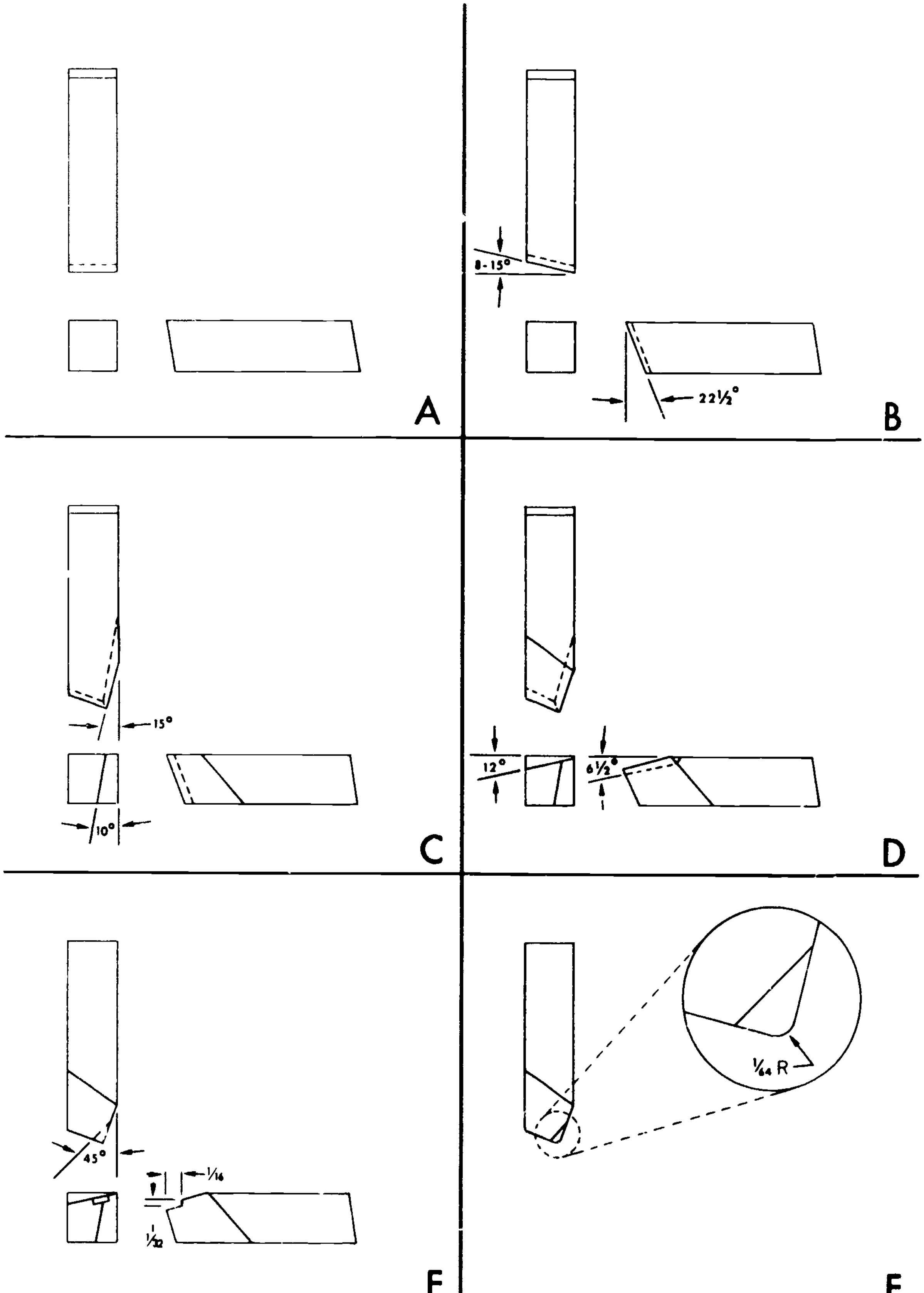


Figure 19. Development of a roughing tool.

little, if any, nose radius. A radius that is too large may cause the tool to chatter because of excessive tool contact.

2-13. *Chip breakers*. Chip breakers are indentations on the top surface of the tool that prevent the formation of long and dangerous chips. Broken or short chips occupy less space, reduce the amount of heat transferred from the chips to the tool, and permit a better flow of coolant to the cutting edge. Figure 17 shows some of the various types of chip breakers. You usually grind chip breakers only on roughing tools.

2-14. **Grinding Cutting Tools.** A tool must be ground to the correct angles; and it must be sharp, or it will not cut properly. Use a protractor head and blade to check the tool angles as the grinding progresses. The following example will help you understand how the various angles are ground. Although the tool we are using for this example is a right-hand roughing tool for high-carbon steel, you can apply the information to nearly any type of tool. We will be referring to figures 18 and 19 frequently during the grinding of the tool.

(1) Assemble the necessary materials and tools.

NOTE: You will need the following items:
- Tool blank, as shown in figure 19,A.
- Protractor head and blade.
- $^{1}/_{64}$-inch radius gage.
- Grinding wheel dresser
- Oilstone.

Wrench to fit the pedestal grinder tool rest.

(2) True and dress the grinding wheel, if necessary, and reposition the tool rest.

(3) Position the tool, as shown in figure 18,A and B, and grind the end relief and end cutting edge angles.

NOTE: The recommended end relief angle for machining high-carbon steel, as given in figure 14, is 8°. The end cutting edge angle can be from 8° to 15°

Stop frequently while you are grinding the tool to measure the angles with the protractor and blade.

NOTE: Do not forget to compensate for the 14½° angle at which the tool will be positioned when it is held in the toolholder. The effective end relief angle of 8° is obtained by grinding an angle of 22½°. Figure 12 will help you visualize the difference between an effective and actual end relief angle.

The tool should look like the one shown in figure 19,B, after you have ground the end relief and end cutting edge angles.

(4) Grind the side cutting edge and side relief angles.

NOTE: Table 2 recommends a side cutting edge angle of 15° and a side relief angle of 10° for machining high-carbon steel.

Position the tool as shown in figure 18,C, at the recommended 15° angle and tilt the tool forward 10° in order to obtain the desired angles. The lower edge of the tool will contact the grinding wheel first when the tool is positioned correctly. Grind the side cutting edge and relief angles until the cutting edge is the length you want.

NOTE: A ⅜-inch length is usually sufficient for tools that are $^{5}/_{16}$ inch square. Longer lengths can be used on larger tools.

Stop frequently and check the surface you are grinding to insure that the angles are correct. The tool should look like the one shown in figure 19,C, at this point.

(5) Grind the back and side rake angles.

NOTE: The recommended back rake angle, given in table 2, for machining high-carbon steel is 8° and the recommended side rake angle is 12°. Remember to compensate for the 14½° toolholder angle when you grind the back rake. You will have to grind an actual back rake angle of *negative* 6½° in order to have an effective back rake angle of 8°.

Position the tool as shown in figure 18,E. Pull the right-hand end of the tool toward you 6½° to provide the negative back rake angle. Tilt the top of the tool toward you to obtain the 12° side rake angle and lower the right-hand end of the tool until the cutting edge of the tool bit is horizontal. When the tool is positioned properly, press it straight into the grinding wheel. Stop and measure the rake angles frequently to insure that you are grinding them correctly. Grind the rake angles until the surface that produces them joins the surface that forms the side relief and side cutting edge angles. When you have completed grinding the rake angles, the tool should look like the one shown in figure 19,D.

(6) If you desire to grind a chip breaker, such as the angular chip breaker shown in figure 17, do it now. Dress the grinding wheel if the corner of the wheel is slightly rounded.

NOTE: The corner of the wheel should be as square as possible. The necessary amount of curvature on the back surface of the chip breaker is provided by the breakdown of the grinding wheel corner as you grind the chip breaker.

Position the tool so that the back edge of the chip breaker is at an angle of approximately 45° to the cutting edge of the tool and the bottom surface of the chip breaker is parallel to the surface that formed the rake angles. Press the tool into the grinding wheel, as shown in figure 18,F. The chip breaker should be about 1/16 inch wide and 1/64 inch to 1/16 inch deep.

NOTE: The best width and depth depends upon the feed, depth of cut, and type of material you are machining. They can be determined only by trial and error.

The tool should look like the one in figure 19,E, after you have ground the chip breaker.

(7) Grind the nose radius. Position the tool so that the entire edge that is formed by the intersection of the side relief and side cutting edge surface lightly contacts the surface of the grinding wheel, as shown in figure 18,D. Grind the radius by swinging the tool in a slight arc. The edge contacting the surface of the wheel should be the pivot point for the arc. Allow the radius that forms to blend smoothly into the end and side surfaces of the tool, as shown in figure 16. Check the nose radius with the 1/64-inch radius gage.

NOTE: If the nose radius is too large, grind either the end or side surface of the tool to eliminate the radius. Then, grind a new nose radius.

When the nose radius is correctly ground, the tool looks like the one in figure 19,F.

(8) Hone the nose radius and each of the surfaces that form the cutting edge of the tool. Place one of the surfaces firmly against the oilstone and slide the tool toward the cutting edge. Repeat this operation until the surface is smooth and the cutting edge is sharp, then hone the remaining surfaces. Finally, hone the nose radius. Place the nose radius against the oilstone and, as in grinding, swing the tool in an arc. Hone the radius until it is smooth and blends into the end and side surfaces of the tool.

NOTE: Proper honing will remove the burrs that are left after grinding, sharpen the cutting edge, and reduce the friction between the chip and the surface of the tool, thereby increasing the life of the cutting edge and producing better finishes on the work. Careless honing dulls the tool and shortens its cutting life, and results in poor finishes. Be careful! Keep the tool and oilstone properly alined with each other and never permit the oilstone to drag across the cutting edge of the tool.

2-15. **Toolholders.** General-purpose toolholders are manufactured in straight; right-hand, and

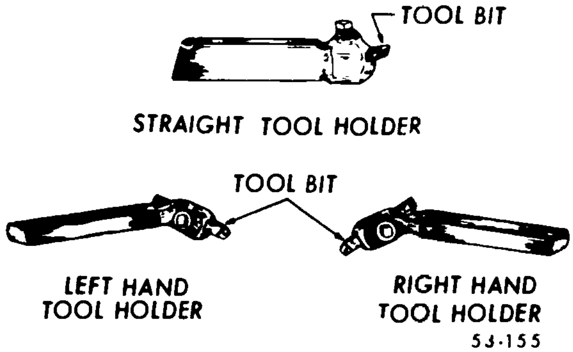


Figure 20. Toolholders.

left-hand types, as shown in figure 20. You usually use a left-hand toolholder with a right-hand tool and a right-hand toolholder with a left-hand tool. This enables the cutting edge to cut in the direction in which it is designed to cut with a minimum of interference from the toolholder and the compound rest. The left-hand toolholder is most often used for straight-turning operations, since it allows the tool to cut closer to the chuck or lathe dog than other types.

2-16. Toolholders have square holes into which the tools are fitted after they have been ground to the desired shape. The forged toolholders are available in right-hand, straight, and left-hand types, with a 30° offset for the offset toolholders. The hand of a toolholder is determined in much the same way as the hand of a cutting tool. Hold the toolholder vertically by the blunt end with the set screw toward you. If the tool angles to the left, it is a left-hand toolholder, and if to the right, a right-hand toolholder. Also, commercial toolholders generally have the letters L or R immediately after the model number embossed into their sides, which indicates the hand of the holder. Other types of toolholders are for special operations, such as parting, boring, and knurling, and will be discussed later in this volume.

3. Workholding Devices

3-1. Many different devices, such as chucks, collets, faceplates, driveplates, mandrels, and lathe centers, are used to hold and drive the work while it is being machined in a lathe. We will discuss these devices in this section. What workholding device should you select? There is no simple answer to this question. Your selection will be based on factors that differ for each particular job. However, all of the following factors should be taken into consideration when you select the workholding device or devices to be used for a particular work project:

- The shape of the work.
- The operation to be performed.

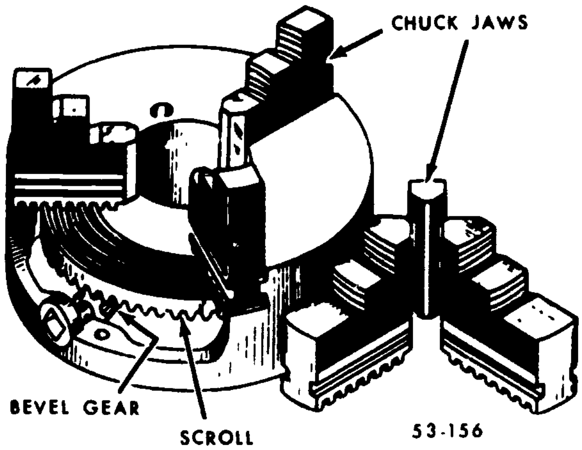


Figure 21. Three-jaw universal chuck.

- The degree of accuracy required in the alinement of the work.

You will find that the selection of workholding devices becomes easier as you gain experience and become more familiar with the capabilities of the various devices. The first workholding devices we will discuss are lathe chucks.

3-2 **Lathe Chucks.** Lathe chucks, which are clamping devices designed to hold and rotate work, are attached to the lathe spindle nose. The two chucks which you will use most often are the universal, with three or six jaws and the independent, with four jaws. You will also use the four-jaw combination chuck and collet chuck.

3-3. *Universal chuck.* The universal chuck is used to hold hexagonal or cylindrical work when extremely accurate alinement is not essential. The universal chuck has self-centering jaws which are opened and closed by a scroll that is driven by a bevel gear, as shown in figure 21. The jaws open and close simultaneously when any of the djusting sockets are turned.

CAUTION: To prevent accidents, always remove the chuck wrench from the chuck immediately after use.

Figure 22. Holding small and large diameter work.

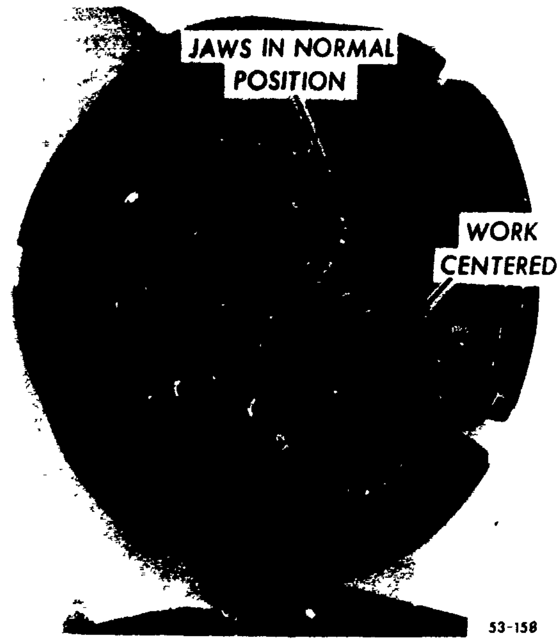


Figure 23. Four-jaw independent chuck.

3-4. You can chuck round or hexagonal material with a universal chuck easily and quickly because it has self-centering jaws. The jaws are not reversible on universal chucks; to enable them to hold large diameter work, some chucks have an extra set of jaws with stepped surfaces facing inward, as shown in figure 22. Others have jaws that are made in two sections; the upper section can be reversed. Remove the screws which hold the upper section in place reverse it, and replace the screws.

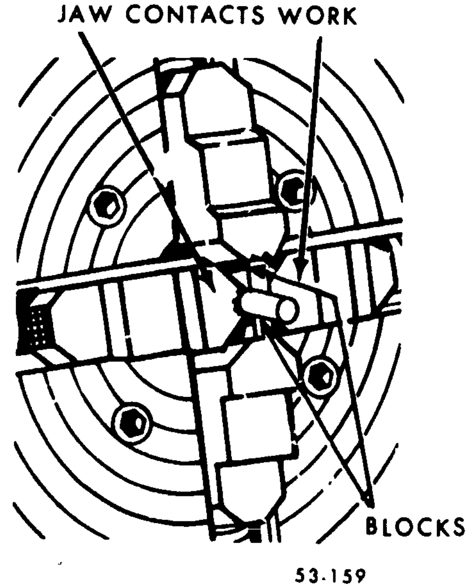


Figure 24. Small work held in a four-jaw chuck.

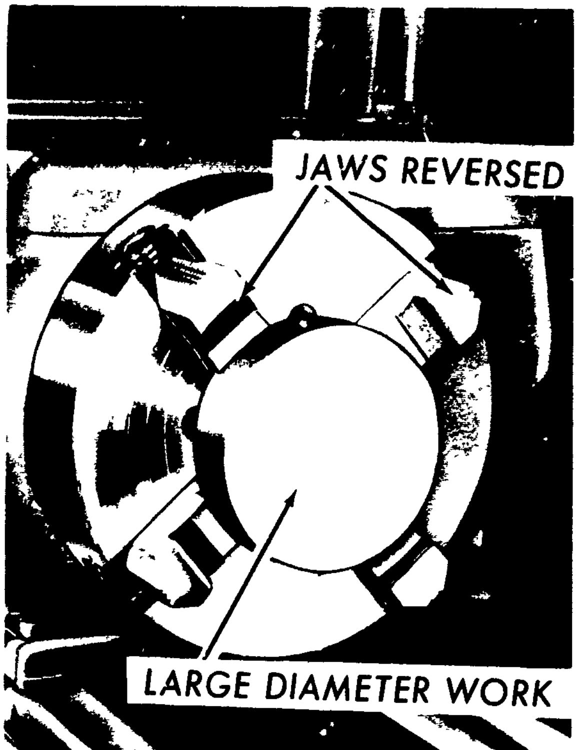


Figure 25. Jaws reversed.

NOTE: The chuck jaws and the jaw guide openings on universal chucks are numbered. Install the jaws in the jaw guide openings that have corresponding numbers and install them in numerical sequence.

3-5. *Four-jaw independent chuck.* The four-jaw independent chuck, as the name indicates, has four jaws that move independently of each other, giving it a wide range of application. You will use the four-jaw independent chuck to hold work that is not cylindrical; cylindrical work that must be alined to run concentric to a previously machined surface; work that must be alined off center; and work requiring the removal of a small amount of material, for instance, a shaft that must have the diameter reduced by 0.002 inch. Figure 23 shows a four-jaw chuck with round stock held on center. Figure 24 shows how very small material may be held with the aid of buildup blocks. The jaws of a four-jaw chuck may be reversed to hold work which is too large to hold normally, as shown in figure 25. The four-jaw chuck has the greatest holding power of any chuck.

3-6. Usually, the stock must run *true* (revolve concentrically) in the chuck. The work may be approximately alined with the aid of the concentric circles machined in the face of the four-jaw chuck by adjusting the jaws until corresponding points on the jaws are located the same distance from a con-

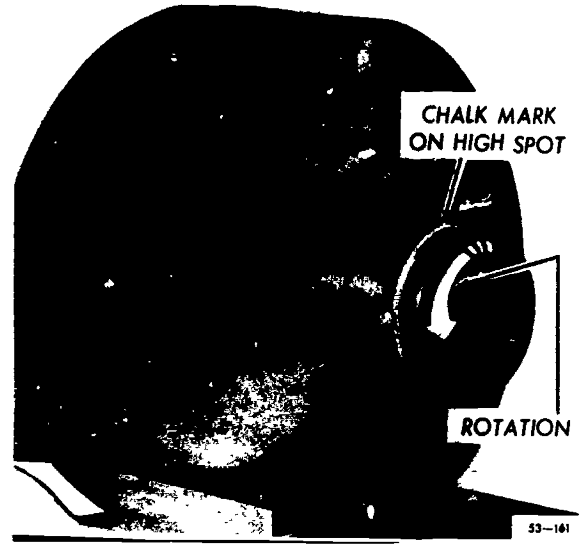
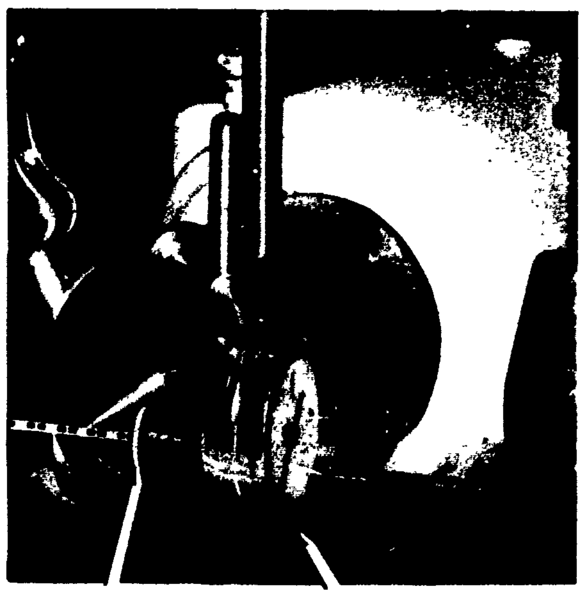


Figure 26. Checking trueness with chalk.

centric circle. You can obtain closer alinement by using a piece of chalk or a pencil. Operate the machine at a moderate speed and bring the chalk into contact with the revolving workpiece, as shown in figure 26. The chalk will contact only the high side of the work. The workpiece can be moved closer to center by loosening the chuck jaw opposite the mark and tightening the jaw nearest the mark. Repeat this operation until the chalk leaves a mark entirely around the workpiece. You can achieve the greatest degree of accuracy, however, by using a dial indicator to aline the work. The indicator plunger should be on center height, as shown in figure 27, and perpendicular to the axis of the



53—162

Figure 27. Positioning indicator at center height.

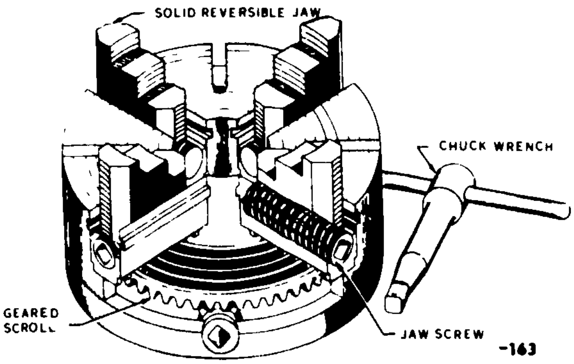


Figure 28. Four-jaw combination chuck.

work. Bring the plunger into contact with the work by means of the cross-slide. Revolve the chuck slowly by *hand* and locate the high point by observing the movement of the indicator needle. After adjusting the jaws, check the work again for runout and readjust as needed. Repeat this operation until the indicator needle does not move.

NOTE: Work should be held lightly in the chuck while it is being alined; it should be secured after it has been trued. A final adjustment may then be required. Soft metal shims should be placed between the chuck jaws and any finished surfaces to prevent marring the work.

3-7. *Four-jaw combination chuck.* The four-jaw combination chuck, shown in figure 28, has the features of both the universal chuck and the independent jaw chuck. You can adjust the jaws independently by turning the adjusting socket which operates the gear-driven scroll. This type of chuck is very useful when you are chucking duplicate pieces of irregular shape. Center one piece by adjusting the jaws independently and center similar pieces by turning the adjusting socket. The jaws may be reversed to accommodate work of large diameters.

3-8. *Collet chucks.* You can use collet chucks to better advantage, in many cases, than either the universal or independent jaw chuck. The accuracy with which collets are made, together with the method of gripping the work, eliminates the need for truing. However, there are limitations to the gripping power of the collets and also to the range of work sizes that can be held in the collet chuck. You should use collets when accuracy in alinement is required and when the object has a suitable finished gripping surface. Do not use collets on surfaces that are not smooth and uniform. You should not perform operations such as thread cutting on work held in collets because of the possibility of the work slipping. The three types of collet chucks in common use are the draw-in collet chuck, the spindle nose collet chuck, and the rubber-flex collet chuck.

3-9. The draw-in collet chuck assembly, shown in figure 29, consists of a split collet, collet adapter, and a handwheel and draw bar. A spindle-nose cap is usually provided to protect the threaded lathe spindle nose, which would otherwise be left exposed. Also, by tightening the nose cap against the back of the collet adapter, you can remove the adapter with little difficulty.

3-10. Split collets are made of hardened and tempered tool steel. They are accurately ground, both inside and outside, so that the hole (which provides a gripping surface), the body, and the tapered end are concentric. The large end of the collet is tapered to fit the conical opening in the collet adapter. This end is slotted to permit the collet to contract or expand when it is drawn into or released from the collet adapter in the lathe spindle. The other end of the collet is threaded to accommodate the draw bar. A keyway is cut in the threaded end to fit a key in the adapter. This key prevents the collet from slipping in the adapter.

NOTE: Always be sure the key is alined with the keyway before tighrening the draw bar. Otherwise, the threads on the collet could become damaged.

Figure 30 shows the various parts of a split collet. Standard collets are made in fractional sizes to hold finished round stock. They are furnished in sets; the sizes increase in steps of 1/64 inch, 1/32 inch, and 1/16 inch. On the smaller bench lathes,

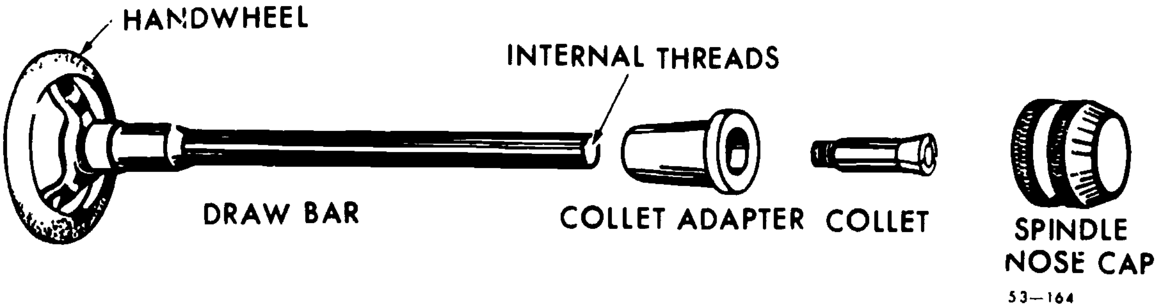


Figure 29. Draw-in collet chuck assembly.

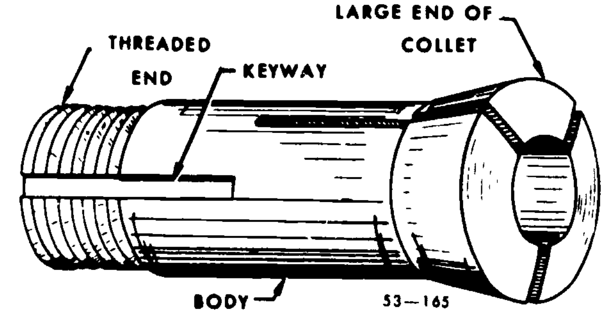


Figure 30. Split collets.

the maximum size stock which can be held in a collet varies from 1/2 inch to 3/4 inch. Each collet is marked on the face to indicate its size. Never select a collet that is no more than 0.005 inch larger and no smaller than the work to be held. Do not tighten a collet if no work is being held in it to avoid springing the collet and impairing its accuracy. Special collets are made to accommodate round material of wire gage sizes and square- and hexagon-shaped material.

3-11. The collet adapter, which is shown in figure 31, is accurately ground inside and outside to hold and center the collet in the lathe spindle. The outside of the adapter is tapered to fit the tapered bore of the lathe spindle. The front of the adapter has a conical opening which provides a seat for the tapered end of the collet and serves to contract the collet as it is "drawn in." The bore of the adapter is straight to accommodate the body of the collet.

3-12. The draw bar, shown in figure 29, consists mainly of two parts: a steel tube threaded at one end to receive the threaded portion of the collet and a handwheel which is attached to the opposite end. The draw bar fits the spindle bore and extends through the hole in the spindle to engage the threaded end of the collet. When the draw bar is rotated clockwise, it draws the collet into the collet adapter, causing the collet to contract and tighten on the work. Turning the handwheel counterclockwise relieves the pressure on the collet against the conical opening of the adapter. This enables the collet jaws to expand and release the work.

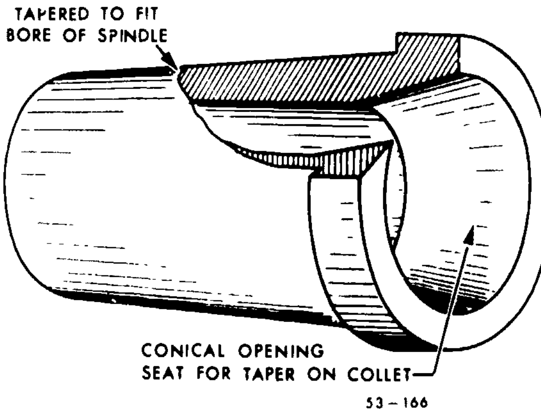


Figure 31. Collet adapter.

3-13. The spindle-nose collet chuck, such as the one shown in figure 32, mounts on the lathe spindle nose. It combines all of the parts of a draw-in collet assembly into one compact unit. The draw bar and collet adapter are part of the collet housing. Turning the handwheel tightens or releases the split collet. The collets are similar in application and restrictions to those which are used in draw-in collet assemblies.

3-14. The rubber-flex collet chuck shown in figure 33, is similar to the spindle-nose type except for the collet, which consists of steel strips embedded in rubber. This allows them to expand and

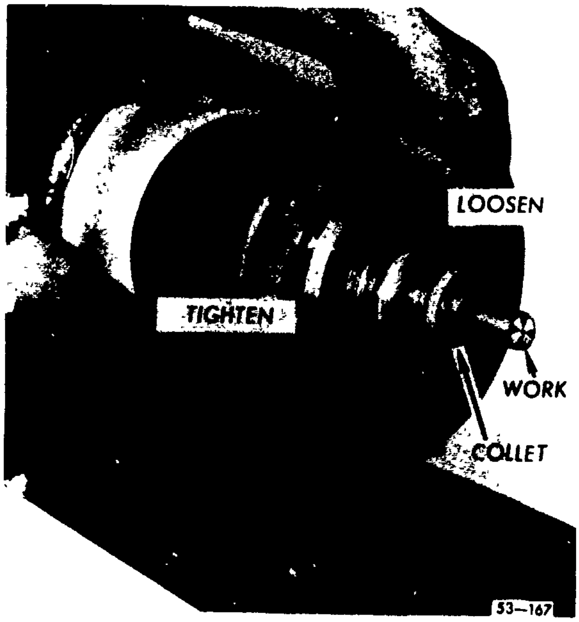


Figure 32. Spindle-nose collet chuck.

contract much farther than the other types, giving them a much greater capacity. It requires only 11 rubber-flex collets to hold work of any diameter from 1/16 inch to 1⅜ inches. Allowing 0.005-inch capacity per collet, it would require 263 solid type collects to cover the same range of sizes. This type of collet is the weakest in holding power, especially with smaller diameter work (under ½"), so don't attempt heavy cuts on work held in a rubber-flex collet.

3-15. **Faceplate.** The faceplate is a large, round plate with T-slots and bolt slots machined in it, as shown in figure 34. It is mounted on the lathe spindle in the same manner as a chuck. You can use a faceplate to hold large, regularly shaped objects that would be difficult to hold in a chuck;

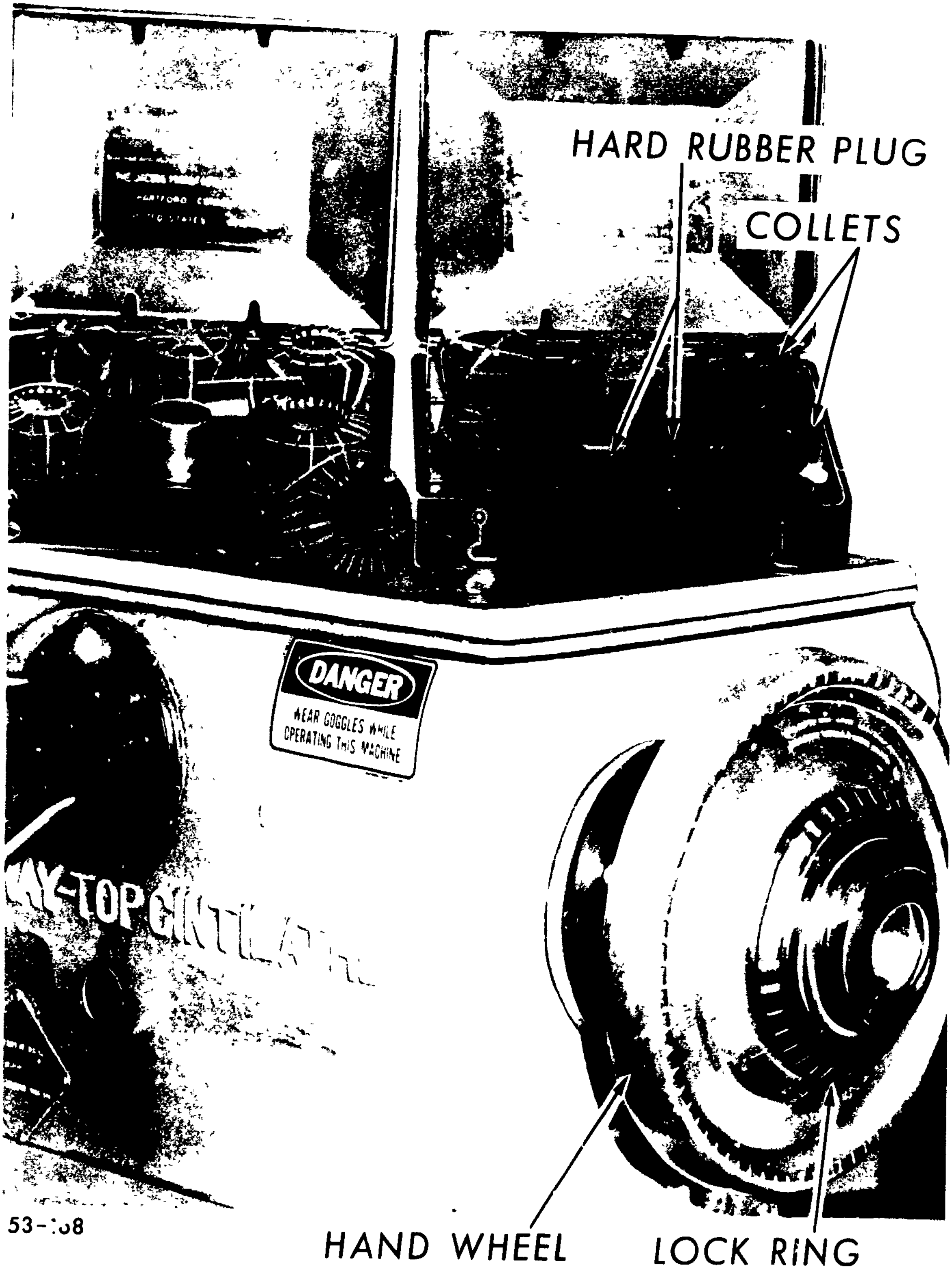


Figure 33. Rubber-flex collet chuck.

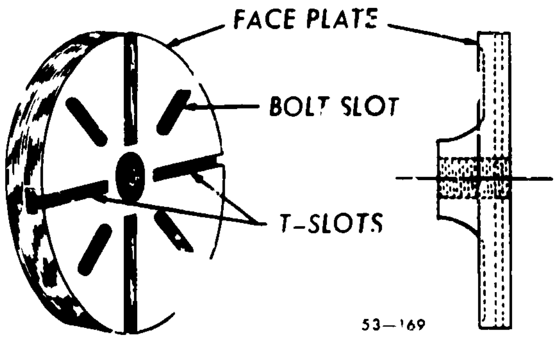


Figure 34. Faceplate.

thin, flat work that would be distorted by the chuck jaws; and work that requires the machining of offset holes. You can use many different clamping devices, such as T-bolts, clamps, and shaper stops, to clamp or hold the work to the faceplate. Figure 35 shows how flat objects can be clamped to a faceplate. An angle plate may be mounted on

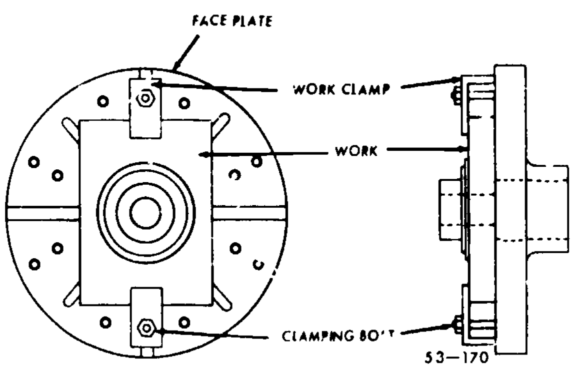


Figure 35. Work clamped on faceplate

the faceplate and, in turn, odd-shaped work, such as the elbow in figure 36, may be fastened to the angle plate. Notice the counterweight in figure 36. When a heavy piece of work is mounted off center, especially when it is fastened to an angle plate, a counterweight should be used to help offset vibration.

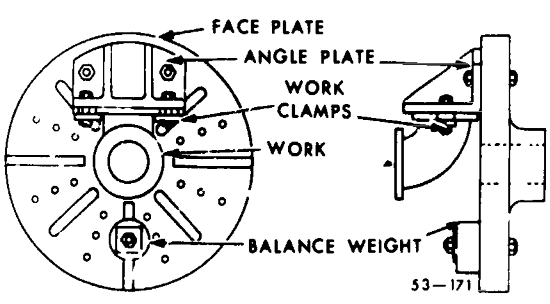


Figure 36. Work mounted on an angle plate.

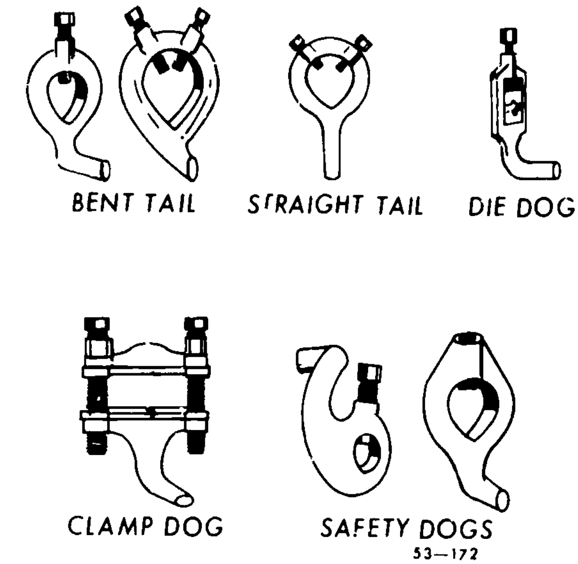


Figure 37. Lathe [illegible]s.

3-16. Prior to mounting work which requires highly accurate machining on a faceplate, it is good practice to first check the faceplate with a dial indicator. If the faceplate does not run true, a light cut may be taken across the face to true the surface. Before taking the cut, be sure to recheck the spindle nose and the threaded or tapered hole in the faceplate, because the runout may be caused by a small chip, by dirt, or by a burr on the thread. If you then determine that a cut is necessary, you should take a cut which does not exceed the maximum amount of runout. Do not mount the faceplate on another lathe, because the faceplate may not run true and you may have to make another truing cut.

NOTE: Take every precaution to protect the surface of the faceplate. Any shifting or slippage from

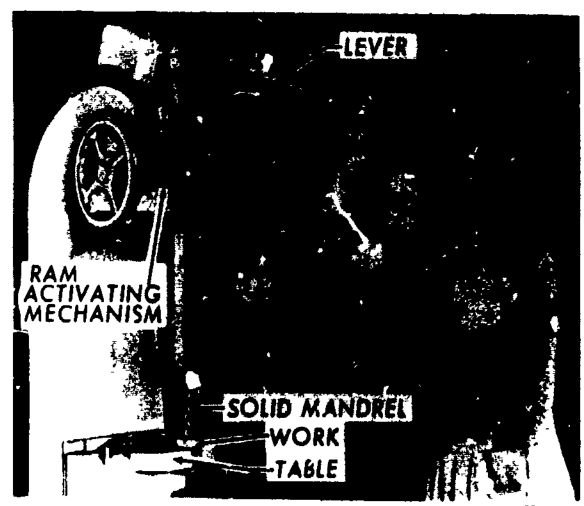


Figure 38. Installing a mandrel with an arbor press.

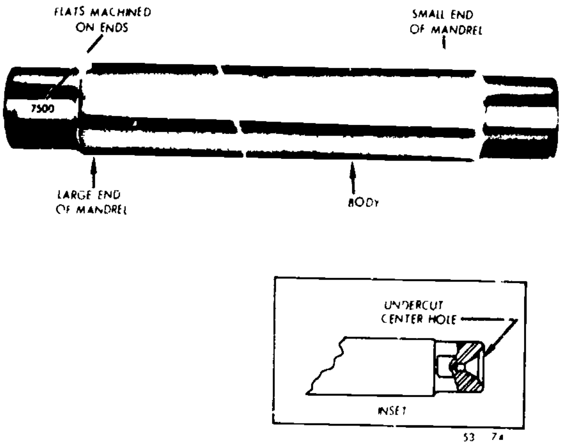


Figure 39. Solid mandrel.

the pressure of the machining will damage the work and mar the surface of the faceplate. Place a piece of paper or shim stock between the work and the faceplate to prevent the work from slipping. When you clamp heavy or awkward pieces to the faceplate. it is sometimes advisable to mount the work with the faceplate lying on a bench.

3-17. **Driveplate.** The driveplate is similar to the face plate and is attached to the lathe spindle in the same manner. It has from one to four radial slots machined in it. The tail of the lathe dog is inserted in a slot when work is machined between centers, and the lathe dog is driven by the driveplate. Lathe dogs are devices which are clamped to the work so that it can be revolved by the driveplate. These lathe dogs are made in various shapes to permit setting up different types of machining operations. Notice the safety lathe dogs shown in figure 37. They are designed to reduce the possibility of clothing being caught on them. You should use such safety dogs whenever possible. You should also place shim stock between the lathe dog and the workpiece to protect any finished surfaces. Do not permit the tail of the dog to bind in the driveplate slot. Driveplates do not have T-slots, and work should not be mounted on them.

3-18. **Mandrels.** Mandrels are cylindrical, metal shafts, which are used to hold and drive work. Work having a drilled, bored, reamed, or threaded hole may be mounted on a mandrel when it is necessary to machine the outer surface of the work concentric with the hole; e.g., the outside diameter of a gear which must be concentric to the hole in it in order to run true on the mounting shaft. An *arbor press* is used to press solid, expansion, and eccentric mandrels into the hole, as shown in figure 38. A thin film of white lead on the surfaces of the hole helps to prevent galling.

3-19. *Solid mandrel.* The solid mandrel, which is shown in figure 39, can only be used on work in which the hole has been accurately machined to a standard size. The solid lathe mandrel is a cylindrical tool steel bar, which is hardened, tempered, and accurately ground to a standard size. The body of the mandrel is ground with a slight taper of approximately 0.006 inch per foot. The taper provides a means of securing the mandrel to the work to insure that the work and mandrel will rotate together. The shank ends of the mandrel are turned smaller than the body size, and they are machined with a flat. The flat surface provides a positive drive when you tighten the setscrew of a lathe dog against it. (The lathe dog, you will recall. is driven by a driveplate.) The center holes on the mandrel are undercut below the ends of the mandrel to prevent their being damaged when the mandrel is pressed into the work. Solid mandrels are made in standard fractional sizes, with the size of the mandrel always marked on the large end to distinguish it from the small end. The small end of the mandrel is usually a half thousandth of an inch

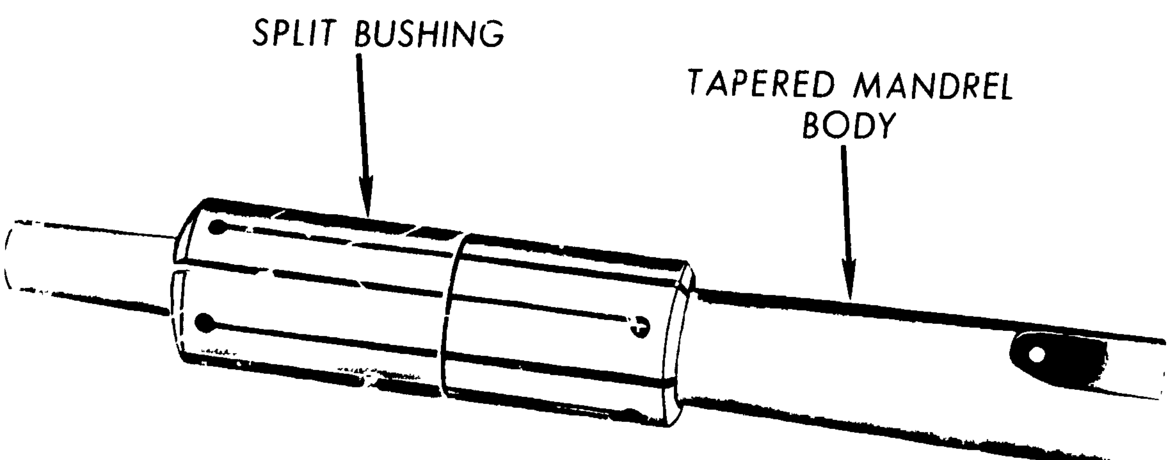


Figure 40. Expansion mandrel.

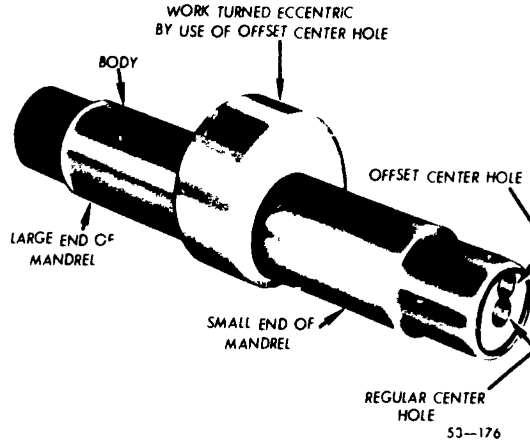


Figure 41. Eccentric mandrel.

smaller than the standard size on mandrels up to 1 inch in diameter. On the larger size mandrels, the small end is ground 0.001 inch undersize. The work is held on the mandrel and is driven by friction. In view of this fact, you should avoid taking excessively deep cuts which could cause the work to slip on the mandrel.

3-20. *Expansion mandrel.* Expansion mandrels are made to accommodate a wider range of variation in hole sizes than are solid mandrels. The most common type of expansion mandrel, which is shown in figure 40, has a tapered body and a split bushing that expands as the mandrel is forced into it. The amount of expansion depends upon the size of the mandrel.

3-21. *Eccentric mandrel.* The eccentric mandrel is identical to a solid mandrel except that it has two or more center holes at each end. By mounting the mandrel between the lathe centers on the offset center holes, you can turn the outside diameter of the work eccentric to its bore, as

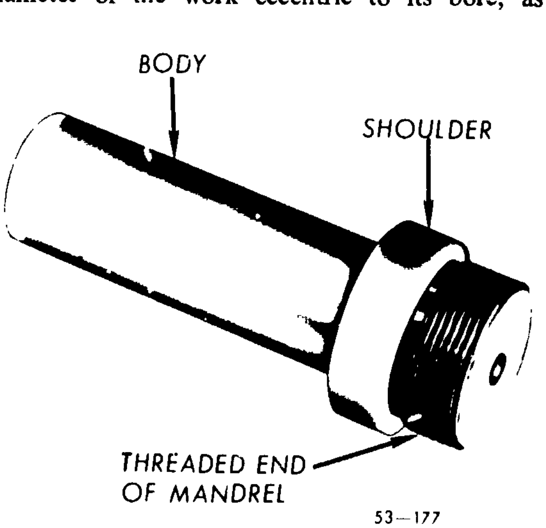


Figure 42. Nut mandrel.

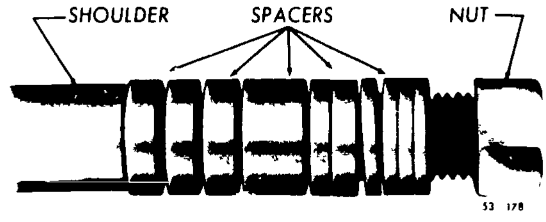


Figure 43. Gang mandrel.

shown in figure 41. The amount of eccentricity which can be machined on the work is governed by the distance of the offset center hole from the original center hole in the mandrel.

3-22. *Nut mandrel.* Nut mandrels, figure 42, are used to hold work when it is necessary to machine surfaces concentric to threaded holes. The work is screwed on the mandrel, and the shoulder acts as a stop and helps drive the work.

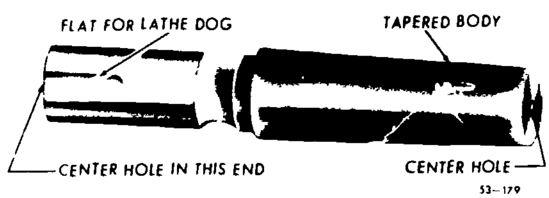


Figure 44. Tapered plug mandrel.

3-23. *Gang mandrel.* Gang mandrels are used to hold several pieces of work which require machining to the same outside diameter so that the outside diameters may be machined together and in proper relation to the reamed or bored holes in the work. The nut holds the workpieces tight against the shoulder of the mandrel. Figure 43 shows several spacers that have been machined on a gang mandrel.

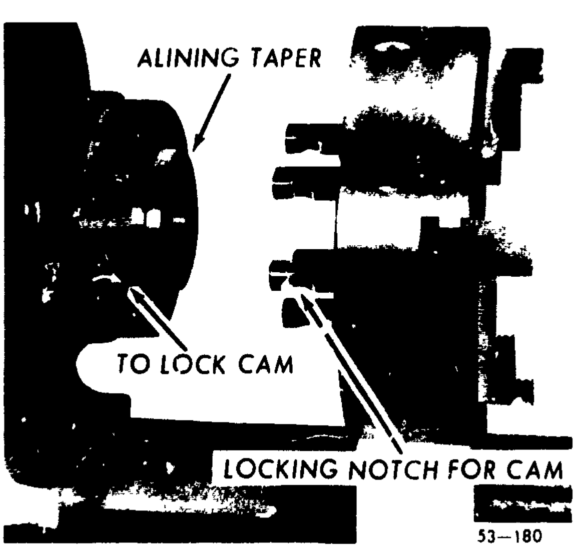


Figure 45. Cam lock spindle.

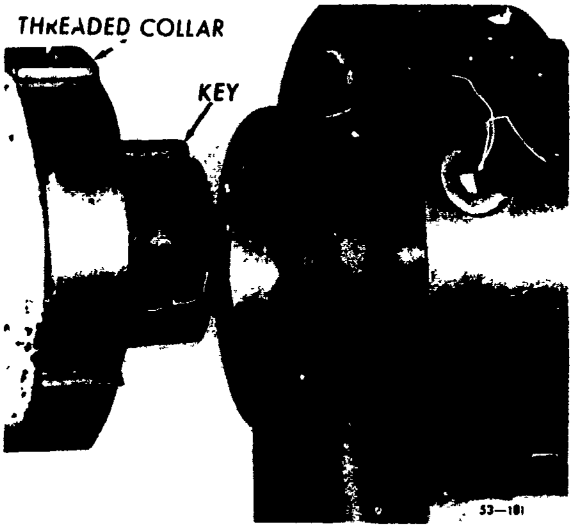


Figure 46. Tapered spindle.

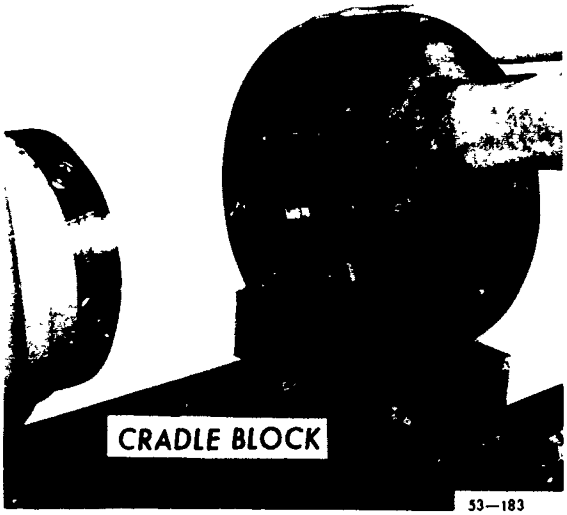


Figure 48. Placing chuck on cradle block

3-24. *Tapered plug mandrel.* Tapered plug mandrels, of the type shown in figure 44, are used to hold work having tapered holes. The taper on the mandrel must correspond to the taper in the work.

3-25. **Mounting Workholding Devices.** Faceplates, driveplates, nose caps, and chucks are all attached to the lathe spindle in the same manner. We will discuss the mounting of a chuck, but most of the information pertaining to mounting will apply to the other devices as well. Three types of spindles are commonly used on lathes: the cam lock, shown in figure 45; the tapered, shown in figure 46, and the threaded, shown in figure 47. Except for the method of securing the chuck to the spindle, mounting a chuck on any type of spindle is done in the same manner.

CAUTION: To prevent accidents, always turn the motor off while you are mounting or dismounting any device on the spindle.

NOTE: Setting the machine for a low spindle rpm helps to lock the spindle and makes installing the chuck easier.

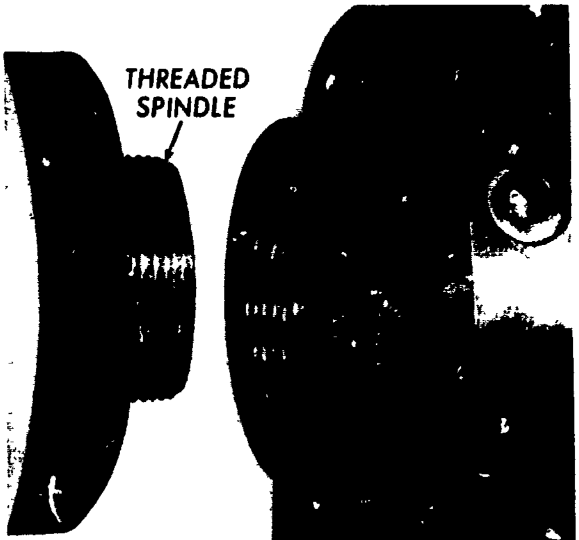


Figure 47. Threaded spindle.


Figure 49. Engaging chuck and a spindle nose.

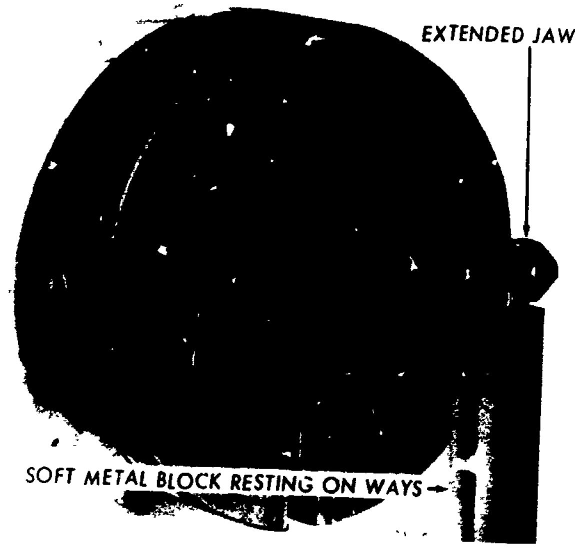


Figure 50. Use of a soft metal block.

3-26 Be sure that the spindel nose and the mounting surfaces on the chuck are clean and free of chips and burrs, or the chuck will not run true. Never place the chuck on the ways of the lathe; place it on a wooden cradle block, such as the one shown in figure 48, to prevent damage to the ways or injury to your fingers. (You do not need a cradle block when you are attaching devices light enough to hold safely by hand.) You should leave the cradle block in place until the chuck is safely engaged The additional support given by the cradle block helps to prevent dropping the chuck. The method which you should use to secure the chuck depends upon the type of spindle Secure chucks that are mounted on cam lock spindles by turning the cam sockets (with the special wrench which is furnished) in the direction indicated by arrows on the spindle nose. Secure chucks that are mounted on tapered spindles by tightening the threaded collar with the special spanner wrench. You may further tighten the collar by striking the handle end of the wrench with a lead mallet. Secure chucks that are mounted on threaded spindles by screwing the chuck within a half turn of the shoulder and spinning it into contact, as shown in figure 49.

3-27. **Dismounting Workholding Devices.** Removing a chuck, or other device, from the lathe spindle requires observing many of the precautions that you must observe when you mount a chuck. Turn the machine off, set it for a low rpm, and use a cradle block when you remove chucks.

3-28. On cam lock spindles, turn the socket counterclockwise until lines machined on the socket and spindle nose are alined, then slide the chuck onto the cradle block.

CAUTION: Keep your fingers out of the way as you guide the chuck onto the cradle.

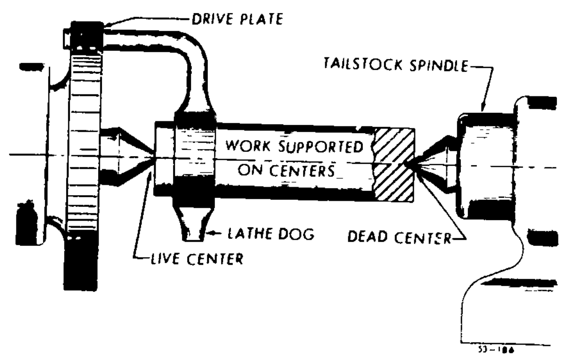


Figure 51. Work mounted between centers.

The threaded collar, which is used to secure the chuck on tapered spindles, is loosened with the spanner wrench. Strike the end of the wrench with a lead mallet, if necessary. Turn the collar by hand until it is free of the chuck, and slide the chuck onto the cradle block. To loosen a chuck on a threaded spindle, partially back out one of the chuck jaws. Insert a chuck wrench in an adjusting socket and, using the chuck wrench as a lever, turn the chuck backward until the extended jaw strikes a soft material such as brass, aluminum, or wood block on the ways, as shown in figure 50. The momentum of the gearing will cause the spindle to break free of the chuck and permit the chuck to be unscrewed by hand. The chuck may drop from the spindle quite suddenly, so be sure that the cradle block is in position before you unscrew the chuck. Some of the chucks for large lathes are very heavy and awkward to handle. To assist in mounting and dismounting these heavy chucks, a short piece of metal (6–8″ long) can be tightened into the jaws and used as a handle.

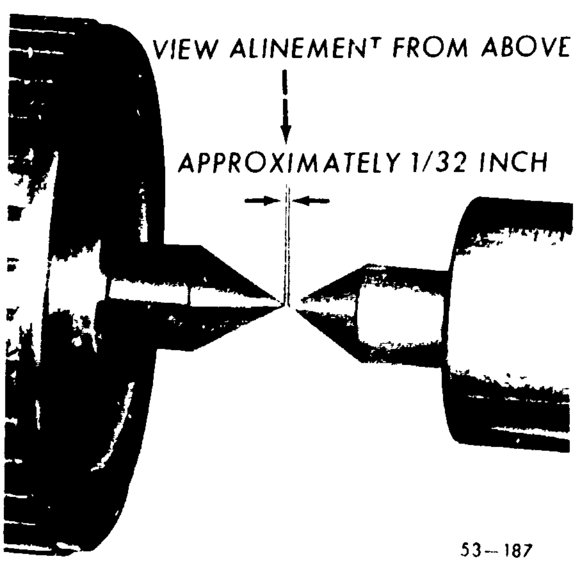


Figure 52. Alining centers visually.

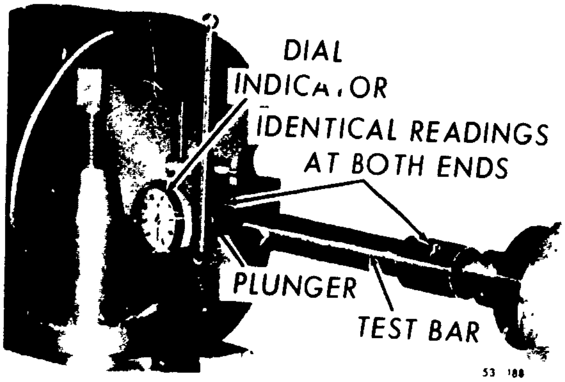


Figure 53. Indicator and test bar.

CAUTION: Do not place your hands under the chuck at any time while it is being removed.

3-29. **Lathe Centers.** Turning work held between centers is a common lathe operation. Figure 51 shows a typical setup for machining work between centers. The chief advantage of machining work that is held in this manner is that you can remove the work from the lathe and replace it later for subsequent machining operations without disturbing the trueness of the turned surface in relation to the center holes. The tailstock center, which does not rotate, is called the *dead center.* The headstock center is called the *live center.*

3-30. *Center alinement.* Work that is to be tur ·d straight and true between cen' requires exact alinement of the centers; i.e., th .s of the live center must aline with the axis oı the dead center. They are alined by moving the tailstock laterally (at a right angle) to the ways of the lathe.

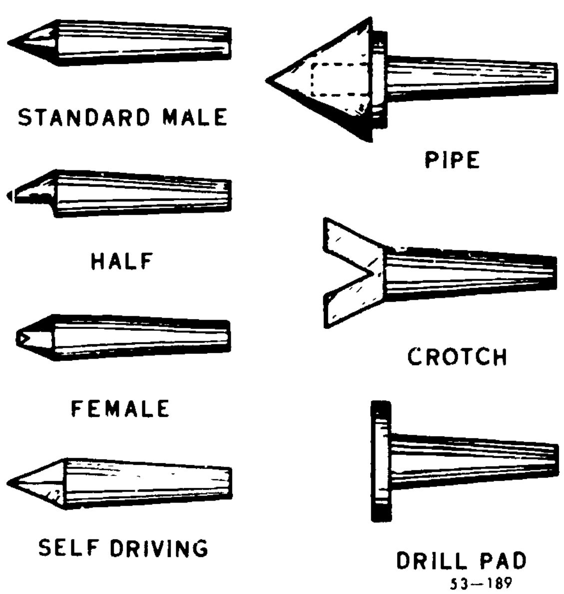


Figure 54. Lathe centers.

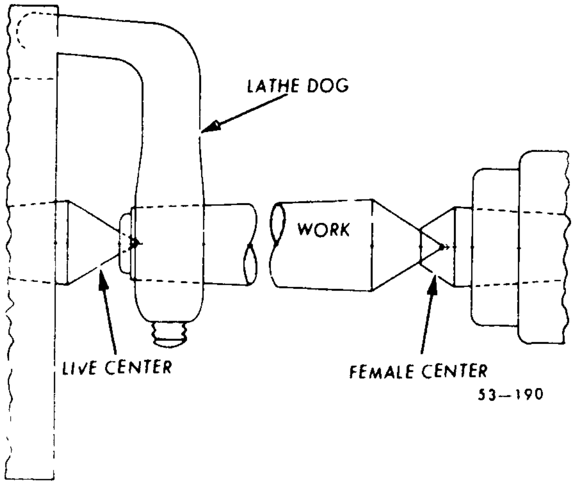


Figure 55. Use of female centers.

3-31. The method of checking the accuracy of alinement depends largely upon the nature of the work. We will now discuss the methods which are most commonly used to check center alinement.

3-32. The *cricket mark* method is an approximate method of alinement and is satisfactory when great accuracy is not required. The cricket marks on the tailstock are adjusted to read 0–0 by moving the tailstock lateral adjustment screws.

3-33. The *visual* method is also an approximate method of alinement; therefore, it is not suitable when great accuracy is required. Move the tailstock forward on the ways until approximately 1/32 inch remains between the points of the headstock and tailstock centers, as shown in figure 52. Then observe the centers visually from *above* for alinement, and adjust the tailstock as necessary.

3-34. The *dial indicator* and *test bar* method is a positive method of alinement which is used when accurate alinement of the centers is necessary. A

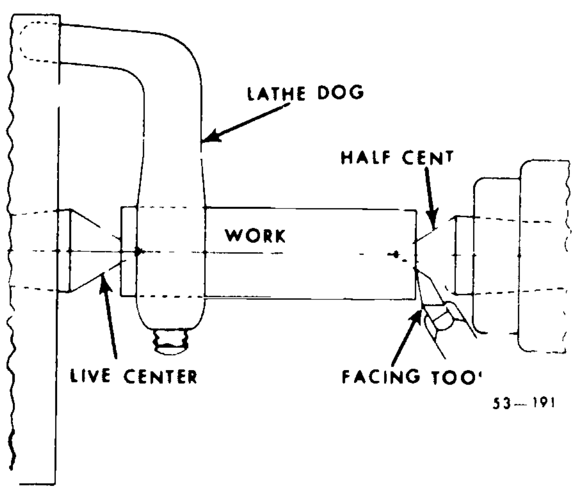


Figure 56. Facing work between centers

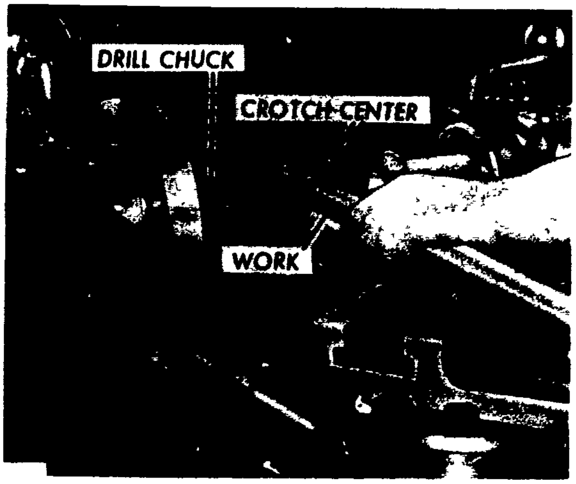


Figure 57. Supporting work with a crotch center.

test bar is an accurately ground cylinder with the same diameter at both ends; the cylindrical surface is concentric and parallel to the axis of the center holes. Test bars are available in various diameters and lengths. Place the test bar between the centers and take a reading at each end of the bar with a dial indicator mounted on the compound rest. Adjust the tailstock until you get identical readings on both diameters of the test bar. Figure 53 shows how an indicator and test bar are positioned to check the center alinement.

3-35. The *cut* and *try* method is also a positive method of alinement; it may be used alone or in conjunction with the other alining methods to obtain accurate alinement. Place the piece to be machined between centers and take a trial cut. Measure the diameter at both ends and move the tailstock toward you for work that is larger at the tailstock end and away from you for work that is smaller at the tailstock end. If it is necessary, take more cuts and adjust the tailstock until the two diameters are the same size or until the amount of taper is acceptable.

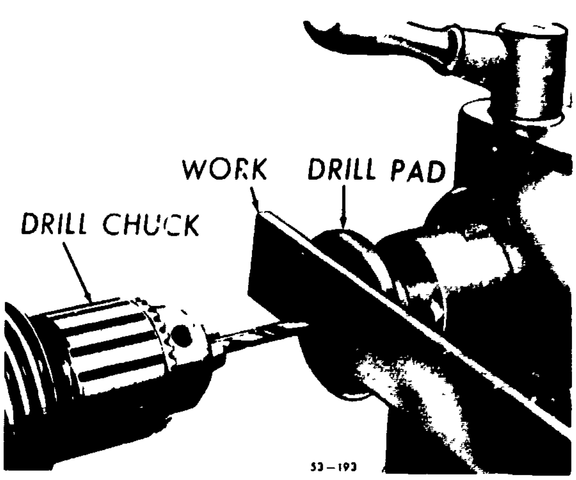


Figure 58. Supporting work with a drill pad.

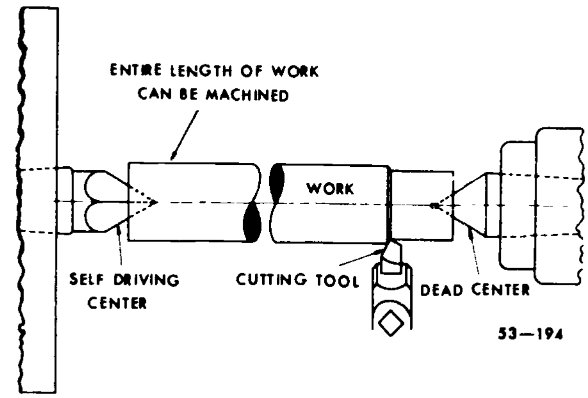


Figure 59. Supporting work with a self-driving control.

NOTE: The length of the test bar or cut and trial piece should equal the length of the work as nearly as possible. This will prevent disturbing the position of the tailstock, which would destroy to some degree the accuracy of alinement.

3-36. *Types of lathe centers.* Various types of centers may be used for general lathe work. The nature of the work must be taken into consideration when you select the type of center to be used. Refer to figure 54 as we discuss the most common types of lathe centers and their uses.

a. Male centers have a conical point with a 60° included angle. The male center is the most common type. Figure 51 shows work mounted on male centers.

b. Female centers have a 60° conical center hole in the tip. They are inserted in the headstock or tailstock spindle, and they support work that has pointed ends, as shown in figure 55.

c. Half centers are similar to male centers, except that a portion of the point is ground away. They are used only for facing work held between centers as shown in figure 56.

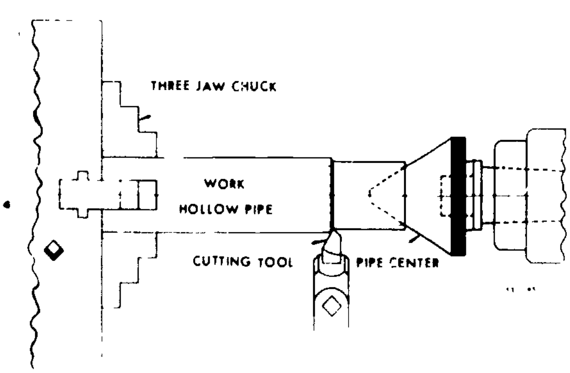


Figure 60. Supporting work with a pipe center.

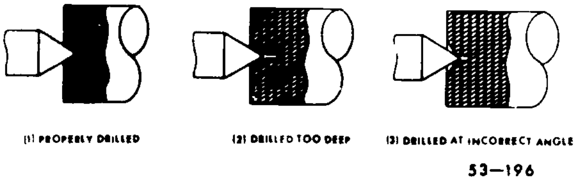


Figure 61. Correct and incorrect center holes

d. Crotch centers have a 90° groove machined in their face. They are used to support round stock when holes are drilled 90° to the axis of the stock, as shown in figure 57.

e. Drill pad centers have a circular disc or pad on one end. When a center is inserted in the tailstock spindle, the pad supports flat work for operations such as drilling, as shown in figure 58.

f. Self-driving centers have a point in the shape of a pyramid; they are frictiction-driven and friction retained. When a center is inserted in the headstock spindle, it supports and drives work on which a full-length cut is required, as shown in figure 59 The material to be machined should be relatively soft so that the center may be easily pressed into one end of the workpiece. Work that is supported between centers in this manner is not as rigid as work that is supported between male or female centers; therefore, only light cuts should be taken.

g. Ball bearing 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 lubrication when this type of center is used.

h. Pipe centers are bearing centers with a very large conical point They are inserted in the headstock and tailstock spindles to support pipe, tubular work, and other hollow work. As shown in figure 60, pipe centers are often used in conjunction with lathe chucks.

NOTE: The points of dead centers may be burned off or damaged if they are allowed to overheat. Avoid overtightening the tailstock center and keep the dead center hole well lubricated with a mixture of white or red lead and lubricating oil.

3-37. *Center drilling work.* Work that is supported between centers must have center holes drilled in the ends of the work to provide bearing surfaces. Drill the center holes to conform to the 60° included angle of the centers, as shown in figure 61. Center holes are usually drilled with a combination drill and countersink, or as it is commonly called, a *center drill.* Center holes are drilled by one of the following methods:

a. With the chuck holding the work and rotating, feed the center drill into the work with the tailstock spindle. Face a small flat area in the center of the work with the point of the facing tool to allow the center drill to adjust itself more readily to the center of the work.

b. Drilling on a drill press.

c. Mount the center drill and drill chuck in the lathe headstock. Hold the work by hand, floating one end of the work on the tailstock center. A deep punch mark helps to support the work on the tailstock center while you are drilling the first center hole.

3-38. Drilling the center hole to the correct size is very important. These are some of the factors which govern the size of a center hole:

- Work diameter. The greater work diameter, the larger the center hole should be. Large center holes in small work are unsightly.
- Hardness of material. The softer the material, the larger the center hole should be. Center holes in soft material tend to spread under the pressure of the cut unless they are made quite large.
- Depth of cut. The heavier the cut, the larger the center hole should be. More pressure is exerted on the work during heavy roughing cuts; therefore, more bearing surface is needed for the centers.
- Operation being performed. Work subjected to heavy pressures, such as knurling operations, requires larger center holes than work subjected to light pressure, such as polishing operations.
- Shape of the work. The shape of the work and the heat treatment that will be required often determine the size of the center hole. Large center holes that are close to flutes and grooves may cause the work to crack during heat treatment.

Turning Operations

IN THIS AND the next two chapters, we will deal with the various machining operations which you will perform on the lathe. This chapter covers turning operations: facing and straight turning; machining shoulders, corners, and grooves; radii and form turning; taper turning; and drilling, reaming, and boring.

4. Facing and Straight Turning

4-1. We will discuss facing and straight turning first in this chapter because these are the two lathe operations which are the easiest for a beginner to learn. Facing and straight turning are the simplest lathe operations. They are the two most common and frequent operations performed. One or both of these operations are often done to prepare work for other turning operations.

4-2 **Facing.** Facing is machining the ends of a piece of stock smooth, flat, and perpendicular to the axis of the stock. You face work to cut it to the desired length and to produce a surface from which accurate measurements may be taken.

4-3. *Facing work held in a chuck.* Facing is usually performed with the work held in a chuck or collet. Allow the workpiece to extend a distance no more than 1½ tim. . the work diameter from the chuck jaws, and use finishing speeds and feeds. You may feed the tool from the outer surface of the work toward the center or from the center toward the outer surface. Normally, you should face from the outer surface toward the center. This method permits you to observe the cutting tool and layout line while you are starting the cut. It also eliminates the problem of attempting to feed the tool into the solid, center portion of the stock. Use a left-hand finishing tool and a right-hand toolholder, positioned as shown in figure 62, when you face work from the outer surface toward the center. Work that has a drilled or bored nole in the center may be faced from the center outward if you use a right-hand finishing tool.

4-4 The proper overhang of the tool, as shown in figure 63, and the toolholder, as shown in figure 64, is important. Too much overhang may cause vibration; too little overhang may not provide enough clearance, causing the chuck to strike the compound rest or toolholder.

4-5. Set the cutting tool on center height to permit it to cut properly and to avoid leaving a stub on the end of the stock. You may set the tool on center by alining it to the point of the tailstock center or to the center reference line machined on the tailstock spindle. You may also set the toll on center by alining it with a surface gage pointer set at center height or by taking a trial cut and setting the tool point to the center of the stub that may be left. Figure 65 shows the result of correct and incorrect positioning of the tool.

4-6. *Facing work mounted between centers.* Work that has been center drilled may be faced between centers se a half center, which allows the point of th tool to enter the center hole to support the tailstock end of the work, as shown in figure 56. The end of the tool must be ground to a point with a very small radius to provide clearance. The tool must be held in a straight or left-hand tool holder. Use fine feeds and light cuts to prevent breaking the tip of the tool. Be sure that the tool will move from the center hole toward the outer surface before you engage the power feed. Replace the half center with a standard male center as soon as you complete the facing operation. The half center does not provide enough support for general machining operations.

4-7. *Facing to length.* To insure that you will have enough material left to face the second end of the stock to length, remove only enough material from the first end to clean it up.

NOTE: If the work is to be mounted between centers, drill the first center hole before you remove the stock from the chuck.

4-8. Various methods are used to face work to the desired length. One method is to lay out the work and face the second end until the layout line is split. You can also measure the piece after every facing cut with a rule. Succeeding cuts are taken until you obtain the desired measurement. If you

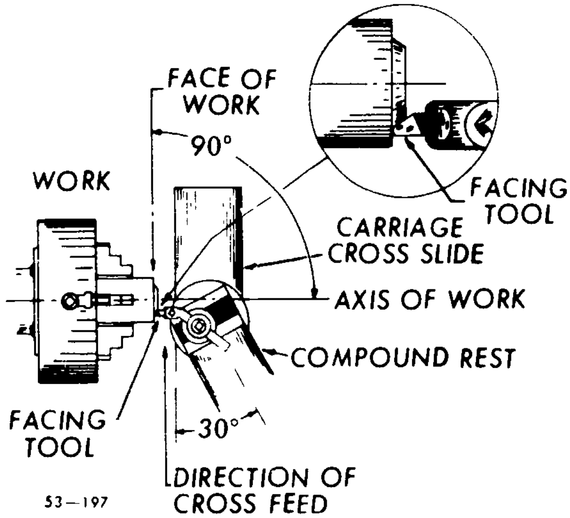


Figure 62. Facing setup.

desire greater accuracy, you can measure the length of the stock with a micrometer or vernier caliper. Make the required depth of cut with the compound rest. Use the graduated collar on the compound rest to indicate the exact depth of cut to be taken. Figure 66 shows the setup for facing to length with the compound rest. Note that the compound rest is parallel to the axis of the work.

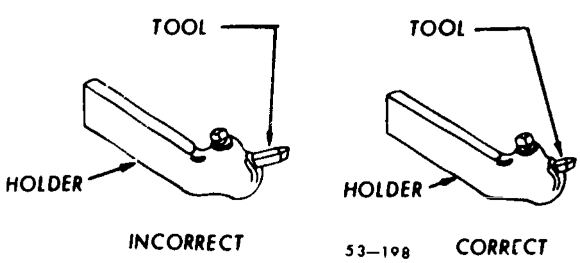


Figure 63. Cutting tool in toolholder.

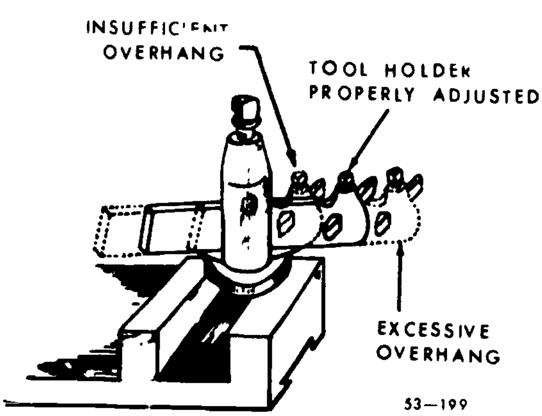


Figure 64. Toolholder overhang.

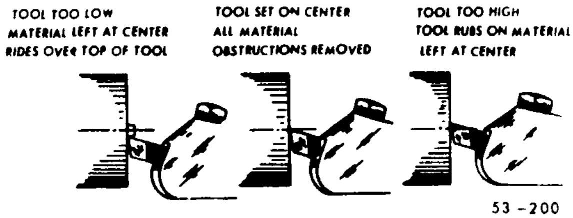


Figure 65. Effec of facing tool height.

NOTE: Lock the carriage to prevent it from moving and producing a surface that is not flat.

4-9. **Straight Turning.** Straight turning is the process of reducing the work diameter to a specific dimension. The carriage moves the tool parallel to the work axis.

4-10. *Machining to size.* Straight turn usually consists of a roughing and a finishing cut. When considerable metal is to be removed, you may take several roughing cuts. The cuts should be as heavy as the work, machine, and cutting tool can withstand.

4-11. If the work has two diameters, machine the larger diameter first. If you should accidentally machine the large diameter undersize, the piece may be salvaged by reversing the stock in the lathe and machining the large diameter on the untouched end.

CAUTION: Do not attempt to machine to a layout line or shoulder with power feed. Disengage the power feed approximately $\frac{1}{32}$ of an inch from the line or shoulder and feed the carriage by hand the remaining distance.

4-12. Usually, you should leave 0.010 inch to 0.020 inch on the diameter of the work for the fin-

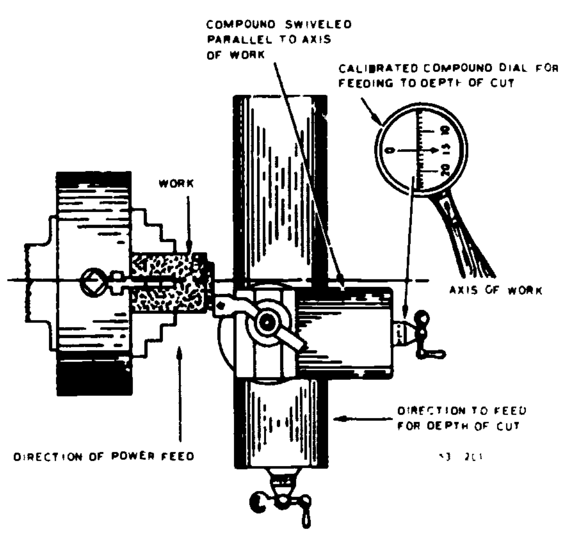


Figure 66. Facing work using the compound rest graduated collar.

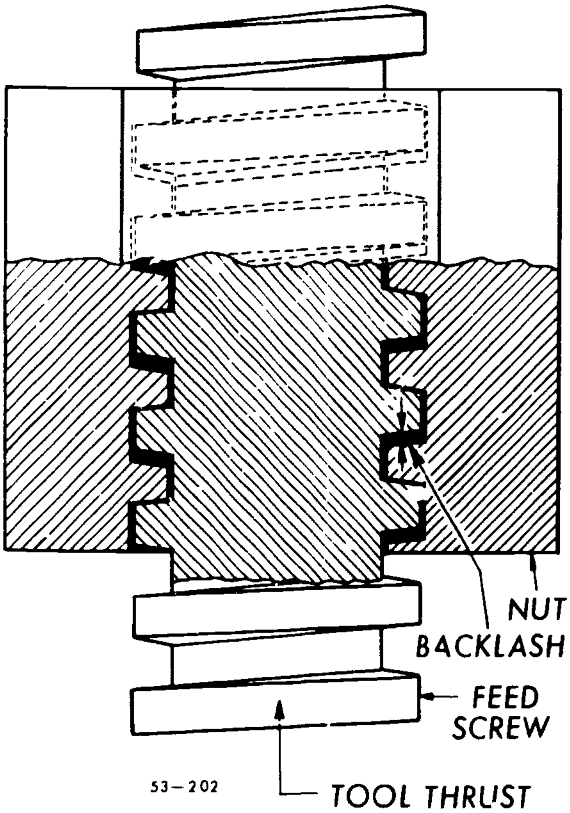


Figure 67. Backlash between feedscrew and nut.

ishing cut. Rough machine the work to within 0.020 inch of the desired diameter, using roughing speeds, feeds, and tools. Replace the roughing tool with a finishing tool, and "pick up" the cut. Pick up the cut by placing a piece of paper between the tool and the work. Feed the tool slowly toward the work until you feel a slight drag on the paper. Allow for the thickness of the paper when you make the required depth of cut. Stop the finishing cut and check the diameter of the work with a micrometer after you have machined ⅛ to ¼ of an inch of the length of the cut. If the diameter is within tolerance, continue the cut to its full length; if not, alternately adjust the depth of cut and make trial cuts until the diameter is within tolerance.

4-13. Use the cross-slide or compound rest graduated collars to set the depth of cut when you wish to remove predetermined amounts of material from the work. The periphery of the collars is graduated to provide a micrometer adjustment for setting the cutting tool.

4-14. The graduations on some makes of lathes represent a tool movement of 0.001 inch, which would reduce the diameter by 0.002 inch. The graduations on other lathes represent a tool movement of 0.05 inch, which would reduce the diameter by 0.001 inch.

4-15. The following example shows how you can use the feed graduated collar to remove a predetermined amount of material from the diameter of the work:

Work diameter	= 1.000 inch
Required diameter	= 0.900 inch
Amount to be removed	= 0.100 inch

If the crossfeed dial is graduated in 0.001 inch, you would move the dial 50 spaces; if it is graduated in 0.0005 inch, you would move the dial 100 spaces.

4-16. Take readings on the graduated collar only in the direction in which the feed screw normally rotates. The lost motion (backlash) between the feed screw and the nut, as shown in figure 67, prevents taking accurate readings when the rotation is reversed. When you must reverse the direction of the feed screw, take up the lost motion by turning the feed screw handle until the movement of the screw causes the cross-slide or compound rest to move. Then turn the feed screw handle in the original desired direction to the required setting.

4-17. *Tooling setup.* For most straight-turning operations, you aline the compound rest parallel to the cross-slide. Then move it to the right 30° and clamp it securely in position. The toolpost should be on the left-hand side of the compound rest T-slot. The danger of running the compound rest into the chuck is reduced when the compound rest and toolpost are in these positions. Position the roughing tool approximately 5° above center height. (This is approximately $\frac{3}{64}$ inch above center per each inch of work diameter.) The finishing tool must be at center height.

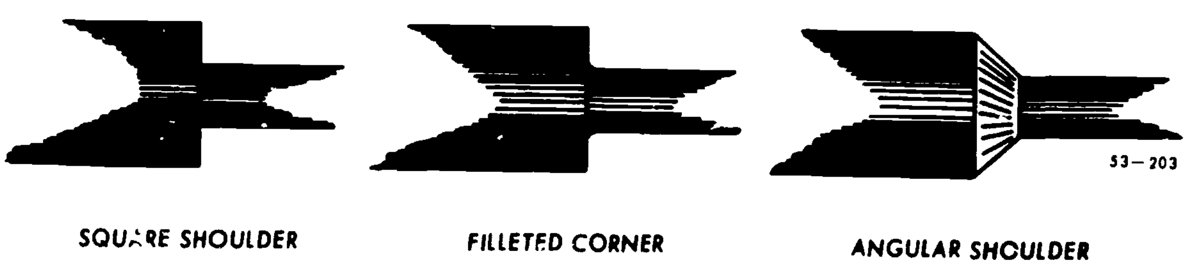


Figure 68. Three common types of shoulders.

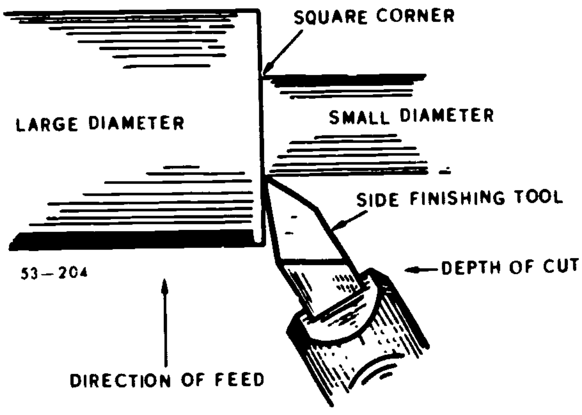


Figure 69 Squaring a shoulder with a side finishing tool.

5. Machining Shoulders, Corners, Undercuts, and Grooves

5-1. Many items which are manufactured on a lathe require the machining of shoulders, corners, undercuts, or grooves. The following information will help you perform these machining operations.

5-2. **Shoulders.** A shoulder is the step or surface which joins two different diameters. Shoulders are turned or formed to various shapes to add strength, make a square corner, fit parts together or improve the appearance of a job. The three common shoulders: square, filleted, and angular, are shown in figure 68.

5-3. *Square* shoulders are used on work that is not subject to excessive strain at the corners. This shape provides a flat clamping surface and permits parts to be fitted squarely together. Usually, you rough out square shoulders with a round nose turning tool and square the shoulders with a side-finishing tool. Figure 69 illustrates the method of feeding the tool to produce a square shoulder.

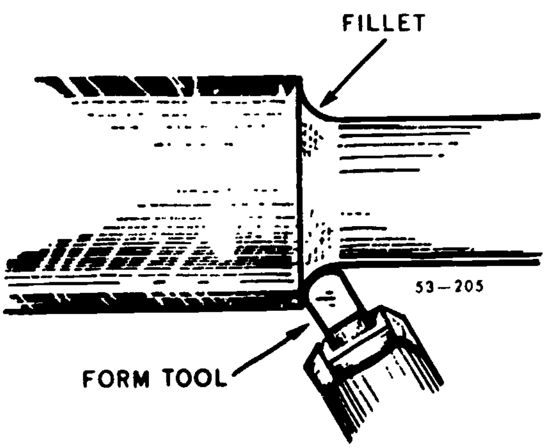


Figure 70 Machining a filleted corner.

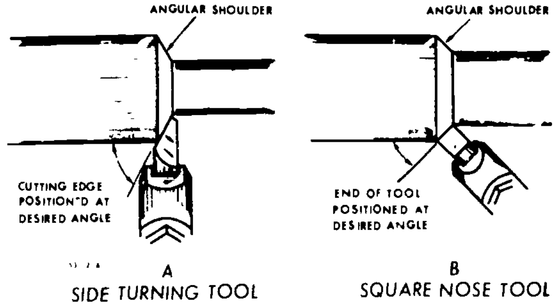


Figure 71. Machining an angular corner.

5-4. To square a shoulder, you should first position the compound rest parallel to the ways of the lathe. Next, position the side-finishing tool as shown in figure 69, making sure that the tool is on center height. Now feed in, picking up the surface of the small diameter with the tool, and set the cross-slide graduated collar at zero. Take a light depth of cut by feeding the tool toward the shoulder with the compound rest. Using either hand or power feed, machine the face of the shoulder by feeding the tool away from the small diameter. If material remains, again move the tool toward the small diameter until the cross-slide collar reads zero, and take another cut on the shoulder. Repeat this operation as required until the shoulder is completely squared.

5-5. *Filleted* shoulders or corners are used on parts which require additional strength at the shoulder. You turn these shoulders with a round-nose tool or form tool ground to the require radius, as shown in figure 70.

5-6. *Angular* shoulders, although not as common as filleted shoulders, are sometimes used to give additional strength to corners, eliminate sharp corners, and improve the appearance of the part. They do not have the strength of filleted shoulders but are more economical to produce. You may turn angular shoulders with the side of the turning tool set at the proper angle, as shown in figure 71,A, or with the end of a square-nose tool, as shown in figure 71,B.

5-7. **Corners.** Corners are turned on the external edges of work in contrast to the shapes which are turned on inner corners at shoulders. You turn corners on work to break down sharp edges and improve the general appearance of the

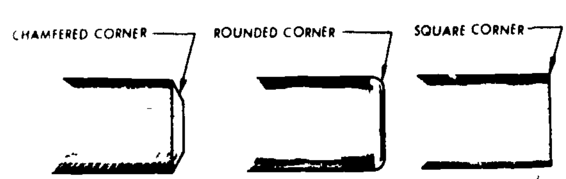


Figure 72. Types of corners.

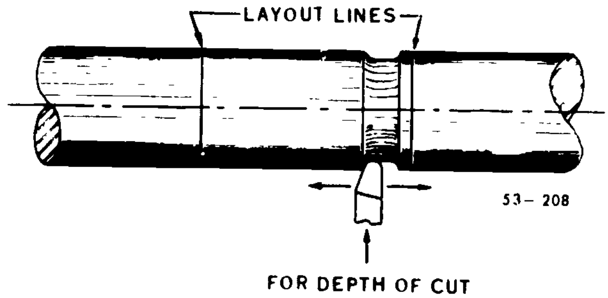


Figure 73. Undercutting a shaft.

job. Figure 72 will enable you to identify chamfered, rounded, and square corners.

5-8. *Chamfered* or *angular* corners are machined with the same tools and setups which are used to machine angular shoulders, as shown in figure 71.

5-9. *Round* corners are produced by turning a small radius on the ends of the work. You may form the radius by freehand manipulation of the cross-slide and carriage, using a turning tool; by using a file to break the sharp corner; or by using a radius form tool.

5-10. *Square* corners are left by facing and straight turning operations. The sharp edge is usually broken down with a file to remove burrs and prevent injury in handling.

5-11. **Undercuts.** An undercut is a reduction in diameter of a portion of the length of a shaft. Undercuts reduce the weight of a shaft and also provide clearance or "runout" for milling cutters, grinding wheels, or other cutting tools. An undercut with filleted shoulders at each end may be produced by using a roundnose turning tool as shown in figure 73. When a large amount of similar work is to be done, a tool without side rake is preferred because it can be fed in either direction. When you wish to turn only an occasional piece, you can easily convert a right-hand turning tool by regrinding the radius on the nose of the tool.

NOTE: While you are taking the depth of cut, move the carriage back and forth slightly to prevent chatter and gouging of the work.

5-12. **Grooves.** Grooves are machined in shafts to provide for tool runout, to allow clearance for assembly of certain parts, to provide a gripping surface, and to provide lubrication channels. Square, round, and "V" grooves and the tools which are used to produce them are shown in figure 74. To cut a round groove of a definite radius on a cylindrical surface, you must use a tool which is ground to fit the proper radius gage. Figure 75 shows how a 5/32-inch radius can be checked with a radius gage. The *grooving tool* is a type of forming tool. It is ground without side rake or back rake, and it 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 higher speeds because of the greater amount of tool contact with the work. Check the depth of the groove or the diameter of the undercut with an outside spring joint caliper or two wires and an outside micrometer, as shown in figure 76.

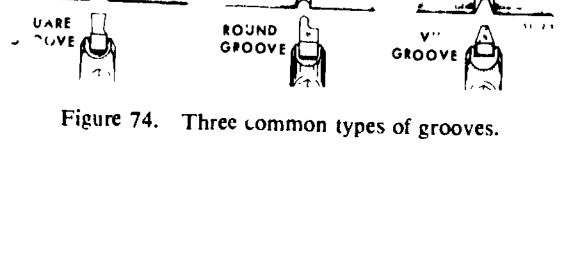


Figure 74. Three common types of grooves.

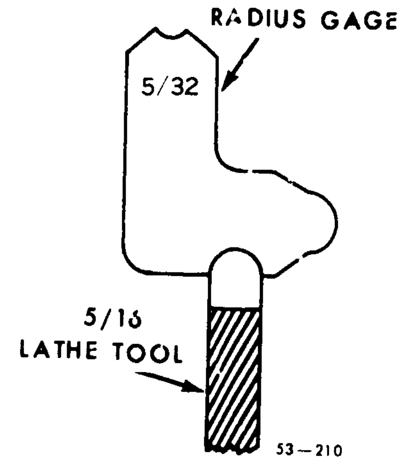


Figure 75 Forming tool ground to fit radius gage.

5-13. **Parting.** 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 special 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 stock, such as tubing, that is impractical to saw with a power hacksaw. Parting is also used to cut off work after other machining operations have been completed. Parting tools can be of the forged type, inserted blade type, or ground from a standard tool blank. In order for the tool to have maximum strength, the length of the cutting portion of the blade should be only slightly greater than half the diameter of the work to be parted.

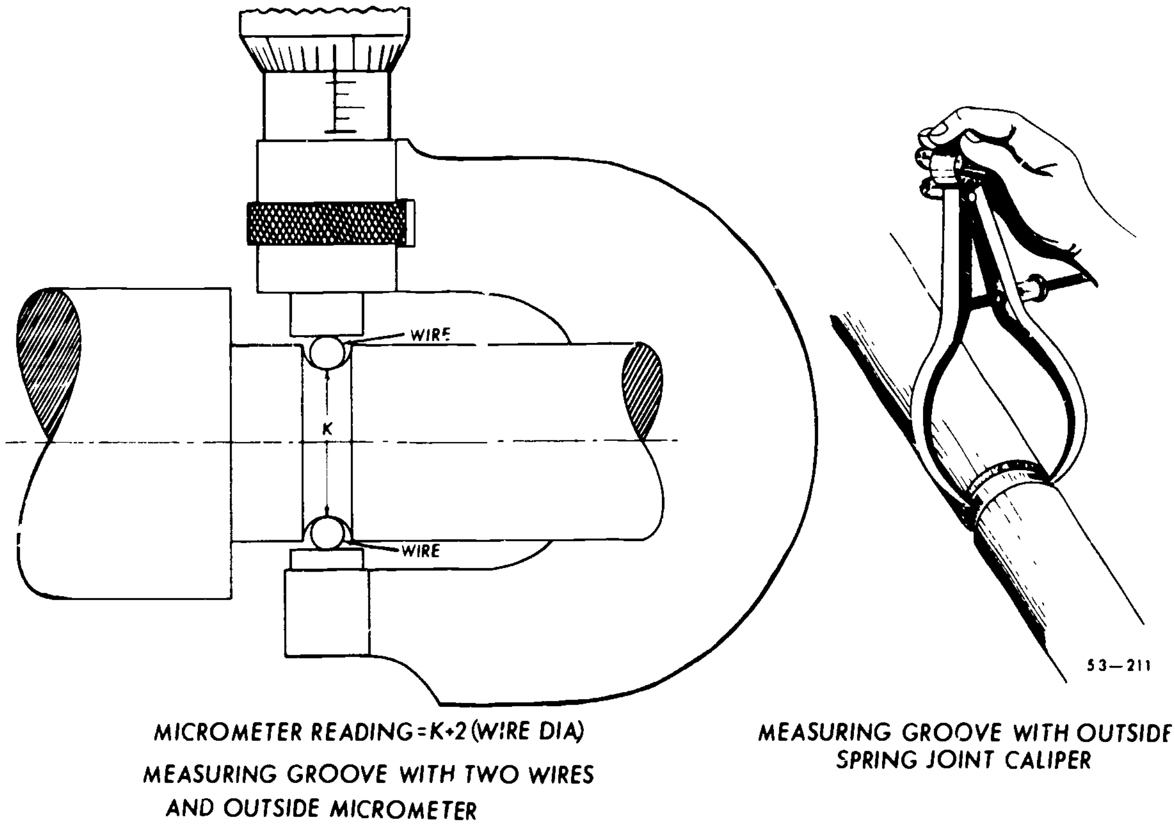


Figure 76. Measuring the depth of a groove.

5-14. Work to be parted should be held in a chuck, with the point at which the parting is to occur as close as possible to the chuck jaws, as shown in figure 77. Always make the parting cut at a right angle to the center line of the work. Feed the tool into the revolving work with the cross-slide until the tool completely severs the work.

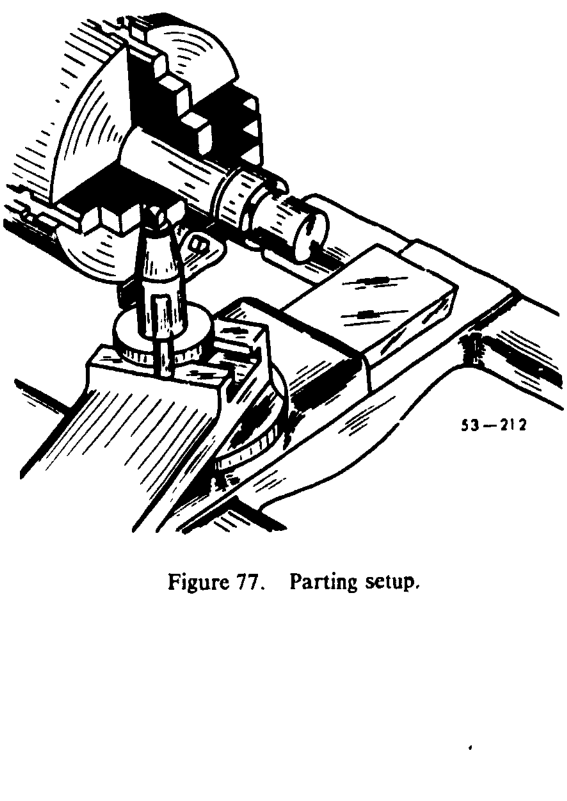


Figure 77. Parting setup.

5-15. Cutting speeds for parting are comparable to turning speeds. You should use a feed that will keep a thin chip coming from the work. If chatter occurs, decrease the speed and increase the feed. If the tool tends to gouge or dig in, decrease the feed. The parting tool should be at center height. It must be square to the work axis to prevent the tool from binding in the cut, as shown in figure 78.

5-16. The length of the portion to be cut off may be measured by placing the edge of a steel rule against the side of the work and the end of the rule against the side of the parting tool, as shown in figure 79. Move the carriage until the desired

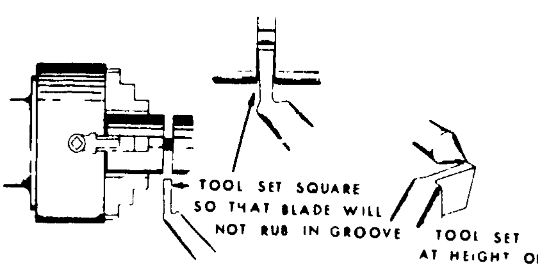


Figure 78. Position of a parting tool.

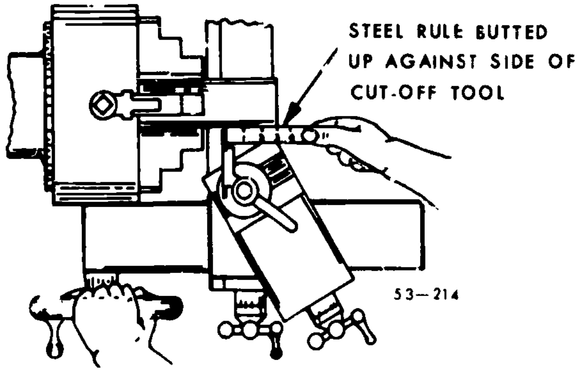


Figure 79. Measuring length to cut off.

length is obtained. You may also aline the parting tool to a layout line scribed on the workpiece.

NOTE: Always lock the carriage in position to prevent it from moving while you are taking the parting cut and use a cutting lubricant.

CAUTION: Never use your hand to catch the piece being parted. It may be hot and have sharp edges.

6. Radii and Form Turning

6-1. You may use several methods to machine radii or irregular shapes. The method will depend upon the shape and size of the object and the number of pieces to be manufactured.

6-2. **Hand Manipulation.** The cutting tool moves on an irregular path when you move the carriage and cross-slide simultaneously by hand. You obtain the desired radius or form by coordinating the moving of the carriage and cross-slide as you observe the cutting action.

6-3. **Form Tool.** You may grind the form tool to any desired shape or form. The only requirements are that the tool must have the proper relief angles and rake, size, and contour. The most practical use of the form tool is in machining several duplicate pieces, since the machining of one or two pieces may not warrant the time spent in grinding the tool. You can use form tools to machine either concave or convex radii. A *concave radius* curves inward and a *convex radius* curves outward. You use a concave tool for a convex radius and a convex tool for a concave radius. Figure 80 shows some typical shapes you can produce with form tools.

6-4. **Template and Pointer.** In this method of form turning, you lay out the full-scale form of the work on a piece of thin sheet metal. Then, clamp the template to the bed of the lathe. Attach a pointer to the lathe cross-slide and, by hand manipulation, follow the scribed outline on the template to produce the form on the work. You will probably have to finish the form by hand filing and polishing. Figure 81 shows a template and pointer being used to produce a contoured surface.

6-5. **Radius Rod.** When you do concave form turning using the radius rod, the length of the rod should be equal to the radius that you want to cut. Place the rod between the cross-slide and tailstock, as shown in figure 82. The cross-slide will then move in an arc when you apply power feed to the cross-slide. The resistance of the cut holds the rod in position.

6-6. **Compound Rest.** When you use this method, the compound rest and tool are swung from side to side in an arc. Form the desired radius by feeding the tool in or out with the compound slide. You can turn either a concave radius by positioning the tool in front of the pivot point, as shown in figure 83,A, or a convex radius by positioning the tool behind the pivot point, as shown in figure 83,B.

6-7. **Radius Attachment.** This attachment may be one of two types. One fits directly on the compound rest and is equipped with a handle to swivel the tool in the desired arc. The other type occupies the place of the cross-slide and compound rest. You can rotate the tool by hand feed or by power if the attachment is geared to the apron of the lathe. Figure 84 shows a hand-operated radius attachment being used to machine a convex radius.

7. Taper Turning

7-1. Many of the tools and parts that you will be using have tapered portions. You must be able to identify the various tapers and be able to machine tapered objects. We will discuss the various standard tapers and their use; the methods of

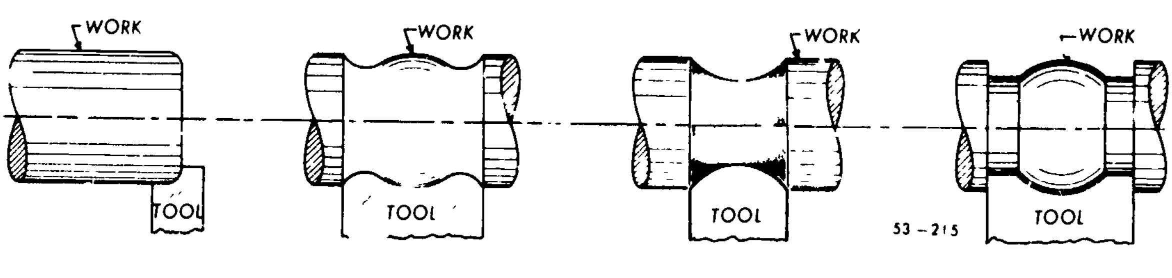


Figure 80. Forming tool.

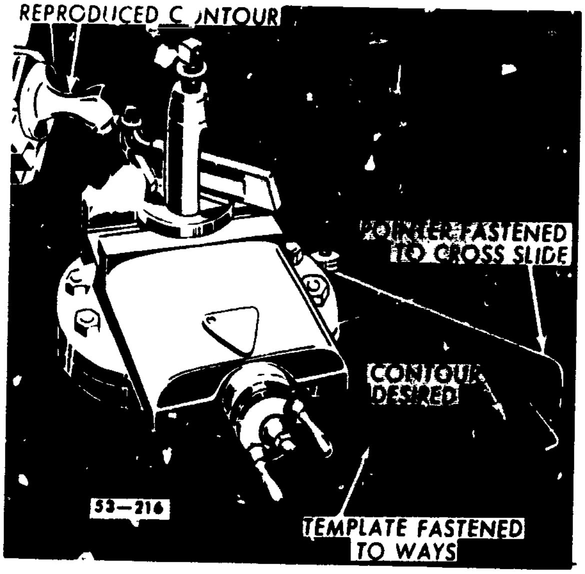


Figure 81. Template and pointer.

checking tapers; and taper turning by means of the compound rest, tailstock, offset, and taper attachment.

7-2. **Standard Tapers.** The tapers on taper-shanked tools and machine parts, such as twist drills, end mills, reamers, lathe centers, drill chucks, etc., are from various standardized taper series. Standard machine tapers are divided into two classes: self-holding tapers and self-releasing tapers.

7-3. *Self-holding (slow) tapers.* The term "self-holding" is applied to the smaller tapers because the angle of the taper is only 2° or 3°. The shank of the tool is so firmly seated in the socket that there is considerable frictional resistance to any torque tending to turn it in the socket. There are several different types of *self-holding tapers.*

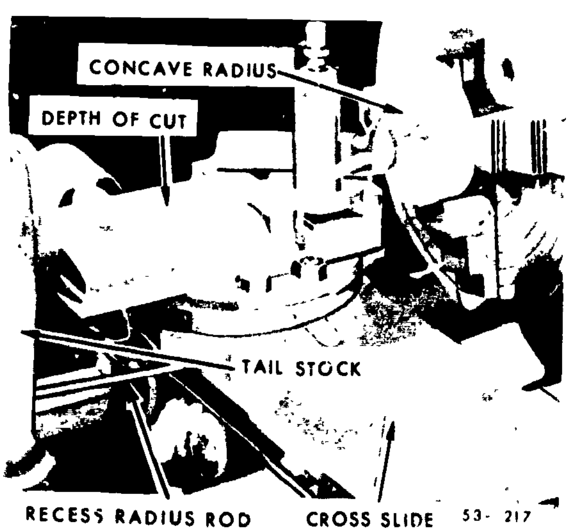


Figure 82. Use of a radius rod.

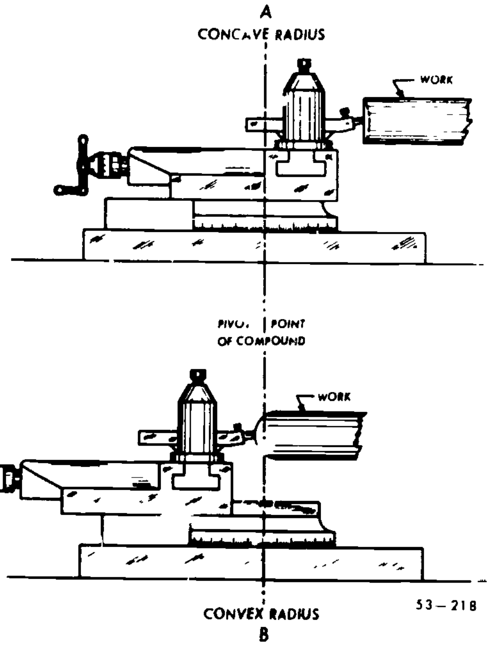


Figure 83. Machining a radius with the compound rest.

a. Morse taper. There are eight different sizes of *Morse tapers.* The taper for each is slightly different, but is approximately ⅝ inch per foot in most cases. Morse taper shanks are used on a variety of tools; they are used exclusively on the shanks of twist drills. Spindles of drilling machines and most lathes are constructed to fit a Morse taper.

b. Brown and Sharpe taper. There are 18 different sizes of Brown and Sharpe tapers. The taper is approximately ½ inch per foot for all sizes except for taper number 10, which has a taper of 0.5161 inch per foot. Brown and Sharpe taper sockets are used for many arbors, collets, and machine tool spindles, and especially for spindles on milling machines and grinding machines.

c. Three-fourths-inch-per-foot taper. These tapers come in 11 sizes ranging from 2 to 12 inches in diameter at the large end. They are larger in size, taking up where the Brown and Sharpe and

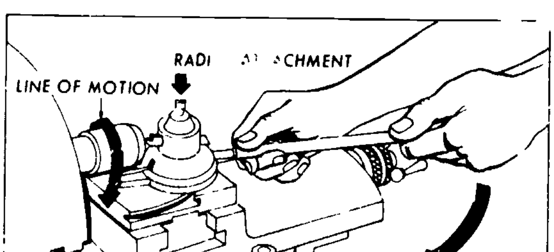


Figure 84. Radius turning attachment.

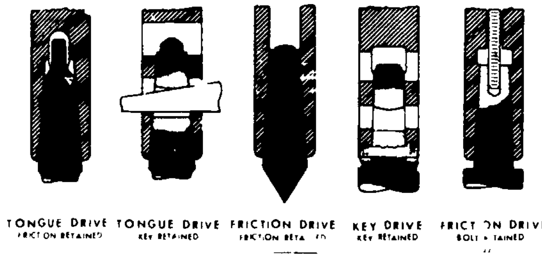


Figure 85. Types of standard taper drives.

Morse tapers stop in the American Standard Self-Holding Taper Series.

d. American Standard Self-Holding Taper Series. Twenty-two taper sizes have been selected to make up the *American Standard Self-Holding Taper Series.* This series contains 3 sizes of the Brown and Sharpe, all 8 sizes of the Morse, and all 11 sizes of the ¾-inch-per-foot taper.

e. Jarno taper. There are 19 different sizes of *Jarno tapers;* the taper per foot on all sizes is 0.600 inch. All the dimensions of any size of Jarno taper may be found by using a simple key based on the taper number. The diameter at the large end is as many eighths, the diameter at the small end is as many tenths, and the length is as many half inches as indicated by the taper number, thus:

$$\frac{\text{Taper number}}{8} = \text{large diameter}$$

$$\frac{\text{Taper number}}{10} = \text{small diameter}$$

$$\frac{\text{Taper number}}{2} = \text{length of taper}$$

For example, a No. 7 Jerno taper has a ⅞-inch large diameter, a $\frac{7}{10}$-inch smaller diameter, and a $\frac{7}{2}$-inch, or 3½-inch, length. The Jarno taper is used on various machine tools, and especially on profiling and die-sinking machines. It has also been used for the headstock and tailstock spindles of some lathes.

f. Taper pins and reamers. Taper pins have a taper of ¼ inch per foot and come in 14 standard sizes. Taper pins are used on assemblies to secure pulleys, gears, and shafts to mating members. Taper pin reamers are used to ream taper pin holes.

g. Other tapers. There are a number of other tapers, but they are used to such a limited extent that full tables are not given in the *Machinery's Handbook.* One, the *Reed taper,* which is used on some lathes, has the same taper as the Jarno taper, 0.600 inch per foot, but it differs in both diameter and length. The Standard Tool Company has two tapers: *standard* and *short.* These tapers vary from 0.600 inch to 0.630 inch per foot. The *Sellers taper* has a 0.750-inch-per-foot taper; it has a keyway the whole length of the taper but no tang.

7-4. *Self-releasing tapers.* The term "self-releasing" is applied to the larger tapers to distinguish them from the relatively small self-holding tapers. A milling machine spindle, with a taper of 3½ inches per foot, is an example of a *self-releasing taper.* The included angle in this case is more than 16° and the tool or arbor always requires a positive locking device to prevent slipping. The shank may be released or removed more readily than the shank of a smaller taper of the self-holding type. There are 12 sizes of American Standard Steep Machine Tapers, all of which have a taper of 3½ inches per foot. Note in figure 85 the various devices that are used to retain and drive standard tapers.

NOTE: Detailed information pertaining to exact dimensions of standard tapers may be obtained from machinists' publications such as the *Machinery's Handbook.*

7-5. **Checking Tapers.** You check tapers for accuracy with protractors, tapered ring gages, or micrometers and scribed lines.

a. Protractors are used to check tapers when extreme accuracy is not required and when the required taper is given in degrees. Figure 86 shows how a protractor head and blade are used to check a steep taper.

b. Tapered ring gages are used to check tapers, as shown in figure 87. Mate the tapered part and the gage and wiggle it. Plug gages are used to check internal tapers. Any movement of the part in the gage indicates that the taper is incorrect.

c. Tapers may be checked by scribing equally spaced lines on the tapered portion of the work and determining the differences in diameters between lines with a micrometer, as shown in figure 88. Careful layout of the lines and alinement of the micrometer will help to insure accurate results when you use this method.

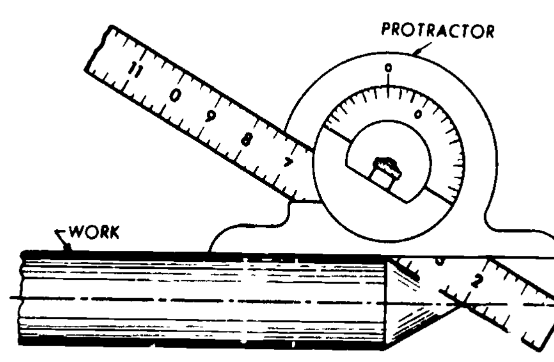


Figure 86. Checking taper with a protractor head.

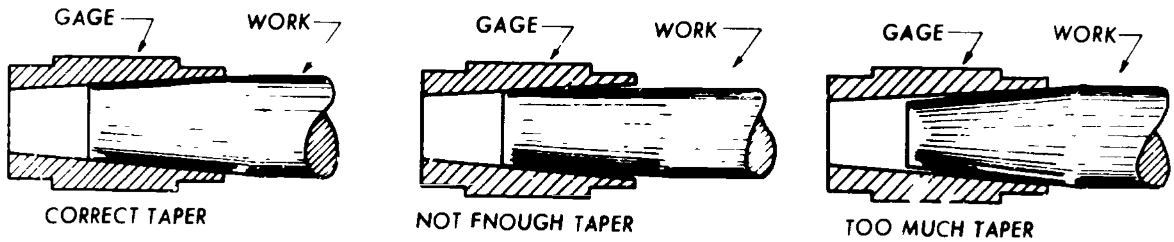


Figure 87. Checking taper with a tapered ring gage.

7-6. **Taper Turning with the Compound Rest.** Both external and internal tapers can be turned with the compound rest. Use the compound rest primarily to machine short, steep tapers, since the length of the taper that can be cut is restricted to the distance the compound can be moved. Position the compound rest at an angle measured from the center line of the work, figure 89,A, or from a line perpendicular to the center line of the work, figure 89,B. For example, the 40° angle in figure 89,B, is measured from a line perpendicular to the center line of the work. In order to machine this angle, you must first position the compound rest perpendicular to the center line, and then move it the required 40°. The graduations on the base of the compound rest swivel represent 1°. You obtain fractions of a degree by estimating the fractional spacing between divisions, as shown in figure 90.

7-7. The amount of taper is often designated as taper per inch (TPI) or taper per foot (TPF). Frequently, no actual designation of the amount of taper is given at all The large diameter (LD), the small diameter (SD), and the length of the taper (L of T) are specified, and you must find the TPI before you can set the compound rest properly. To determine the angle at which the compound rest should be set, use the following formulas:

$$\text{TPI} = \frac{\text{LD} - \text{SD}}{\text{L of T}}$$

$$\text{Tangent of the angle } (\tan \angle) = \frac{\text{TPI}}{2}$$

Let us suppose that a TPI of 0.800 is specified.

$$\text{Tan } \angle = \frac{\text{TPI}}{2} = \frac{0.800}{2} = 0.400$$

It would now be possible to compute the angle at which to set the compound rest; however, it is more convenient to obtain it from table of trigonometric functions (often called *trig tables).* This table may be found in machinists' publications, such as the *Machinery's Handbook,* and in trigonometry handbooks. To determine the size of the angle, go down the "tangent" column until you find the tangent of the angle. The nearest value is 0.39997. In the "M" column at the left side of the page opposite 0.39997 you will read 48 minutes. In the upper left corner of the page, above the minutes column, you will read 21° in boldface nu-

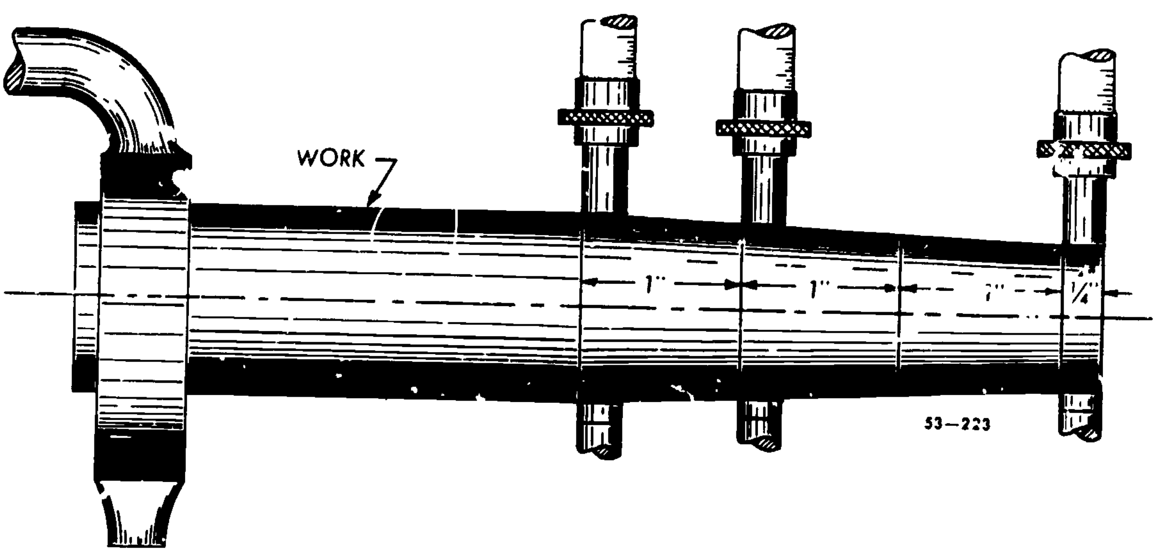


Figure 88. Measuring taper per inch with a micrometer.

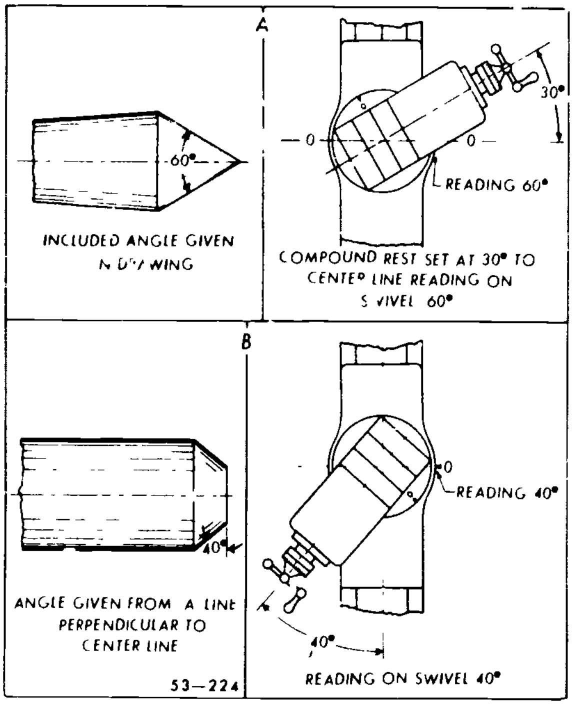


Figure 89. Setting the compound rest.

metals. Thus, 0.39997 is the tangent for 21° 48′. You would set the compound rest for 21¾° to machine a taper per inch of 0.800. (If TPF had been specified, it would have been necessary to convert it into TPI by dividing TPF by 12.)

7-8. If the dimensions of a taper are given, as in figure 91, the computation would be as follows:

$$\text{TPI} = \frac{\text{LD} - \text{SD}}{\text{L of T}} = \frac{1.250 - 1.000}{0.500} = 0.500$$

$$\text{Tan } \angle = \frac{\text{TPI}}{2} = \frac{0.500}{2} = 0.2500$$

which is the tangent for 14° 2′.

7-9. **Taper Turning by Offsetting the Tailstock.** You use the *offset tailstock method* to turn long slow tapers because of the limited travel of the compound rest. The main reason for using this method is that not every lathe is equipped with a taper attachment. When you offset the dead center by moving the tailstock out of alinement, the center line of the work and the line of travel of the turning tool are no longer parallel, and a taper will be turned on the work, as shown in figure 92. Offset the tailstock, *after it has been unclamped,* by turning the tailstock lateral adjustment screws. Position the cutting tool at center height in order to turn a true taper.

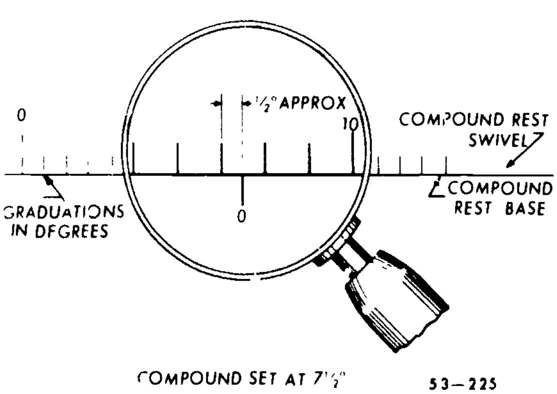


Figure 90. Determining fractional setting of compound rest.

7-10. *Calculating offset.* When the taper per inch and the length of work are given, you can determine the amount of offset required to cut a taper by using the following formula:

$$\text{Tailstock offset (TO)} = \frac{\text{taper}}{2} \times \text{length of work}$$

Example: To cut a taper having 0.050 TPI, 5 inches in length on a piece of work 10 inches long, as shown in figure 93, the tailstock offset would be calculated as follows:

$$\text{TO} = \frac{\text{TPI}}{2} \times \text{LW}$$

$$\text{TO} = \frac{0.050}{2} \times 10 = 0.025 \times 10$$

$$\text{TO} = 0.250 = \tfrac{1}{4} \text{ inch}$$

The amount of offset required to cut a taper, when the included angle and the length of work are given, may be determined in the following manner:

(1) First divide the included angle by 2 to determine the angle you must machine:

$$\text{Machine angle} = \frac{\text{included angle}}{2}$$

(2) Now, determine the offset by using the following formula:

Tailstock offset = tangent of angle × length of work
$\text{TO} = \tan \angle \times \text{LW}$

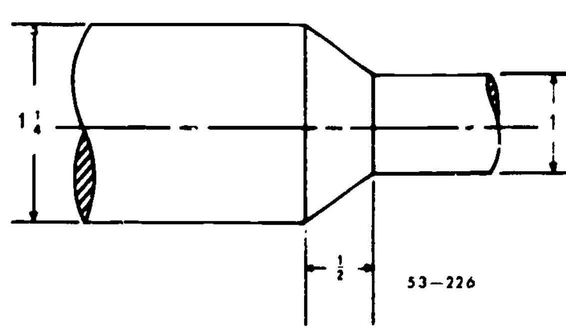


Figure 91. Dimensions of a taper.

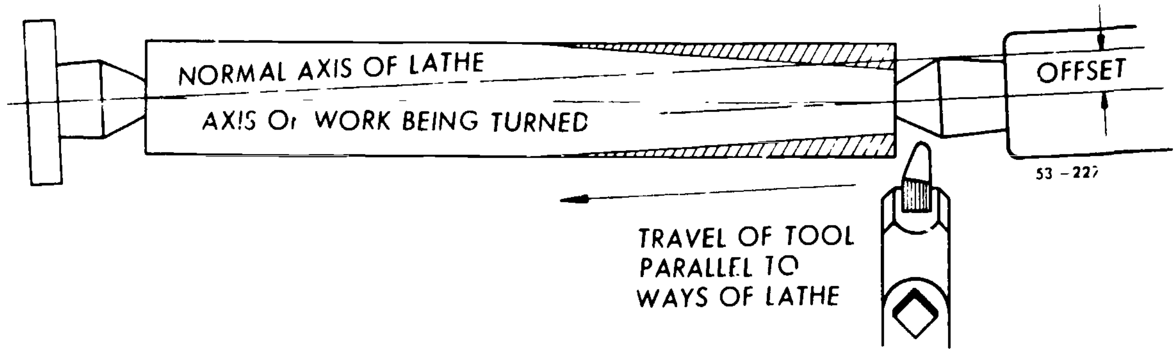


Figure 92. Effect of tailstock offset.

Example: If you wish to cut a taper with an included angle of 7° on a piece of work 12 inches long, as shown in figure 94, the calculations would be as follows:

(1) Angle $= \frac{7°}{2}$

Angle $= 3\frac{1}{2}°$

(2) $TO = \tan 3\frac{1}{2}° \times 12$

$TO = 0.6116 \times 12$

$TO = 0.7339 = \frac{47}{64}$ inch

NOTE: You should note that the length of taper was not taken into consideration in the foregoing formulas when you calculate the tailstock offset. You use only the overall length of the work and the TPI or the overall length of the work and the included angle of the taper.

7-11. When you determine the proper offset for the tailstock, remember that any change in the length of work between centers requires resetting the tailstock if the taper per inch is to remain constant. A given offset does not give a fixed degree of taper, because the taper increases as the length of work decreases, as shown in figure 95.

7-12. *Measuring offset.* You may measure the amount of the tailstock offset by various methods, depending upon the nature of the job. On work that does not require a great degree of accuracy, you may measure the amount of offset by reading the graduations (called *cricket marks*) on the base of the tailstock, as shown in figure 96. Another method is to measure the lateral distance between centers with a machinist's rule, as shown in figure 97. On work requiring a great deal of accuracy, you may measure the amount of offset with an inside caliper, with a dial indicator, by using the crossfeed calibrated collar, or by using the cut and try method.

7-13. When you check the offset with an inside caliper, bring the toolpost to bear lightly against the side of the tailstock spindle, as in figure 98,A, and then back it away from the tailstock spindle a distance equal to the predetermined caliper setting, figure 98,B. Then offset the tailstock until the tailstock spindle again contacts the toolpost, as shown in figure 98,C.

7-14. When you use the dial indicator method, first mount the instrument on the toolpost. Then position the indicator plunger to bear lightly against the side of the tailstock spindle. Then read the amount of offset on the indicator as you move the tailstock laterally, as shown in figure 99.

7-15. When you use the crossfeed (cross-slide) calibrated collar method, position the side of the toolpost near the tailstock spindle and then move the toolpost *away* from the tailstock spindle with the cross-slide, as shown in figure 100,A. This will eliminate the backlash. Then set the cross-slide graduated collar at zero and move the toolpost toward the tailstock spindle with the compound rest until you feel a slight drag on a strip of paper be-

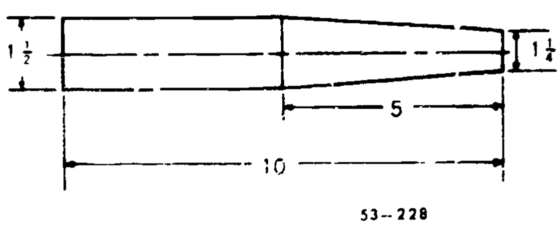


Figure 93. Calculating tailstock offset when the dimensions are given.

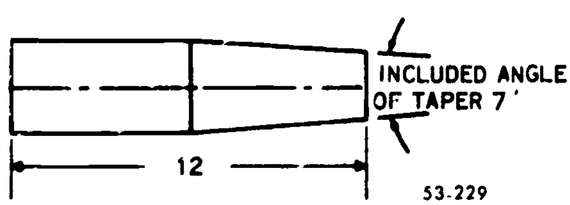


Figure 94. Calculating tailstock offset when the included angle is given.

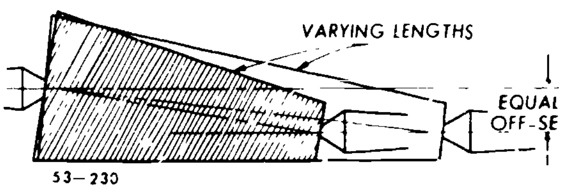


Figure 95. Effect of work length on taper.

tween the tailstock spindle and the toolpost when you pull on the paper, as shown in figure 100,B. Next, using the cross-slide graduated collar to indicate the amount of travel, move the toolpost away from the tailstock spindle a distance equal to the desired offset. Thereafter, move the tailstock laterally until you feel a slight drag on a strip of paper between the tailstock spindle and the tool post when you pull on the paper, as shown in figure 100,C. At this point, you have obtained the desired offset.

7-16. In the cut and try method, you offset the tailstock an approximate amount and take a trial cut on the work. Then measure the taper per inch, readjust the taiistock, and take more trial cuts until you obtain the desired taper per inch. The cut and try method is most accurate because the required taper is actually produced on the work.

7-17. While the off̶set method of taper turning is widely used, it has a number of disadvantages. A slight variation in the length of the work or the depth of center holes will result in a variation in the amount of taper when duplicate pieces are being turned. The center holes of the work do not bear uniformly on the lathe centers; as a result, the center holes may be distorted, and the centers may be scor‑ ' Also, you must realine the centers for straight turning after the tailstock has been offset for taper turning. The offset method is limited to work held between centers; therefore, only external tapers may be turned. The degree of taper that can be turned is governed by the maximum

Figure 96. Tailstock cricket marks.

amount of tailstock offset and the length of the work. The range of offset of the tailstock varies from approximately ½ inch on small lathes to 1½ inches on larger lathes.

7-18. **Taper Turning with Taper Attachments.** The taper attachment is used to machine both internal and external tapers. The length of travel is considerably greater than that of the compound rest; however, the angle to which it may be set is limited. For several reasons, it is more desirable to machine a taper with the taper attachment than with the offset tailstock method. The lathe centers are in alinement; the wear on the centers and center holes is not as great. Most important, duplicate tapers may be machined on pieces of different lengths without changing the taper setting. Also mating internal and external tapers can be machined without changing the taper setting.

7-19. *Description.* The essential parts of the taper attachment shown in figure 101 and the purpose each serves are as follows:

(1) The carriage bracket (A) is attached to the saddle of the lathe and supports the attachment.

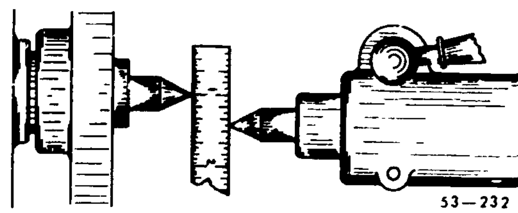


Figure 97. Measuring offset with a steel rule.

(2) The guide bar or swivel (B) acts as a guide for the guide block. It is swiveled and set to produce the desired amount of taper.

(3) The shoe or guide block (C) travels on the guide bar and is ; ‑hed to the cross-slide.

(4) The guide block base (D) supports the guide bar.

(5) The clamping screws (E) clamp the guide bar to its base.

(6) The bed bracket (F) clamps the guide bar to the lathe bed.

(7) The guide block gibs (not shown) take up the wear between the guide block and the guide bar.

(8) The draw bar (G) connects the cross-slide to the guide block by means of a clamp (H) and relieves the push and pull on the crossfeed screw.

7-20. *Operation.* Taper attachments vary in design with different manufacturers; however, the operation of all of them is basically the same. Swivel the guide bar to the ‑ired degree of taper and clamp it in position. As the carriage travels

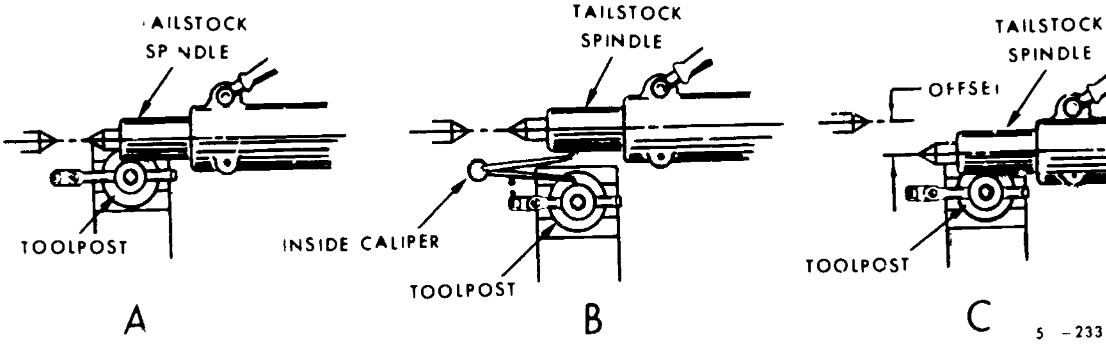


Figure 98. Measuring offset with inside calipers.

along the ways, the guide block will slide on the guide bar, causing the draw bar to push the cross-slide toward, or pull it away from, the work. This, in turn, will cause the tool to move in a plane parallel to the guide bar, and a taper will be machined on the work.

7-21. On taper attachments that are not equipped with a draw bar, the push or pull is directly on the crossfeed screw, and there will be a certain amount of lost motion or backlash between the crossfeed screw and its nut. If backlash is not eliminated, a straight portion will be turned on the work. The backlash may be eliminated by one of two methods, as follows:

(1) Move the carriage and tool slightly past the start of the cut, then return the carriage and tool to the start of the cut.

(2) Move the cross-slide and the tool in the same direction in which the taper runs; that is, make the last movement of the cross-slide away from you when the small end of the taper is toward the headstock, or toward you when the large end of the taper is toward the headstock. Any further movement of the carriage, other than in the direction of the cut, will automatically put backlash in the system.

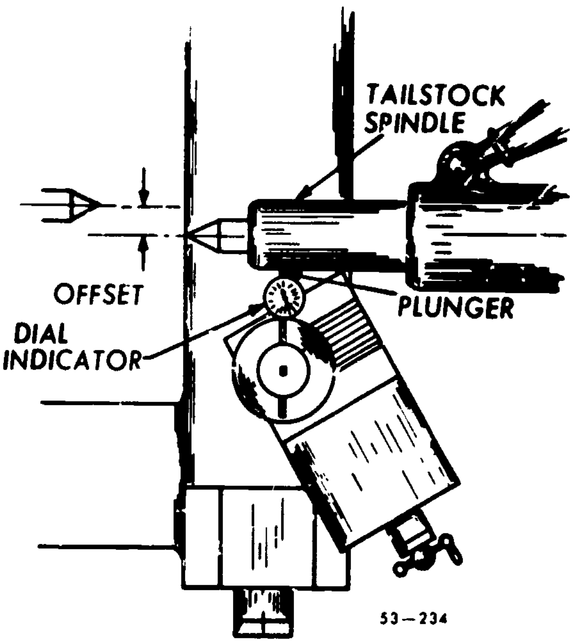


Figure 99. Measuring offset with a dial indicator.

7-22. Taper attachments equipped with a draw bar do not have backlash. The draw bar connects the cross-slide directly to the guide block and takes the push and pull off the crossfeed screw and its nut. You set the compound rest at a right angle to the ways and take the depth of cut with the compound rest. This is necessary since the draw bar locks the cross-slide directly to the guide block, and the cross-slide cannot be moved with the crossfeed screw. The guide bar, which is swiveled and set to give the desired taper, is graduated in taper per foot on one end and degrees of taper on the opposite end, as shown in figure 102. To set the guide bar for the proper taper when the taper per inch is given, multiply the TPI by 12 to get the taper per foot and set the graduated scale to the nearest fractional setting. Table 3 will help you to change the decimal to the equivalent fraction.

7-23. Let us assume that you are setting up a taper attachment prior to machining a drill sleeve for which a number 2 Morse external taper is specified. What setting would you use for the first trial cuts if you had decided to use the taper-per-foot graduations? The degree graduations? You can determine the taper-per-foot setting in the following manner:

(1) Find the taper per inch of a number 2 Morse taper in the machinist publication, such as the *Machinery's Handbook*. (TPI = 0.04995)

(2) Convert the taper per inch into taper per foot.

$$TPF = TPI \times 12$$
$$TPF = 0.04995 \times 12$$
$$TPF = 0.5994$$

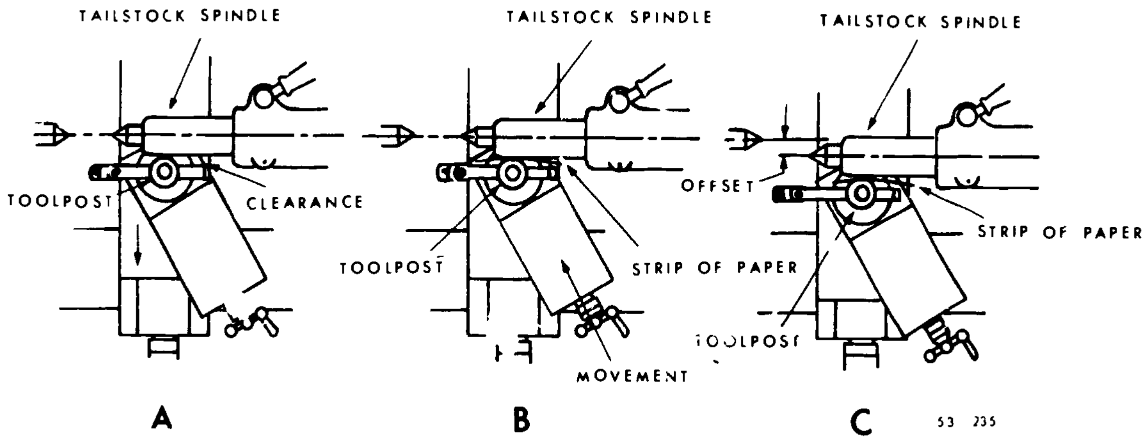


Figure 100. Measuring offset with the cross-slide graduated collar.

(3) Convert the decimal to a fraction, using a decimal equivalent table such as the one in table 3.

$$0.5994 = \frac{19}{32}\ (0.5937)$$

(4) Set the taper attachment to obtain a 19/32 per foot taper.

NOTE: Taper attachments are often graduated in ⅛-inch-per-foot increments; therefore, you may have to estimate settings that fall between the ⅛-inch graduations. You can obtain sufficient accuracy for the first trial cuts by setting the taper attachment for a taper of slightly less than ⅝ inch per foot.

7-24. If you decided to use the degree graduations, you would determine the degree setting in the following manner:

(1) Find the tangent of the angle. (Tan $= \frac{TPI}{2}$)

$$\text{Tan} = \frac{0.04995}{2}$$

$$\text{Tan}\ 0.02497$$

(2) Look in a table of trigonometric functions and find the angle represented by the tangent you have calculated.

$$\text{Tan}\ 0.02497 = 1°25'$$
or
approximately 1½°

NOTE: The degree graduations on most taper attachments represent the *included* angle of the taper. Therefore, you would set the taper attachment to indicate *twice* the angle you have calculated, or 3°.

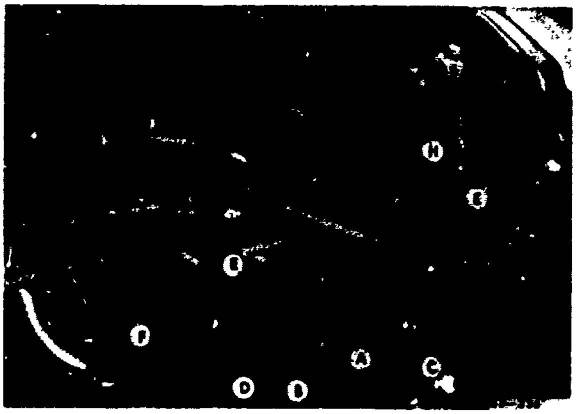


Figure 101. Taper attachment.

8. Drilling, Boring, Reaming, and Recessing

8-1. You will frequently manufacture items in which holes must be drilled, bored, reamed, or recessed. These are common lathe operations. The information in this section will help you to perform these operations.

8-2. **Drilling.** You will often drill holes before you perform other internal operations. There are several methods of holding the drill. It may be held in a drill chuck, held in the tailstock spindle,

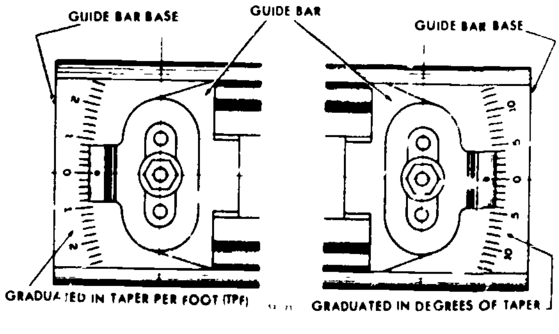


Figure 102. Guide bar graduations.

TABLE 3
DECIMAL EQUIVALENTS

DECIMAL EQUIVALENTS									
Fraction	16th	32nd	64th	Decimal	Fraction	16th	32nd	64th	Decimal
			1	015625				33	515625
		1	2	03125			17	34	53125
			3	046875				35	546875
	1	2	4	0625		9	18	36	5625
			5	078125				37	578125
		3	6	09375			19	38	59375
			7	109375				39	609375
1/8	2	4	8	125	5/8	10	20	40	625
			9	140625				41	640625
		5	10	15625			21	42	65625
			11	171875				43	671875
	3	6	12	1875		11	22	44	6875
			13	203125				45	703125
		7	14	21875			23	46	71875
			15	234375				47	734375
1/4	4	8	16	250	3/4	12	24	48	750
			17	265625				49	765625
		9	18	28125			25	50	78125
			19	296875				51	796875
	5	10	20	3125		13	26	52	8125
			[illegible]	328125				53	828125
			[illegible]	34375			27	54	84375
			23	359375				55	859375
3/8	6	12	24	375	7/8	14	28	56	875
			25	390625				57	890625
		13	26	40625			29	58	90625
			27	421875				59	921875
	7	14	28	4375		15	30	60	9375
			29	453125				61	953125
		15	30	46875			31	62	96875
			31	484375				63	984375
1/2	8	16	32	500	1	16	32	64	1 000

floated on the dead center, or mounted in the headstock spindle.

8-3. Usually, the drill is held in a drill chuck that is mounted in the tailstock spindle. The drill is fed into the work by means of the tailstock handwheel, as shown in figure 103.

8-4. Taper-shank drills may be held directly in the tapered bore of the tailstock spindle. When the tapers are unlike, use a socket reducer of the proper size to fit the shank of the drill to the tailstock spindle. Attach a lathe dog to the neck of large drills and allow the tail of the dog to rest on the compound rest. The dog prevents the drill from turning in the tailstock spindle bore, as shown in figure 104. Before inserting any taper shank tool into a taper bore, you should clean both

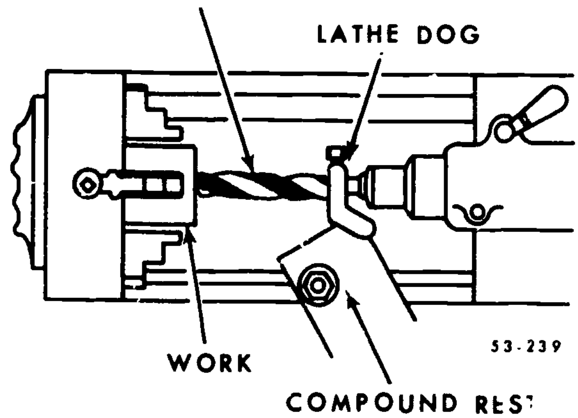


Figure 104. Supporting a drill in the tailstock spindle.

the shank and bore. This will prevent damage to their finished surfaces and permit a tighter fit between them.

8-5. A straight or taper-shank drill of large diameter may be supported in a drillholder, which is floated on the tailstock center, as shown in figure 105. Also the drill may be floated directly on the tailstock center, as shown in figure 106. When the drill is floated on dead center, attach a lathe dog to it or allow the arm of the drillholder to rest on the

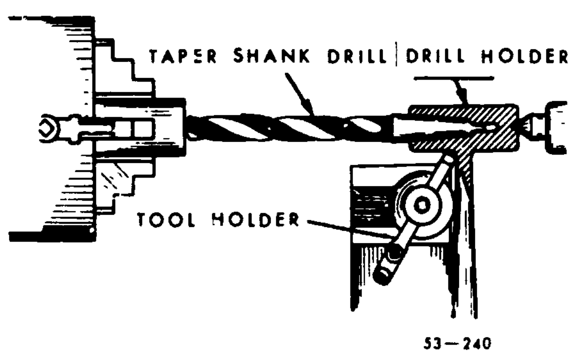


Figure 105. Drill holder floated on the dead center.

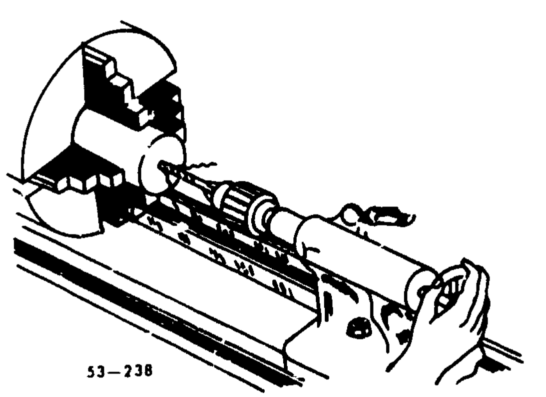


Figure 103. Feeding a drill into the work.

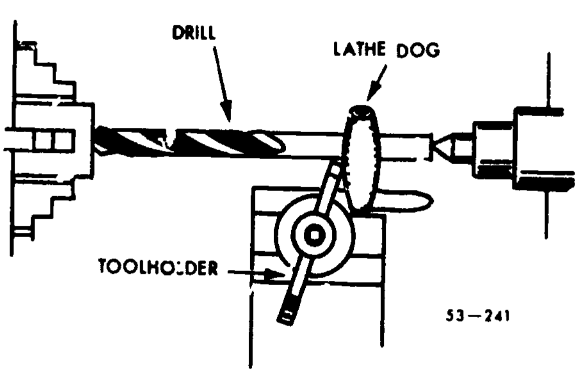


Figure 106. Drill floated on the dead center.

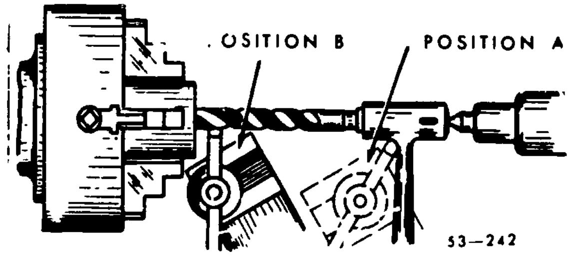


Figure 107. Steadying a drill.

compound rest. The lathe dog or the drillholder arm prevents the drill from turning because of the force of the cut. There is also a tendency for the drill to pull into the work. You can prevent this by clamping a toolholder in the toolpost with the butt end of the toolholder against the side of the dog or drillholder, as shown in figures 105 and 106. The lathe dog or drillholder arm will push the carriage forward as you feed the drill into the work. The carriage will offer enough resistance to keep the drill from digging into the work, and it will also hold the drill on the dead center as it breaks through the work at the completion of the cut.

CAUTION: The work must not be turning when you position the drill *Never,* under any circumstances, withdraw the drill from the drilled hole without first stopping the lathe spindle.

8-6 If the drill wobbles at the start of the cut, it may be supported near the point with a steadying tool until the hole is started true. Change the position of the toolholder so that the butt end serves as a steadying tool, as shown in figure 107, position B. After the drill has been started true, return the toolholder to its original position, figure 107, position A, to hold the drill back against the center.

8-7. You regulate the depth of the hole by using the graduations on the tailstock spindle, as shown in figure 108,A. You can also regulate it by scribing a pencil line on the tailstock spindle and measuring the distance the spindle travels with a

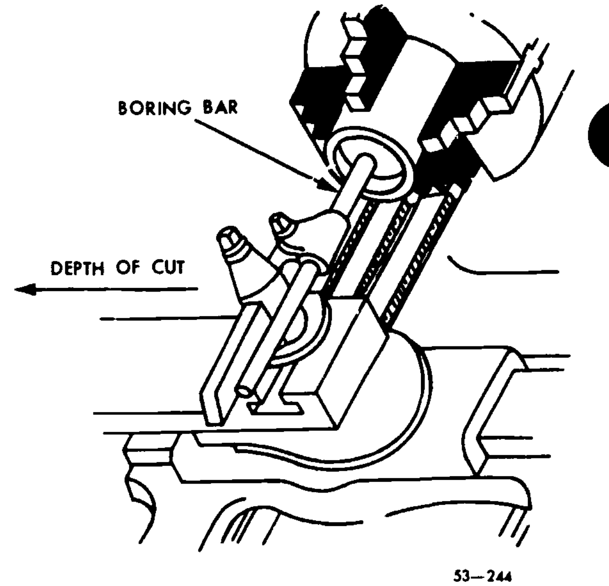


Figure 109. Boring in the lathe.

steel rule, as shown in figure 108,B. Also, a pencil line or chalk mark maybe made on the drill to regulate the depth.

8-8. Certain jobs may require that the drill be mounted in the headstock spindle. The work is then supported on a drill pad center or a crotch center mounted in the tailstock spindle. The work is then fed into the drill by turning the tailstock handwheel.

8-9. **Boring.** Boring is the process of enlarging and truing a drilled or cored hole with a boring tool. The boring tool is mounted in the lathe toolpost. As you feed the tool longitudinally into the revolving work, the cutting edge of the tool enlarges the hole, as shown in figure 109. Boring is one of the more dependable methods of machining holes which are round and concentric. Boring can be used to true holes, so that in following reaming operations, the reamer will follow the true hole. Another advantage is that you can finish bore holes to any given size and leave the proper allowance for

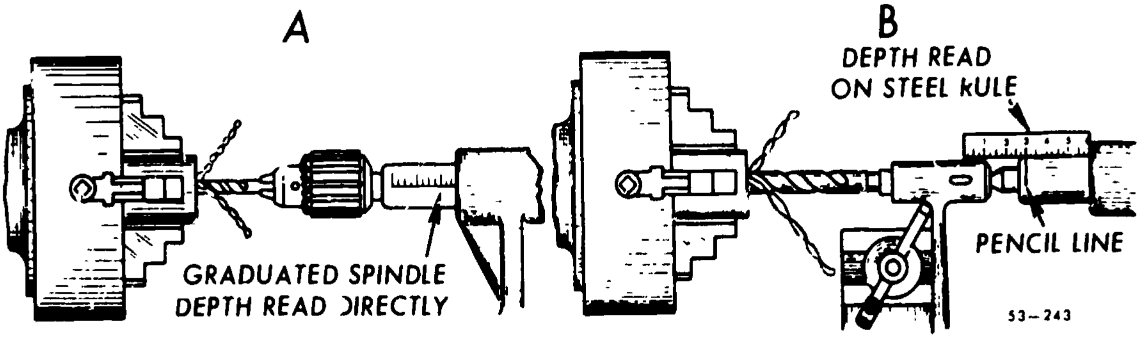


Figure 108. Measuring drill depth.

Table 4
Drill Sizes

Size	Decimal Equivalent	Size	Decimal Equivalent	Size	Decimal Equivalent
1/2	0.500	3	0.213	3/32	0.0938
31/64	0.4844	4	0.209	42	0.0935
15/32	0.4688	5	0.2055	43	0.089
29/64	0.4531	6	0.204	44	0.086
7/16	0.4375	13/64	0.2031	45	0.082
27/64	0.4219	7	0.201	46	0.081
Z	0.413	8	0.199	47	0.0785
13/32	0.4062	9	0.196	5/64	0.0781
Y	0.404	10	0.1935	48	0.076
X	0.397	11	0.191	49	0.073
25/64	0.3906	12	0.189	50	0.070
W	0.386	3/16	0.1875	51	0.067
V	0.377	13	0.185	52	0.0635
3/8	0.375	14	0.182	1/16	0.0625
U	0.368	15	0.180	53	0.0595
23/64	0.3594	16	0.177	54	0.055
T	0.358	17	0.173	55	0.052
S	0.348	11/64	0.1719	3/64	0.0469
11/32	0.3438	18	0.1695	56	0.0465
R	0.339	19	0.166	57	0.043
Q	0.332	20	0.161	58	0.042
21/64	0.3281	21	0.159	59	0.041
P	0.323	22	0.157	60	0.040
O	0.316	5/32	0.1562	61	0.039
5/16	0.3125	23	0.154	62	0.038
N	0.302	24	0.152	63	0.037
19/64	0.2969	25	0.1495	64	0.036
M	0.295	26	0.147	65	0.035
L	0.290	27	0.144	66	0.033
9/32	0.2812	9/64	0.1406	67	0.032
K	0.281	28	0.1405	1/32	0.0312
J	0.277	29	0.136	68	0.031
I	0.272	30	0.1285	69	0.0292
H	0.266	1/8	0.125	70	0.028
17/64	0.2656	31	0.120	71	0.026
G	0.261	32	0.116	72	0.025
F	0.257	33	0.113	73	0.024
E-1/4	0.250	34	0.111	74	0.0225
D	0.246	35	0.110	75	0.021
C	0.242	7/64	0.1094	76	0.020
B	0.238	36	0.1065	77	0.018
15/64	0.2344	37	0.104	1/64	0.0156
A	0.234	38	0.1015	78	0.016
1	0.228	39	0.0995	79	0.0145
2	0.221	40	0.098	80	0.0135
7/32	0.2188	41	0.096	--	------

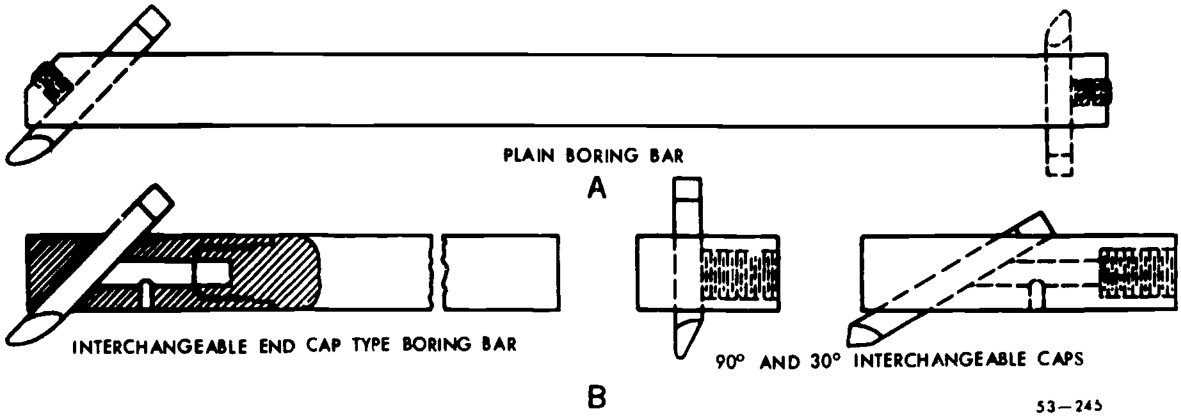


Figure 110. Types of boring bars.

reaming. Holes of odd sizes must always be bored to size when reamers or special sizing tools are not available.

8-10. Boring is usually preceded by drilling unless the work is already provided with a hole. The drilled hole should be large enough to permit the boring bar to enter and, at the same time, sufficient material should be left to permit machining the hole true to size. Generally, the drilled hole diameter is $\frac{1}{32}$ inch smaller than the diameter of the bored hole. Table 4 will help you to select the proper drill. Always use the largest size boring bar that will clear the hole. This insures greater rigidity and will decrease the tendency of the tool to chatter.

8-11. *Speed and feed*. The speed for boring a hole is the same speed you would use if you were straight turning work of the same external diameter. Hole size is used for calculating rpm as opposed to work diameter used in turning operations.

8-12. Use feeds for boring which are considerably slower than those you use for straight turning, because there is less rigidity in a boring setup. Decrease the depth of cut for the same reason.

8-13. *Boring tools and holders*. Boring tools, shown in figure 110, are available in a variety of types. Some of these tools are forged or ground from a solid bar, whereas others are high-speed tools which are held in boring bars and holders.

8-14. A *solid boring tool* may be ground from an ordinary lathe tool blank, as shown in figure 110,A; or it may be a tool steel rod that has been bent and ground, as shown in figure 110,B. The round type can be placed between two small "V" blocks and clamped in the toolpost of the lathe. Boring tools ground from high-speed tool blanks are especially handy for boring small diameter holes.

8-15. A boring tool that is held in a *boring bar* which in turn is held in a boring bar holder, can be adjusted for overhang. You can extend or retract the boring bar in the boring bar holder to suit the length of the hole, thus providing the greatest rigidity for a given hole. Another advantage of the boring bar is that you can easily remove the tool for sharpening or replace it without disturbing the position of the holder. The boring bar consists of a cylindrical steel bar with a square hole broached at either end or both ends to receive a cutting tool and a setscrew or clamping device to lock the tool in position. Ordinarily, these tools are held in the bar at a 30°, 45°, or 90° angle. Two types of bars are commonly used: the plain solid bar, figure

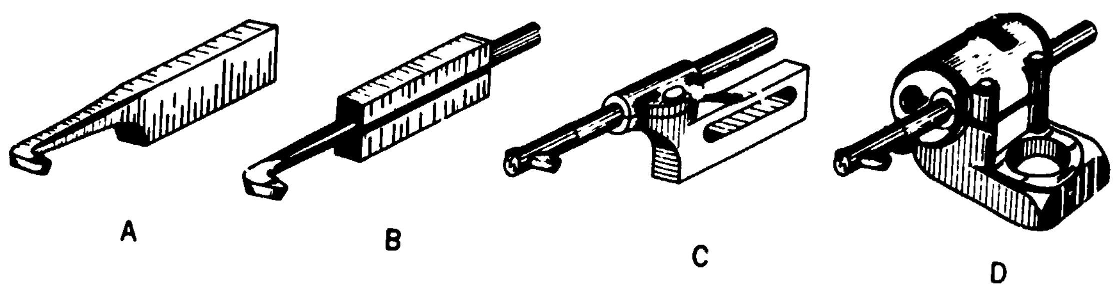


Figure 111. Types of boring tools.

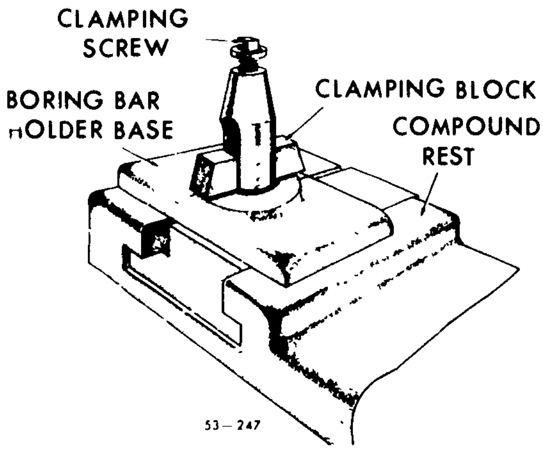


Figure 112. Use of the clamping block.

110,A, and the threaded cap type, figure 110,B. On the *plain bar type*, the tool is held by means of a setscrew. On the *end cap type*, the tool is locked in position by the wedging action of a hardened plug against the tool when the cap is tightened on the bar.

8-16. The boring bar is held in a boring bar holder, as shown in figure 111,C and D. The type of boring bar holder shown in figure 111,C, is held in the toolpost in the same manner as an ordinary toolholder. The type of boring bar holder shown in figure 111,D, is placed over the toolpost after the toolpost rocker and ring have been removed. A block of metal is then placed in the toolpost between the clamping screw and the base of the boring bar holder, as shown in figure 112. Tightening the clamping screw secures the boring bar holder in place.

8-17. *Boring tool geometry.* The angles ground on boring tools may vary as required to suit the material being machined. Boring tools with 10° side relief and back rake angles, figure 113,A, and

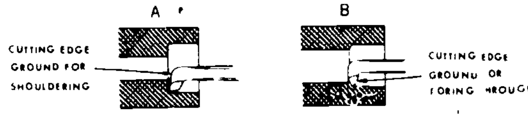


Figure 114. Cutting edge angles

a 15° side rake angle, figure 113,B, are commonly used for boring steel and give good results with other materials as well. The end relief, however, varies according to the diameter of the hole. No definite amount of end relief can be specified because the angle increases for smaller diameters. The end relief angle should be ground so that the heel of the tool will not rub the surface of the hole. Figure 113,C, shows the result of insufficient end relief.

8-18. The end of the boring tool may be ground, as shown in figure 114,A, for boring to a shoulder. It may be ground with a 20° entrance angle, as shown in figure 114,B, for boring completely through the hole.

8-19. *Tool setup.* Position the boring tool, as shown in figure 115, for straight boring. Note that the tool is positioned 5° above center; for taper boring it must be on center. Extend the boring bar only far enough to bore to the required depth. Be-

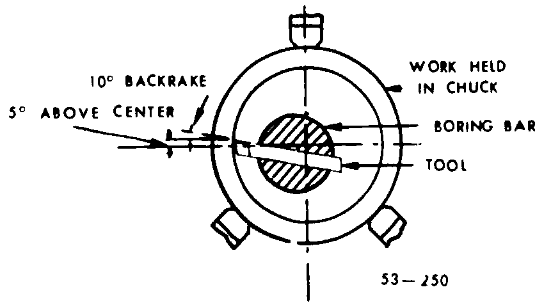


Figure 115. Boring tool position.

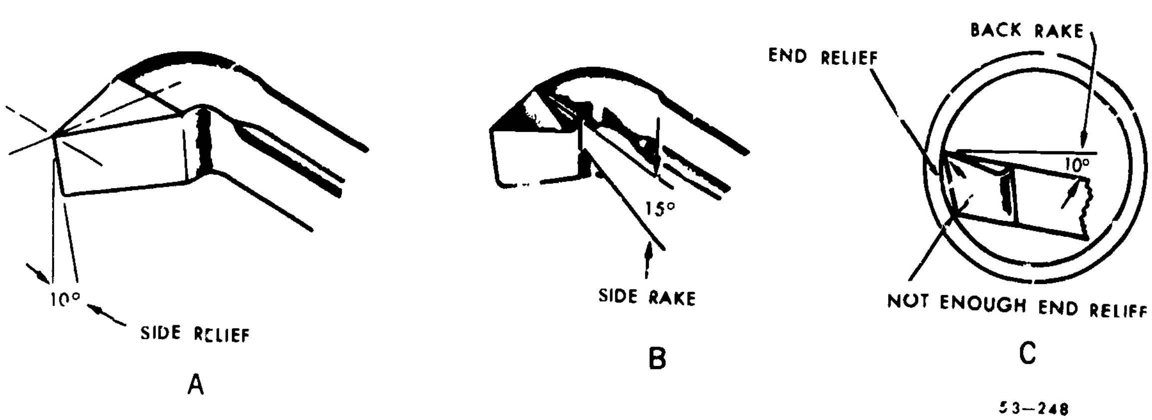


Figure 113. Boring tool angles.

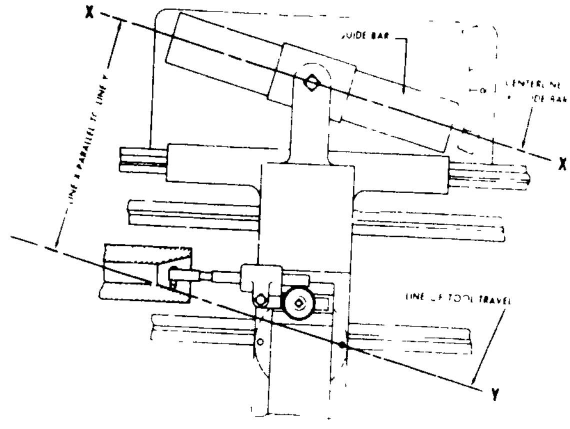


Figure 116 Alining taper attachment guide bar

fore starting the machine, move the boring tool into the hole to check for clearance.

NOTE: A pencil line drawn on the boring bar may be used to gage the distance to feed the tool into the hole; or a micrometer carriage stop, which

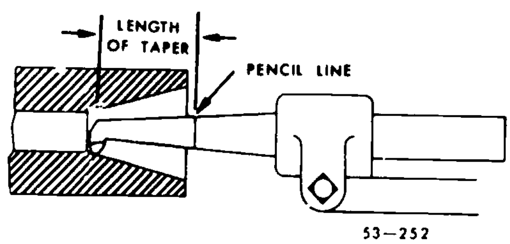


Figure 117. Marking boring bar.

we will discuss later in this volume, may be used to accurately bore to a given depth. (CAUTION: Always feed the carriage by hand for the last part of the cut when you are boring to a shoulder to prevent damage to the tool or the work).

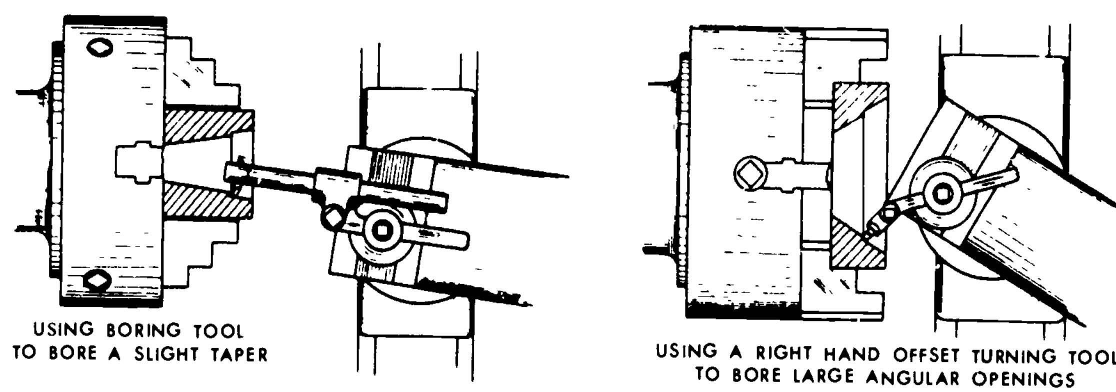
8-20. A bell-mouthed hole (larger at the start of the hole than at the end) may result from the spring in the boring bar, possibly caused by a dull tool or too heavy a cut. You can correct this by occasionally feeding the tool out of the hole with the same setting that you use to feed it into the hole, sharpening the tool, or reducing the feed or depth of cut.

8-21. The diameter of straight-bored holes may be measured with a micrometer and inside calipers. It may be measured with a micrometer and a telescope gage. Also, it may be measured with a go-no-go gage or mating parts.

8-22. *Taper boring.* Holes may be taper bored by using the taper attachment or the compound rest in the same manner as for taper turning. However, the cutting action takes place on internal surfaces instead of external surfaces, and boring tools instead of turning tools are used.

8-23. When you bore tapers by using the taper attachment, position the compound rest at a right angle to the ways. Mount the boring tool on the compound rest at center height. Then, swivel the taper attachment to give the desired taper. Swivel the taper attachment guide bar parallel to the line of travel of the cutting tool required to machine the desired taper, as shown in figure 116. Then, move the boring bar into the hole the desired depth. Scribe a line on the bar to indicate the approximate distance to feed the bar on succeeding cuts, as shown in figure 117. When you bore blind taper holes, you should use a drill size of the same diameter as the small end of the taper. If the exact drill size is not available, use a drill size a few thousandths of an inch larger. When you bore through holes, the drill size should be enough smaller than the small diameter of the taper to allow sufficient material for truing the hole.

8-24. The compound rest is convenient for boring short tapers and internal angles. The com-



A B 53—253

Figure 118. Boring short tapers.

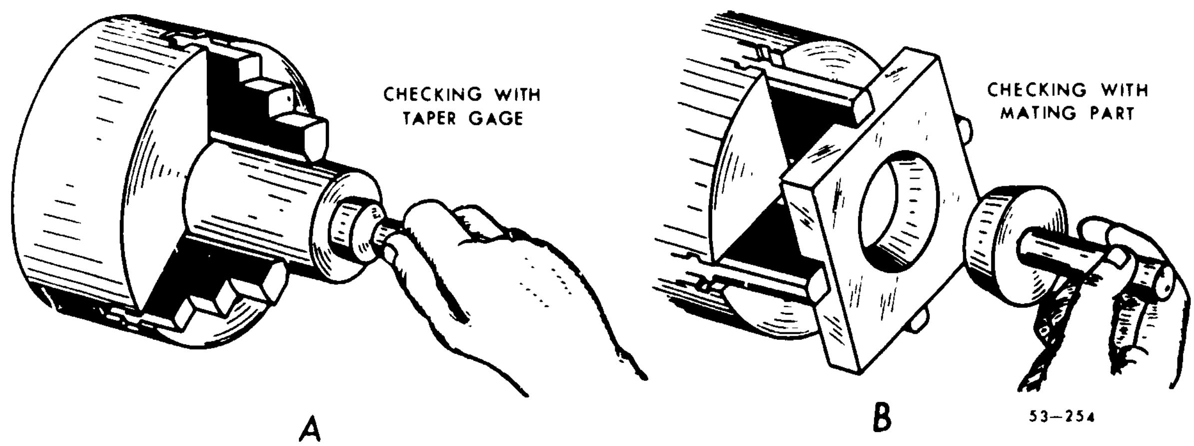


Figure 119. Checking internal tapers.

pound rest also permits you to bore steeper tapers than it is possible to bore by using the taper attachment. However, the length of taper that can be bored with the compound rest is limited by the length of travel of the compound rest slide. Use either a boring bar, figure 118,A, or a regular turning tool, figure 118,B, to turn short tapers, depending upon the size of the opening. Mount the tool in the lathe toolpost. The tapered hole is bored by feeding the compound rest by hand with the compound rest set at the required angle.

8-25. *Checking internal Tapers.* Internal tapers are generally checked for accuracy with a taper plug gage, figure 119,A, or with the mating part, figure 119,B. Insert the gage in the bored hole and wiggle it. Any movement of the gage indicates that the taper is incorrect. Adjust the taper attachment guide bar, or the compound rest when it is being used, and take trial cuts until the taper of the bore corresponds to the taper of the gage. Then bore the hole until the gage enters the bored hole far enough to aline the gage line with the surface of the work, as shown in figure 120. When the opening of the taper is quite large, you may check the angle with a protractor, as shown in figure 121.

8-26. **Reaming.** Reamers are used to produce smoother, more accurate, and more uniformly shaped holes than can be produced by drilling. Hand reaming and machine reaming are done in a

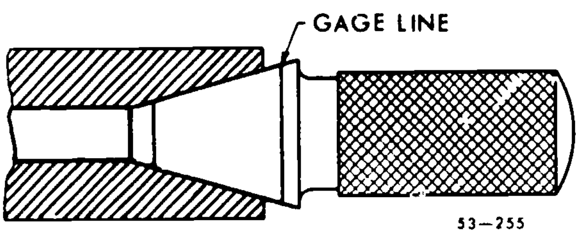


Figure 120. Gage line.

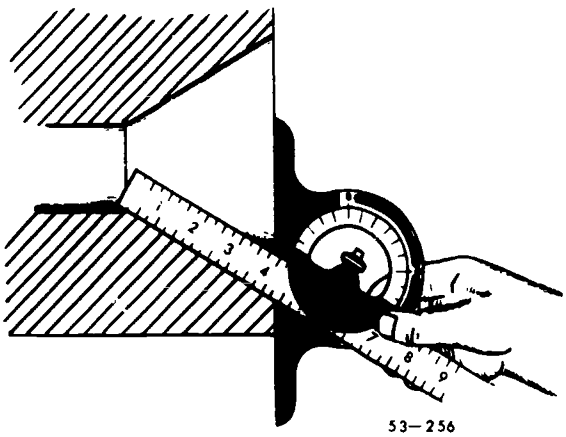


Figure 121. Measuring taper angle with a protractor.

lathe in the same manner as in a drill press, except that the reamer is in a horizontal rather than a vertical position. Always use the recommended cutting lubricant, shown in table 3, when you are reaming.

8-27. *Hand reaming.* Hand reaming is used to produce highly accurate holes. Usually, you should

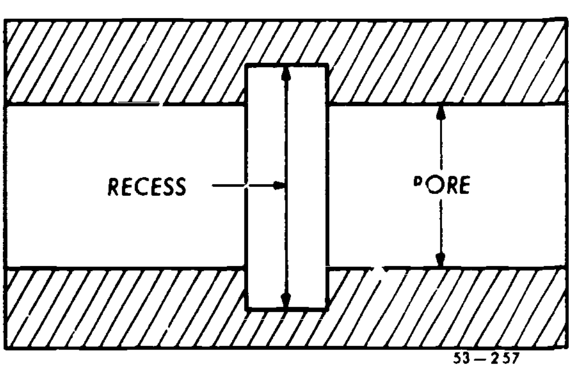


Figure 122. Recess.

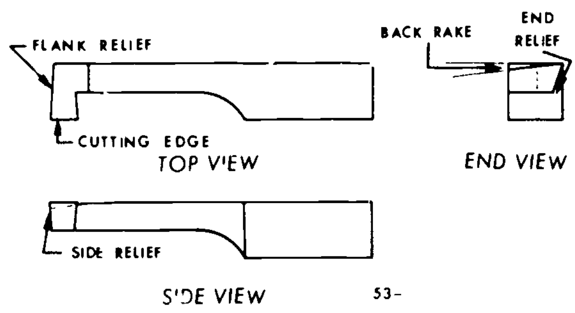


Figure 123. Recessing tool.

first drill the hole approximately 1/32 inch undersize, and then bore it within 0.003 to 0.005 inch of the final diameter. To hand-ream the hole, hold the work stationary in a chuck with the hand reamer supported on the tailstock center. Turn the reamer with a wrench as you feed it into the work with the tailstock handwheel.

CAUTION: Never attempt to use a hand reamer for machine reaming.

8-28. *Machine reaming.* Machine-reamed holes are usually less accurate than hand-reamed holes, but more accurate than drilled holes. You should drill or bore the hole to within 0.010 to 0.015 inch of final size and then ream it. The machine reamer is held in the same manner as a drill with the same type of shank. It is fed slowly and smoothly into the work. A constant rate of feed produces the best results. The spindle speed for machine reaming is half the speed which you would use to drill a hole of the same diameter.

8-29. **Recessing.** Recessing, sometimes called *channeling* or *chambering,* is the process of machining a groove, as shown in figure 122, in a drilled, bored, or reamed hole. Recesses are usually machined to provide room for tool "runout" for subsequent operations. Grooves are often machined to accommodate snaprings or O-rings and for lubrication.

8-30. High-speed tool blanks, ground to the proper shape and held in a boring bar and holder, can be used as *recessing tools.* However, the recessing tool must have flank relief on both sides of the cutting edge. A large high-speed tool blank which is ground as shown in figure 123 makes a satisfactory tool for machining small recesses. Be careful not to exert too much pressure on this type of took, as it is brittle and breaks easily.

Threading Operations

ONE OF THE MOST important operations that can be performed on a lathe is the cutting of threads. Bolts, studs, nuts, and threaded shafts are used extensively as holding and measuring devices. A knowledge of threads, thread systems, and thread cutting is essential to you as a machinist. This chapter will be divided into two sections. The first will be devoted to basic information and the second to the machining of threads.

9. Threads and Thread Measurement

9-1. In this section we will discuss basic thread information, such as threading terms, the Unified Thread System, thread designation, thread measurement, and threading tools.

9-2. **Threading Terms.** To be able to understand threads and threading operations, you must know the meaning of certain terms. Figure 124 will help you to understand the following definitions.

a. Thread. A thread is the ridge or projection remaining after a uniform, helical groove is cut on the outside or inside of a shaft or hole.

b. Threads per inch. Threads per inch is the number of threads per inch measured parallel to the thread axis. It is used in conjunction with the outside diameter to designate the size of the thread. For example, 3/4—10 indicates 10 threads per inch on a piece of stock 3/4 inch in diameter.

c. Thread angle. The thread angle is the angle formed by the intersection of the two sides of the thread groove.

d. Helix angle. The helix or lead angle is the angle formed by the inclination of the thread and a plane perpendicular to the thread axis.

e. Major diameter. The major diameter is the largest diameter of an external or internal thread.

f Pitch diameter. The pitch diameter is the diameter of an imaginary cylinder that is concentric with the thread axis and whose periphery passes through the thread profile at the point where the width of the thread and the thread groove are equal. The pitch diameter is the diameter which is measured when the thread is machined to size. A change in pitch diameter changes the fit between the thread being machined and the mating thread.

g. Nominal size. The nominal size is the size which is used for identification. For example, the nominal size of a 1/2—20 thread is 1/2 inch, but its actual size is slightly smaller to provide clearance.

h Actual size. The actual size is the measured size.

i. Basic size. The basic size is the theoretical size. The basic size is changed to provide the desired clearance or fit.

j. Pitch. Pitch is the distance from a point on a thread to a corresponding point on the next thread measured parallel to the thread axis.

k. Lead. Lead is the lateral distance a thread moves per revolution. On a single-lead thread, the lead and the pitch are identical; on a double-lead thread, the lead is twice the pitch; on a triple-lead thread, the lead is three times the pitch; etc. Figure 125 shows the difference between the lead and the pitch of a double-lead thread.

l. Crest. The crest of a thread is the top surface that joins the two sides of the thread.

m. Root. The root of a thread is the bottom surface that joins the two sides of adjacent threads.

n. Truncation. Truncation is the perpendicular distance from the crest of a thread or the root of a thread and the point of intersection that would be created if the sides of the thread were extended to form a sharp "V."

o. Crest clearance. Crest clearance is the perpendicular distance between the crest of a thread and the root of a mating thread when it is engaged.

p. Thread depth. Thread depth is the perpendicular distance between the crest and root of a thread.

q. Width of a basic crest or a basic root. The width of a basic crest or a basic root of an American Standard Unified thread is one-eighth of the pitch.

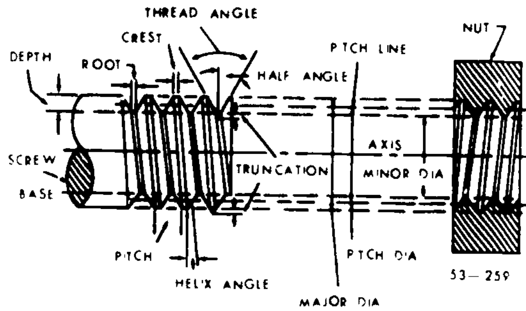


Figure 124. Thread parts.

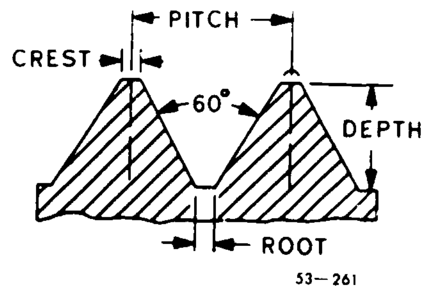


Figure 126. Unified and National thread form.

9-3. Now that you are familiar with the terms which are used to describe the various parts of a thread, we will discuss the *American Standard United Thread System,* or as it is commonly called, the *Unified Thread System.*

9-4. **Unified Threads.** The Unified Thread System is the thread system which is currently used for most screws, bolts, and threaded products produced in the United States. It is replacing the American National Thread System which has been used for many years. The two systems are very similar; the main differences are the tolerances and the allowances and the manner in which they are applied. The Unified Thread System was established by the United States, the United Kingdom, and Canada to provide a thread system that would permit all three countries to interchange threaded parts.

9-5. *Form.* The Unified thread form is practically the same as the form which has always been used in the United States. The preferred American practice is to retain a flat crest, whereas the British prefer a rounded crest. The root of the thread, as machined in the United States, may be either flat, as it is cut by a new tool, or it may have whatever rounding that results from a worn tool. The British prefer the rounded root. However, these minor variations in machining do not interfere with one thread being interchanged with the other. Figure 126 shows the forms of the Unified thread and the American National Thread.

9-6. *Series.* Thread series are groupings of diameter and pitch combinations that differ in the number of threads per inch for a thread of a given diameter. The three common series are the coarse thread series, the fine thread series, and the extra fine thread series. In addition to these, there are series based on a constant number of threads regardless of the diameter. The constant pitch series are based on 4, 6, 8, 12, 16, 20, 28, and 32 threads per inch. Tables listing the diameter and pitch combinations for the various thread series

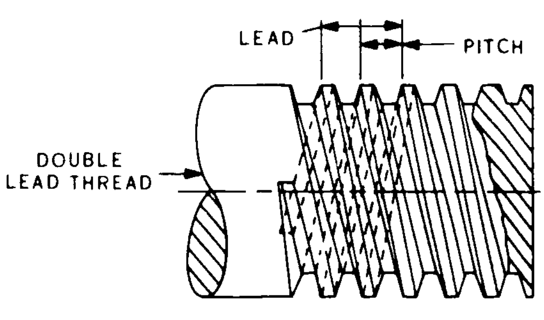


Figure 125. Double-lead thread.

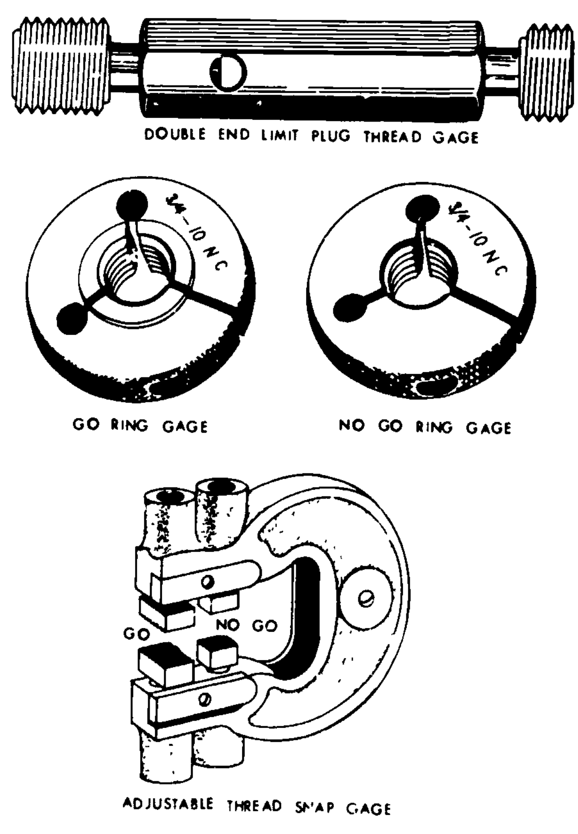


Figure 127. Thread gages.

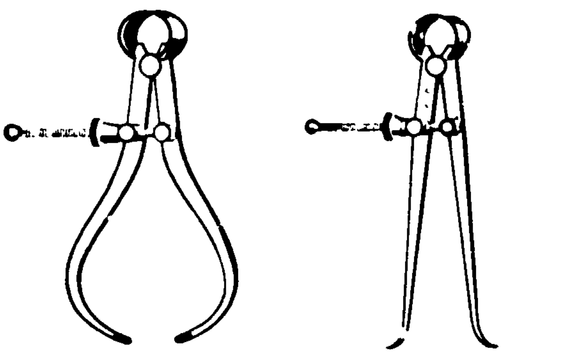


Figure 128. Thread calipers

can be found in machinists' publications, such as the *Machinery's Handbook.*

9-7. The *coarse thread series* is most commonly used for general applications. It is also used when threaded holes are required in soft metals, such as aluminum, brass, bronze, magnesium, etc., because the threads are deeper. The *fine thread series* is often used for aircraft nuts, bolts, etc., because vibration won't loosen them as easily as coarse threads. The fine series is also used when only a short length of thread engagement is possible and when the wall thickness of material prohibits the use of the deeper coarse thread series. The *extra fine series* is used for applications similar to the fine series when even finer threads are required, such as adjustment screws in a throttle of an aircraft.

9-8. **Thread Designation.** A thread is designated according to the nominal size, the number of threads per inch, the series symbol, and the class symbol, in that order. For example, the designation 1/4—20 UNC—3A is explained as follows:

1/4 = nominal thread diameter
20 = number of threads per inch
UNC = series (Unified coarse)
3 = class
A = external thread

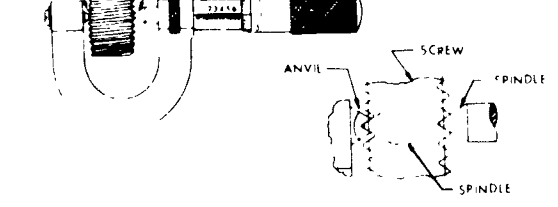


Figure 129. Measuring threads with a thread micrometer.

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. The symbols used to identify the thread series are these:

UNC- Unified coarse
UNF—Unified fine
UNEF—Unified extra fine
UNS—Unified special

9-9. *Constant pitch series* are identified by the number of threads per inch of the series, followed by the Unified symbol (UN), such as 8UN or 32UN. The same symbols are used to identify the old National Thread System, except that the letter U is omitted. For example: 1/4—20NC—2.

9-10. The *class symbol* designates the tolerance grouping to apply to a given thread. Each size of thread has three classes of fit, which are identified

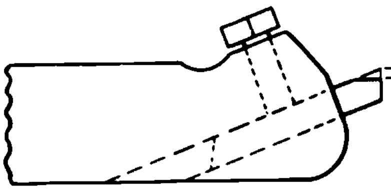


Figure 130 Position of external threading tool in toolholder.

as class 1, 2, or 3. In addition to the numerical designation, the letter A indicates an external thread class and the letter B indicates an internal thread class. The tolerances for class 1 threads are greater than those for class 2, and the tolerances for class 2 threads are greater than those for class 3 threads. Any desired fit can be obtained by using an external thread of one class with an internal thread of another class, such as a 2A bolt and a 3B nut. Complete listings of Unified threads and the dimensions of the threads for each class can be found in machinists' publications, such as the *Machinery's Handbook.*

9-11. **Thread Measurement.** Thread measurement is necessary to insure that the thread and its mating part will fit properly. It is important that you know the various measuring methods and the calculations which are used to determine the dimensions of threads.

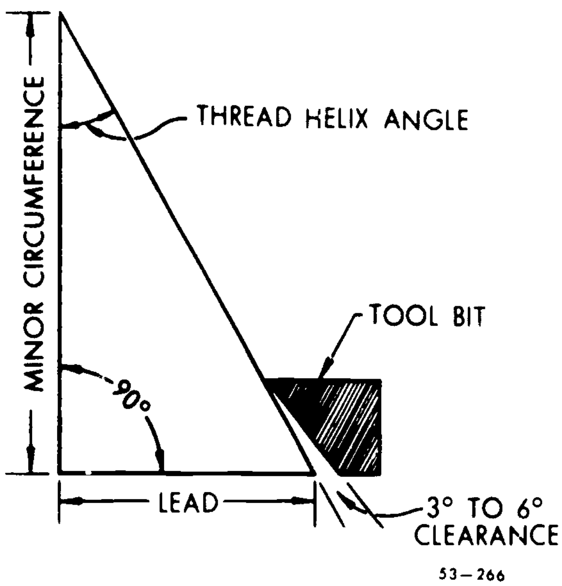


Figure 131. Calculation of helix angle.

9-12. Several methods can be used to check threads. The one which you will use will depend upon the accuracy required for the particular thread which you are machining.

9-13. *Mating part.* The use of a mating part is a common practice when average accuracy is required. The thread is simply machined until the mating part will assemb'?. A snug fit is usually desired, with very little play, if any, between the parts.

9-14. *Thread gages.* Go and no-go gages, such as those shown in figure 127, are often used to check threaded parts. The thread should fit the *go* portion of the gage; but should not screw into the *no-go* portion. The threaded plug gage is one of the most exact means of checking internal threads.

9-15. *Thread calipers.* Thread calipers, shown in figure 128, are similar to common calipers, except that the legs are ground to the shape of a thread. They are used to measure from a finished thread to the thread being machined and are fairly accurate.

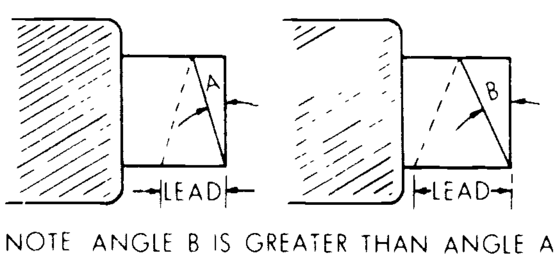


Figure 132. Helix angle increases as lead increases.

9-16. *Thread micrometers.* Thread micrometers are used to measure the pitch diameter of threads. They are graduated and are 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 129. Thread micrometers come in the same size ranges as ordinary micrometers: 0 to 1 inch, 1 to 2 inches, etc. In addition, they are available in various pitch ranges. The number of threads per inch must be within the pitch range of the thread micrometer. The micrometer method is one of the most accurate methods for measuring pitch diameter.

9-17. **Threading Tools.** Before you can cut a thread on a lathe, you need tools that have been properly ground for thread cutting. The following information pertains to threading tools for Unified and National threads.

9-18. The threading tool must be sharp, smooth, and properly shaped to machine the desired thread form. The Unified and National

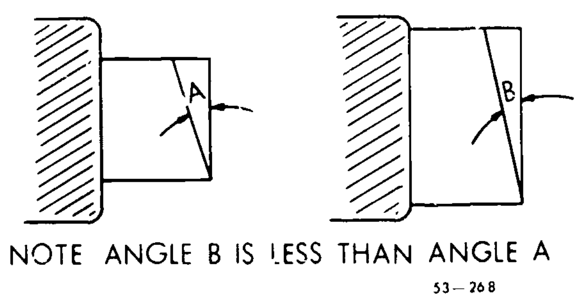


Figure 133. Helix angle decreases as diameter increases.

thread forms have a 60° included angle and straight sides. The threading tool must be ground to produce this form. It is good practice to grind two threading tools, one for rough threading and one for finish threading. The roughing tool should have a 10° to 15° side rake angle and 0° back rake angles.

9-19. The tops of both roughing and finishing tools must be ground as shown in figure 130 to eliminate the back rake given the tool by the toolholder. The cutting edge of the tool must be horizontal in the toolholder and set at center height. The effective end relief angle should be the same as for turning tools, approximately 8° to 10°. The side relief angle depends upon the helix angle of the thread. You can find the helix angle of the thread in a machinist publication, such as the *Machinery's Handbook*, or by using the following formula:

$$\text{Tangent of the helix angle} = \frac{\text{lead of thread}}{\text{thread circumference at minor diameter}}$$

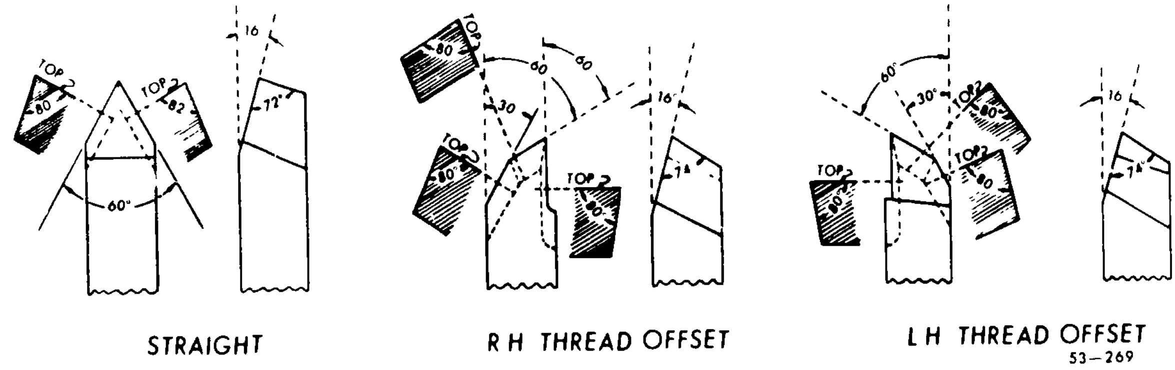


Figure 134. Straight and offset threading tools.

9-20. The side relief angle is ground 3° to 6° more than the helix angle, as shown in figure 131. Figure 132 shows why more side relief is required as the thread lead increases. Figure 133 shows why less side relief is required to cut a given number of threads on a large diameter shaft than on a shaft of smaller diameter.

9-21. It is not practical to grind a tool for every pitch thread; therefore, you should grind the tool to accommodate a range of pitches. A tool with 8° to 10° side relief will safely cut threads which have more than eight threads per inch. Side relief should be ground only on the leading or cutting edge, although it is good practice to grind a slight relief on the trailing side. Check the form of the threading tool with a 60° center gage. The roughing tool should be ground from 1/4° to 1/2° less than the gage angle, while the finishing tool should be ground to fit the 60° gage angle. Both sides of the thread are finished when you use a finishing tool of this shape. For cutting fine pitch threads and threads that do not require great accuracy, you may finish the thread to size with the roughing tool. The truncation or flat on the end of the tool should be from one-half to two-thirds that required by the thread in order to produce sufficient relief at the root of the thread. You may grind the threading tool to the most convenient shape for the job at hand, straight or offset, as shown in figure 134. Grind tools for internal threading with end relief to conform with the radius of the bore.

9-22. Right-hand and left-hand threading tools differ in the same manner as right-hand and left-hand turning tools. When you view the threading

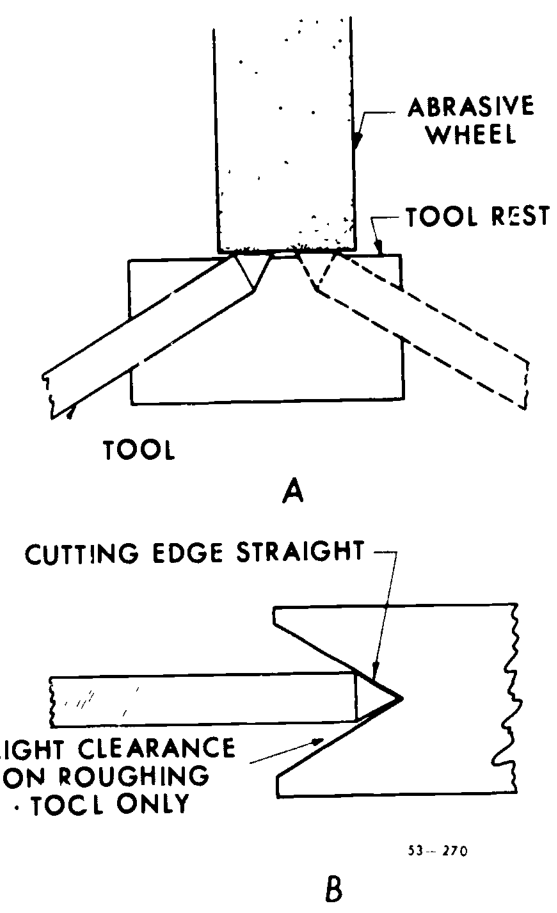


Figure 135. Grinding threading tool form.

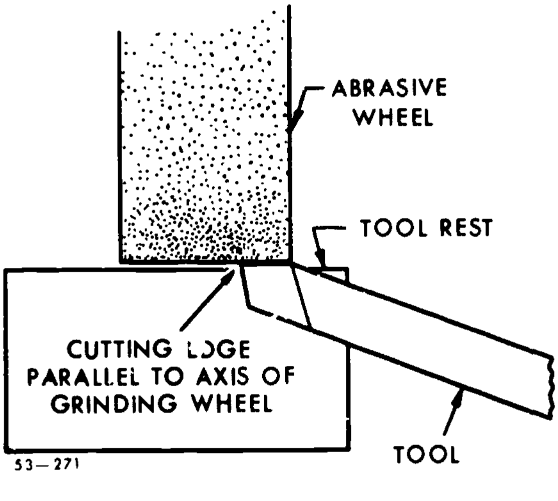


Figure 136. Grinding side rake.

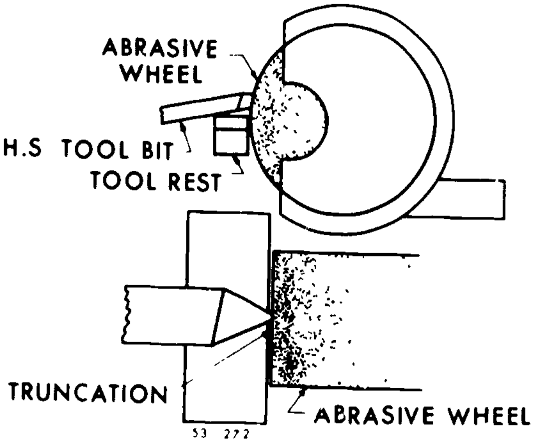


Figure 137. Grinding truncation.

tool from the front or cutting end, the cutting edge is on the right side of a right-hand tool and on the left side of a left-hand tool.

9-23. You would grind a threading tool by first grinding the end relief angle. Then grind the top of the tool to eliminate back rake, as shown in figure 130.

NOTE: You may place the tool in a toolholder to check the 0° back rake angle.

Next, grind the included angle of the tool and, at the same time, grind the side relief angle on the cutting edge, as shown in figure 135. Tilt the tool as you grind it so that the cutting edge is parallel to the axis of the grinding wheel. Check the included angle with a center gage, and check the cutting edge for straightness, as shown in figure 135. Beginning at the top of the trailing edge of the tool, grind the side rake angle until the surface "runs out" at the cutting edge. Keep the cutting edge parallel to the wheel axis, as shown in figure 136.

NOTE: Omit doing this when you are grinding a finishing tool. Grind the truncation on the point of the tool, as shown in figure 137. For very fine threads, use an oilstone for this purpose. Grasp the

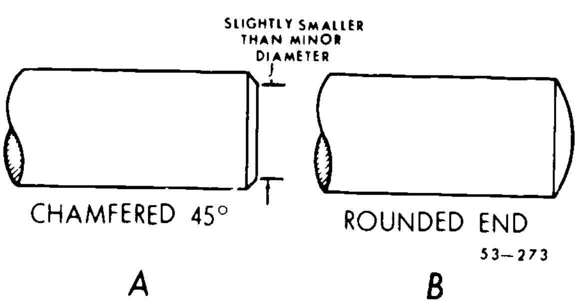


Figure 138. End of work prepared for threading.

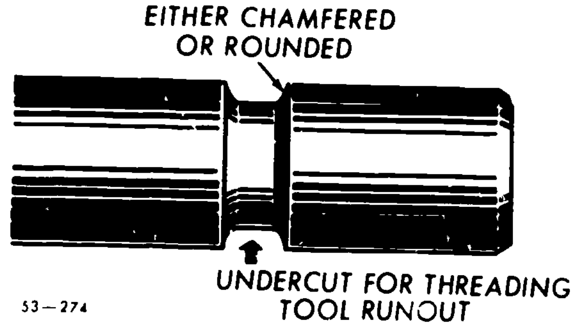


Figure 139. Undercut for threading tool runout.

tool close to the end being stoned and stone the tool with the relief and rake surfaces held flat against the oilstone. This method prevents rounding the cutting edges and changing the shape of the tool.

10. Machining American Standard and Unified Threads

10-1. All of the thread cutting you do involves either the machining of external or internal threads. The machining of the latter differs mainly in the way in which the work and the cutting tool are set up. In this section, we will discuss machining external and internal threads, threading with a tap and die on a lathe, left-hand threads, multiple threads, and pipe threads.

10-2. **Machining External Threads.** Thread cutting is done by causing the carriage and tool to move along the work a distance equal to the required lead of the thread for each revolution of the work. This operation is known as chasing threads. The tool is ground to the shape of the thread to be cut, and the resulting thread is a helical ridge of uniform section. The lead screw of the lathe is rotated by means of a gear train connection between

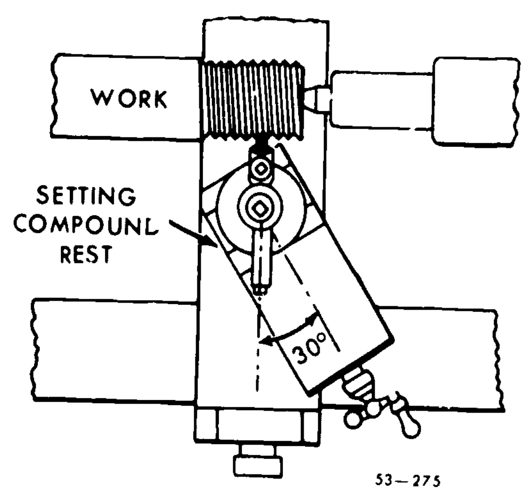


Figure 140 Position of compound rest.

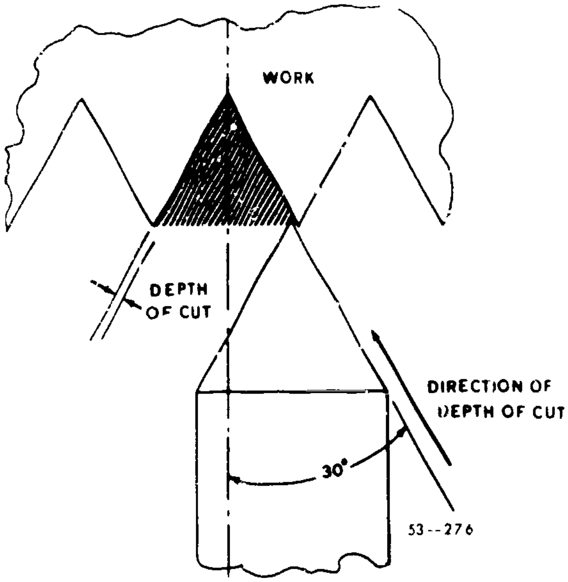


Figure 141. Action of threading tool.

the spindle and lead screw. The split nut mounted in the apron engages the lead screw and actuates the movement of the carriage. On lathes having both a feed rod and lead screw, it is necessary to disengage the feed rod and power feed clutches and to transfer the power from the quick-change gearbox to the lead screw by making the necessary lever changes.

10-3. To obtain good results in cutting threads, you must make sure that (1) the work is properly prepared, (2) that the tool is correctly set up, and (3) that the lathe gearing is properly arranged before you begin the actual threading operation. First, we will discuss the preparation of the work;

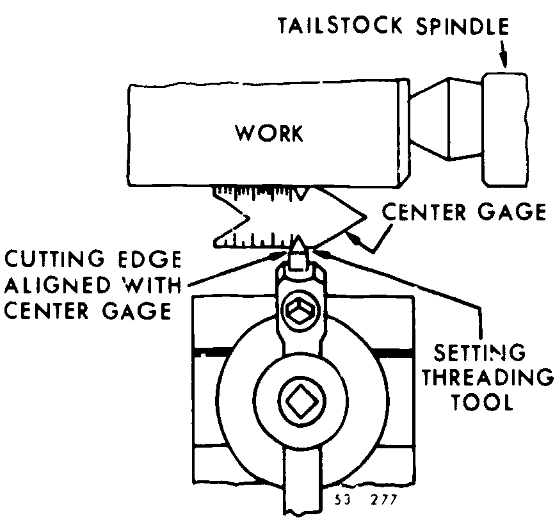


Figure 142. Positioning threading tool

then we will discuss the tooling setup, the lathe gearing, and the thread-cutting operation.

10-4. *Preparation of the work.* The portion of the work to be threaded must be machined to the major diameter of the thread. You can use any suitable method to hold and drive the work. If the work is held between centers, the tail of the lathe dog should be wedged securely against the side of the driveplate slot which contacts the tail. A small wooden wedge can be used to do this. Drive the wedge in tight enough to insure that it will remain in position. If the dog is not held solidly in the slot, the threading tool may "grab" when a cut is started and damage the threads or tool.

10-5. Either chamfer the end of the portion of the work being threaded at a 45° angle to a depth slightly greater than the thread depth, figure 138,A, or machine it in the form of a radius, figure 138,B. Chamfering prevents the formation of a sharp, easily damaged first thread. It also helps to aline the mating threads.

10-6. A groove is often machined at the headstock end of the thread portion prior to cutting the thread to provide a place for the threading tool to

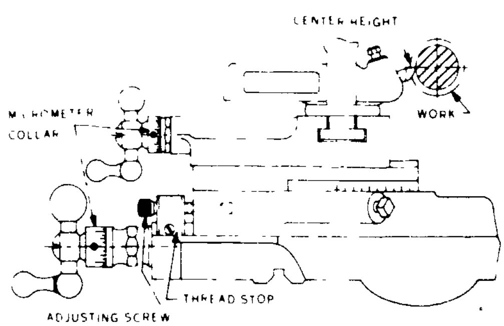


Figure 143. Thread stop.

run out. This allows threaded parts to screw on all the way to a shoulder. The groove should be slightly wider than the thread pitch, and the depth should be equal to or slightly greater than the thread depth. The side of the groove adjacent to the portion to be threaded should be chamfered, or rounded, as shown in figure 139. This prevents cutting a sharp edge of thread during the threading operation. You may use a roundnose tool or a parting tool which has slightly rounded corners to machine the groove.

10-7. *Threading tool setup.* Position the compound rest 30° to the right of the perpendicular position, as shown in figure 140, to rough cut 60° right-hand threads. The thread is produced by a series of cuts which are made by feeding the threading tool into the work at the 30° angle with the compound rest, as shown in figure 141. This

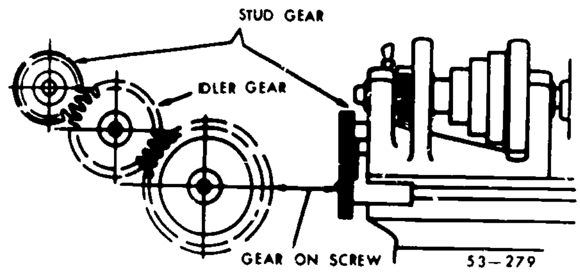


Figure 144. End gearing.

produces the correct angle on the trailing side of the thread groove. You obtain the angle on the leading side of the thread groove by setting the cutting edge to the proper angle. Set the threading tool at center height and aline the cutting edge with a center gage, as shown in figure 142.

10-8. Use the crossfeed to withdraw the tool at the end of the cut and to reposition the tool at the start of each cut. A thread stop may be used to quickly bring the cross-slide and tool to the original setting at the beginning of each cut. Some thread stops directly control the movement of the crossfeed screw, whereas others fasten to the cross-slide, as shown in figure 143. Set the thread stop adjusting nuts to stop the threading tool just before it touches the surface of the work when it is moved toward the work with the cross-slide. When a thread stop is not used, set the cross-slide graduated collar to read "zero" just before the threading tool touches the work. Pick up the surface by feeding the tool the remaining distance with the compound rest.

10-9. *Lathe gearing*. The ratio between the spindle rpm and the lead screw rpm determines the number of threads per inch that will be produced. If the spindle revolves 12 times while the threading tool advances 1 inch, 12 threads per inch will be produced. Changing the ratio between the spindle and the lead screw changes the number of threads per inch. On modern lathes a quick-change gearbox is used to change the ratio. The number of threads per inch desired is obtained by positioning the gearbox levers, as indicated by the plate that is usually fastened to the gearbox. The gearbox levers shift the gears within the gearbox and vary the ratio between the spindle rpm and the lead screw rpm. This will give the lead screw rpm to obtain the desired number of threads per inch.

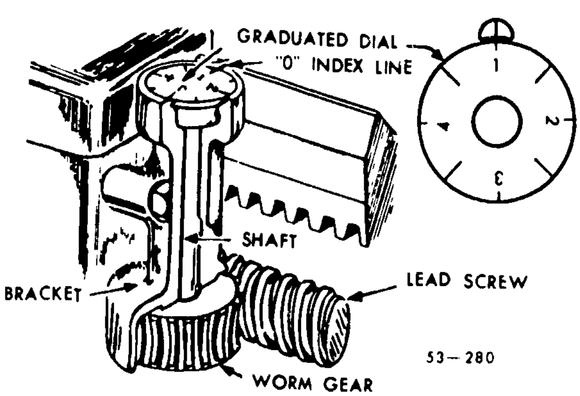


Figure 145. Thread chasing dial.

10-10. Occasionally, a gearbox will not produce the number of threads per inch that must be cut. However, it is possible to obtain the desired number of threads per inch by replacing the spindle stud gear, shown in figure 144, with a gear having a different number of teeth. You would use the following formula:

$$N = \frac{A \times B}{C}$$

N = number of teeth on the replacement gear.
A = number of teeth on the spindle stud gear.
B = number of threads per inch to be cut.
C = number near to B on the gearbox that will make N an even number.

10-11. Assume that you wish to cut 27 threads per inch and that 27 is not available on the gearbox. Count the number of teeth on the spindle stud gear and use this number as A in the formula. In this case, assume that it is 32. B is equal to 27. The formula now reads:

$$N = \frac{32 \times 27}{C}$$

Now, select a number of threads per inch nearest 27 that are available on the gearbox plate and solve the formula, using each number as C, until N finally is an even number. By trial and error, you find that 24 is the number nearest 27 that makes N come out as an even number:

$$N = \frac{32 \times 27}{24}$$

Solving the formula, you obtain a value of 36 for N. If the original spindle stud gear of 32 teeth is replaced with a gear of 36 teeth and the gearbox levers are positioned to cut 24 threads per inch, 27 threads per inch will actually be produced.

NOTE: Always replace the original spindle stud gear after the thread has been cut. The lathe will not cut the number of threads shown on the gearbox plate if the substitute is left in the lathe.

10-12. Now the work has been prepared and the threading tool is properly positioned; arrange the gearbox or end gearing to produce the number

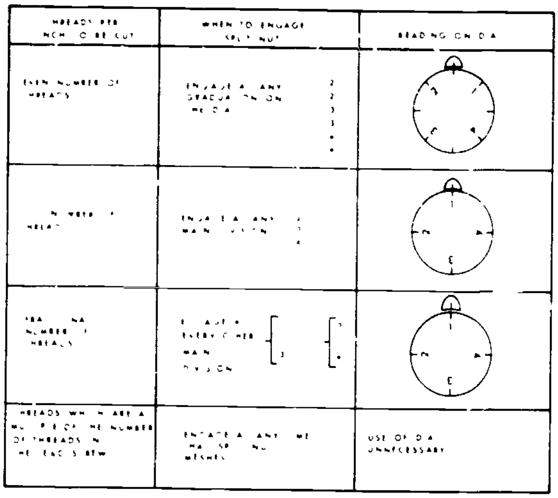


Figure 146. Thread chasing dial instructions

of threads per inch desired. You are ready to cut the thread.

10-13. *Cutting threads.* In cutting threads, first pick up the surface of the work with the threading tool. Then move the tool to the right of the work. Make a depth of cut with the compound rest. Position the split-nut lever to engage the split nut with the lead screw in order to start the thread-cutting operation. Engaging the split nut with the lead screw causes the carriage to move as the lead screw revolves. Cut a thread by making a series of cuts in which the threading tool follows the original groove for each cut. Use the thread-chasing dial, shown in figure 145, to determine when to engage the split nut so that the threading tool will track properly. The dial is attached to the carriage and is driven by means of the lead screw. An instruction plate, which is usually attached to the apron of the lathe, provides instructions about the use of the thread-chasing dial for various threads; read them carefully before you start the threading operation. Some types of lathes are not equipped with a thread-chasing dial; instead, they have a reversing lever. When you cut threads with this type of lathe, it is necessary to withdraw the tool at the end of the cut and reverse the direction of motion of the lead screw. When the thread being cut is a multiple of the threads per inch on the lead screw, the split nut may be closed at any position. In this case, it is not necessary to use the thread-chasing dial. Figure 146 shows a typical thread-chasing dial and the instructions on its use.

10-14. Allow the threading tool to feed to the end of the threaded section. Back the tool away from the surface of the work with the cross-slide and, *at the same time,* disengage the split nut. Move the tool to the right end of the work with the carriage and stop the lathe. Check the number of threads per inch cut on the work with a screw pitch gage, a center gage, or a rule, as shown in figure 147. Check the side relief angle of the threading tool visually by using a rule, as shown in figure 148. If the number of threads per inch and the clearance angle are correct, move the tool with the cross-slide until it is stopped by the thread stop, or move it until the graduated collar is zeroed, if you are using the collar. Take an additional depth of cut with the compound rest and engage the split nut at the proper time, as shown by the thread-chasing dial. At the end of the thread, disengage the split nut and back the tool away at the same time. Move the tool to the right end of the thread and make additional cuts until you obtain the desired pitch diameter.

10-15. Use spindle speeds that are much slower than those you would use for a regular turning operation. Faster speeds can be used as you gain skill and experience. The depth of cut varies with the size of the thread, the material being cut, and the

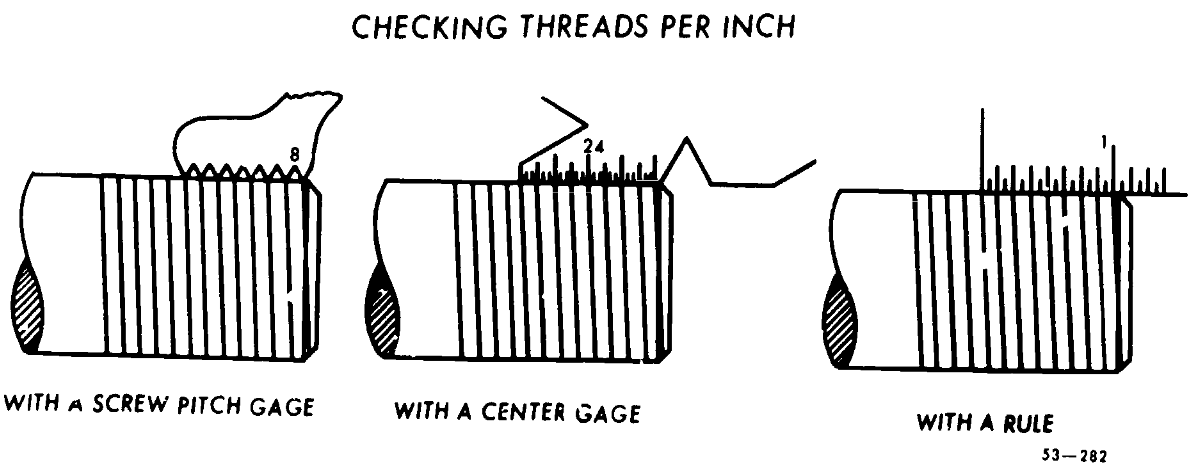


Figure 147. Checking thread pitch.

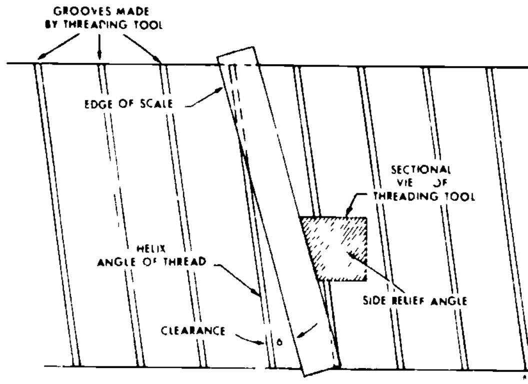


Figure 148. Checking side clearance of threading tool.

kind of finish desired. Very coarse threads permit greater depths of cut than finer threads. Threads that do not require finishing should be roughed to size by making smaller depths of cut to ensure a smoother back side of the thread. For cutting 8 to 10 threads per inch, the depth of cut for the first few cuts may be from 0.005 to 0.010 inch. As the thread becomes deeper, the depth of cut should be decreased until only 0.002 to 0.003 inch is taken. Always use the cutting lubricant recommended for the material being machined.

NOTE: The movement of the compound rest to cut the thread to depth will be approximately ¾ of the pitch, or $\frac{0.75}{\text{number of threads per inch.}}$

10-16. For very fine threads and for threads requiring only an ordinary degree of finish, you may omit the finishing step. Finish the thread to size with the roughing tool by taking light cuts during the roughing operation. For precision thread cutting, the thread is roughed out, leaving a sufficient amount of material for finishing (0.005 to 0.010 inch on the thread depth). The amount will depend upon the coarseness of the thread pitch.

10-17. You may use several methods to finish threads. A simple and effective method for finishing the general range of thread pitches is to use a finishing tool ground to a 60° included angle. Feed the tool with the compound rest set at 29°. The 29° setting of the compound rest allows a fine, light cutting action with the leading or cutting edge of the tool on the front side of the thread. It also produces a slight cutting or scraping action on the back side of the thread. Therefore, both sides of the thread are finished simultaneously. Take successive passes over the work until the thread is cut to the proper depth. For best results, the last few finish cuts should be very thin (0.0005 inch or less, if possible). A cut of this depth will produce a mirrorlike finish, provided that all other conditins are correct. The tool must take a positive chip from the work at all times. If the tool is not sharp or if too light a cut is taken, the tool will ride over the work and the surface will be hardened to some extent. On the next pass, the tool may take a heavy cut and gouge the work.

10-18. Very coarse pitch threads (6 pitch and coarser) are sometimes finished by first finishing the back side of the thread and then finishing the front side The finishing tool is ground to an included angle of slightly less than 60° (58° to 59°) to allow clearance between the tool and the opposite side of the thread as each side of the thread is being finished. The threading tool has neither side nor back rake. When you use this method of finishing, you pick up the back side of the thread and feed the tool straight into the work with the cross-slide to the thread depth. Then, reset the tool, pick up the front side of the thread, and finish it by feeding the tool with the compound rest set at 30° until the thread depth is reached. This method of finishing, although effective, is very exacting. Any error in setting the tool and picking up the thread will be reproduced on the thread being cut.

10-19. After removing the threading tool to sharpen it or to replace it with a finishing tool, you have to pick up the threading cut. You should mount the new threading tool at center height and at the correct angle. Back the tool away from the work, engage the split nut, and stop the lathe after the tool has traveled part way along the threaded portion. LEAVE THE SPLIT NUT ENGAGED. If you place a piece of white paper on the cross-slide, it will assist you by providing better visibility between the tool and the work. Position the threading tool in the thread groove by using the cross-slide and compound rest. Pick up the back side of the thread groove with the point of the tool by placing a piece of paper between the thread surface and tool to act as an aid. Leave some clearance between the cutting edge of the tool and

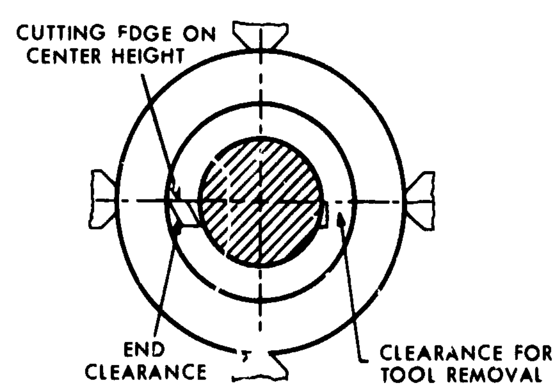


Figure 149. Position of internal threading tool.

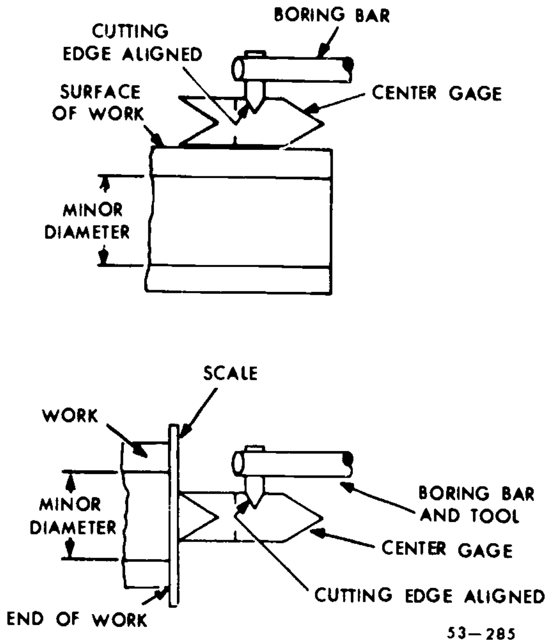


Figure 150. Alining cutting edge of internal threading tool.

the front side of the thread groove. Now set the cross-slide graduated collar at zero or lock the thread stop. Withdraw the tool and begin a new cut by engaging the split nut as the work revolves. Pick up the cut on the front side of the thread by slowly feeding the tool in with the compound rest as it travels along the thread groove. Continue machining the threads in the usual manner.

10-20. **Machining Internal Threads.** The rules controlling internal thread cutting are much the same as those for external thread cutting. However, for internal thread cutting you will encounter the same clearance restrictions and tool problems that are present in internal recessing opera[illegible]. Use the sam[illegible] toolholders for internal thr[illegible] ting that you use for boring, except that [illegible] tool has a thread form which is fed at th[illegible] cutting ratios of feed-to-spindle revolutions. Another difference between boring and internal threading is in the cutting ang[illegible] at which the cutting too[illegible] approaches the work. As in ext[illegible]nal thread cutting, the internal threading tool must engage the work on dead center. It must be held so that the cutting edge has 0° back rake, as shown in figure 149. Use a center gage to aline the cutting edge of the threading tool. using one of the methods shown in figure 150.

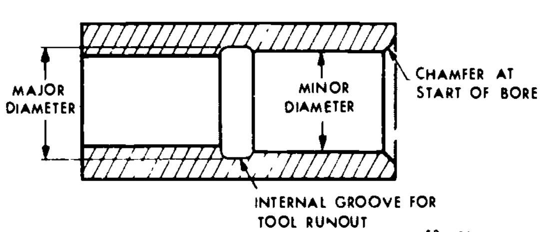


Figure 151. Preparation of work for internal threads

10-21. An internal threading tool requires greater end and side clearance than an ordinary threading tool. The length of the tool held in the boring bar is restricted. Internal threading tools must have enough clearance at the back side of the bore to permit moving the tool clear of the thread for removal, as shown in figure 149. Before cutting an internal thread, you must first bore the work to the minor diameter of the internal thread. Grind the end relief of the cutting tool to conform to the radius of the bore. Holes to be threaded on the lathe are usually bored to size. The hole size or minor diameter for the internal Unified thread is found by the formula:

$$\text{Minor diameter} = \text{major diameter} - \frac{1.0825}{\text{number of threads per inch}}$$

10-22. If the thread does not extend completely through the hole, an internal groove with a diameter equal to the major diameter of the thread should be machined at the end of the threaded portion for tool runout, as shown in figure 151. A chamfer of 45°, also shown in figure 151, should be machined at the start of the bored hole to help aline the mating threads and to eliminate a sharp first thread.

10-23. The compound rest in internal threading is normally positioned 30° to the left of the perpendicular position, as shown in figure 152. If there is insufficient clearance between the compound rest and the work, lathe chuck, or driveplate, position the compound rest 180° from the position shown in figure 152. The feed for successive cuts is toward, rather away from you, and the setting of the thread cutting stop is therefore reversed. Due to the extension of the cutting tool from the toolpost, cuts must be lighter, and overhang should be reduced to a minimum in both tool and work setup. An extra finishing cut should be taken without changing the setting of the compound rest.

10-24. Internal threads are cut in all the thread forms which are used for external threads. One factor that calls for special attention in cutting internal threads of special shapes is the difference in the clearances between nut and screw recommended for the different thread types. There may be times when you have to determine the size of the bore and a listing of these special recommended clearances is not available. A safe rule of thumb is to cut an internal thread from 0.005 inch

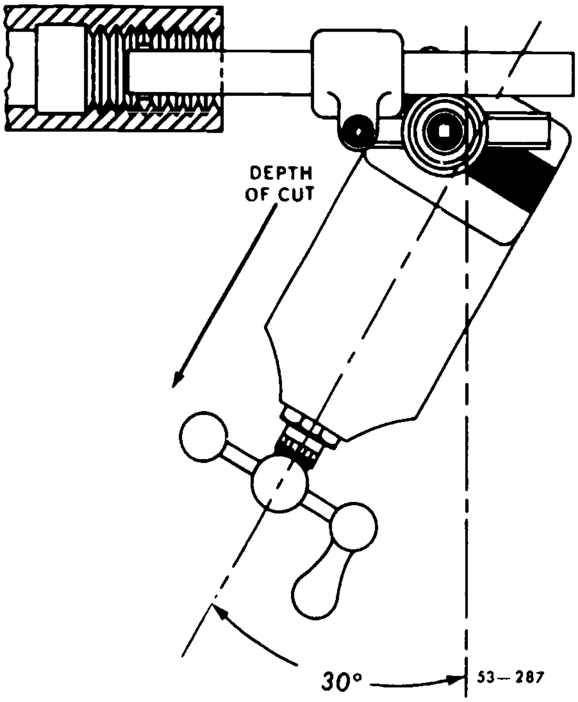


Figure 152. Position of compound rest for internal threads.

to 0.010 inch larger per inch of major diameter of the screw. Additional information may be found in the *Machinery's Handbook* or other publications.

10-25. **Tap and Die Threading in the Lathe.** Threads that do not require the high degree of accuracy and finish obtained by chasing the threads can be cut by taps and dies. These methods are faster than chasing in the lathe.

10-26. *Tapping threads in a lathe.* Tapping can be done in the lathe by power or by hand. Regardless of the method, the hole must be drilled with the proper size of tap drill and chamfered. The

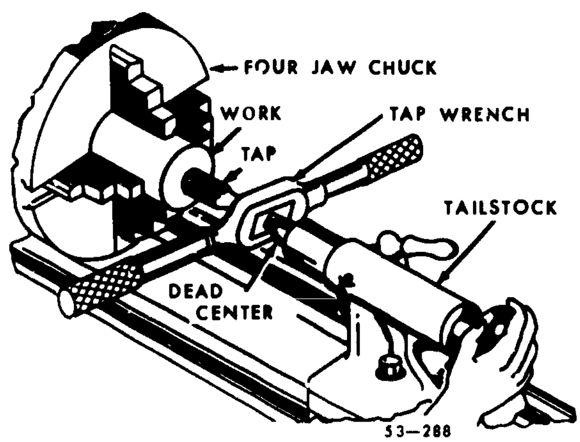


Figure 153. Tapping setup.

shank end of the tap is supported by the tailstock center. A slight pressure is maintained against the tap to keep its center hole on the center and to help the cutting teeth of the tap engage the work, as shown in figure 153.

10-27. The work rotates when you are tapping by power. Use a very slow spindle speed (10 rpm to 30 rpm) and plenty of lubricant for power tapping. The tap is prevented from rotating by a tap wrench; the handle of the wrench should contact the compound rest, as shown in figure 153. The use of power is not recommended for taps under ½ inch diameter or in tapping steel. Be sure that the tap wrench handle contacts the compound rest before you engage the spindle, and keep your fingers away from the tap wrench. Do not attempt to start the tap in the hole with the work revolving. Keep the center snug in the center hole to prevent the tap from becoming misalined and then breaking.

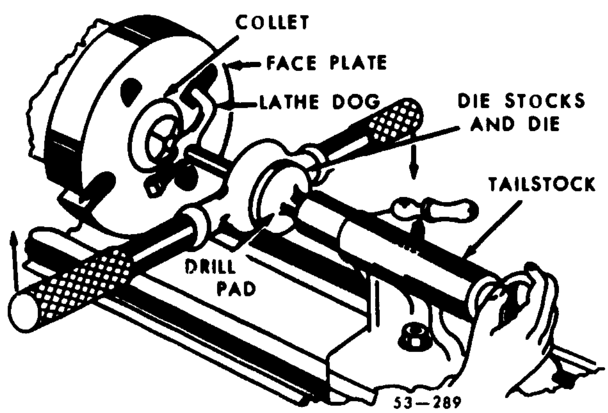


Figure 154. Die threading setup.

10-28. The setup for hand tapping in a lathe is similar to the power tapping setup and to tapping in a drill press. Set the lathe for a low spindle speed with the motor turned off. This helps to prevent the spindle from turning because of the cutting pressure. Lathe chucks can be positioned so that one of the chuck jaws contacts the compound rest to prevent the chuck from turning when large taps are used.

10-29. Whenever it is possible, work held in a collet should be set up as shown in figure 154. The lathe dog prevents the work from slipping in the collet. This method of securing the work can be used only with draw-bar collet attachments that permit the driveplate to be mounted while the collet attachment is being used.

10-30. An open-end wrench can be used in place of the tap wrench if the tap wrench is too long to turn. The center prevents the tap from being pulled "off center." The center does not need to support the tap after sufficient threads have been cut to insure proper alinement.

NOTE: Back off the tap frequently to break the chip, just as in tapping on a drill press, and use a lubricant

10-31. *Die threading in a lathe.* Die threading in a lathe is very similar to tapping in a lathe, except that the die is alined perpendicular to the work axis by pressure exerted against the back surface of the die, as shown in figure 154 The pressure can be applied by means of a drill pad as illustrated; by the front surface of the tailstock spindle; or, for very small dies, by the front surfaces of the drill chuck jaws. The jaws should be closed enough to insure that they are contacting the die, and not the diestock. Die threading can be done by power or by hand, using the same procedures as in tapping in a lathe. If the diestock handles do not clear the ways of the lathe, when you are threading by hand, remove the handles from the diestock. Install one of the handles in the diestock and rotate the diestock with the handle as far as possible. Remove the handle and install it in the opposite end of the diestock. Again, rotate the diestock as far as possible. Continue alternating the handle from end to end and turning the diestock until the threads are the length you want. Power can be used to remove the die from the threaded work by reversing the direction of the spindle rotation. If the lathe spindle is not reversible, place it in neutral and rotate the spindle by hand to remove the die from the work. It is difficult to cut very coarse threads with a die because

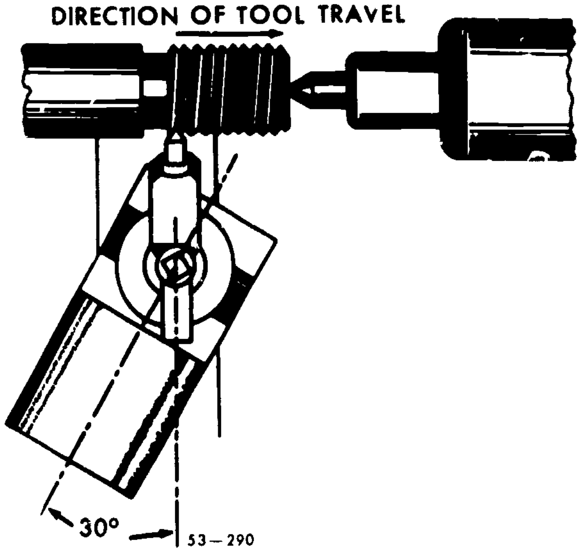


Figure 155. Setup for left-hand external threads.

of the great amount of force needed to turn the die. It is advisable to first rough out the threads with the die in the full open position; then perform a finish cut with the die closed down. Use a lubricant during both the cutting and removal operations.

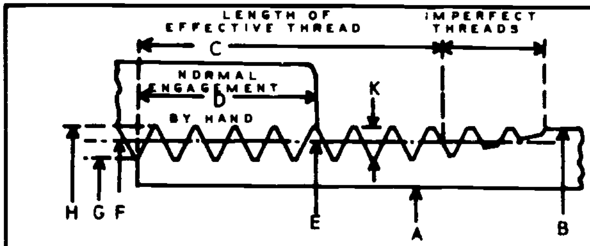
10-32. **Left-Hand Threads.** Threads may be either right-hand or left-hand. A thread is a *right-hand thread* if it is advanced by turning it in a clockwise direction. A thread is a *left-hand thread* if it winds in a counterclockwise and receding direction when it is viewed axially. Most threads are right-hand; therefore, a thread is considered to be right hand unless the symbol LH is used on drawings, taps, dies, etc. Left-hand threads are used when the direction of motion required is opposite to that obtained with a right-hand thread, such as crossfeed screws or the left end of turnbuckles; or when a slippage between a part and a nut would tend to loosen a right-hand nut, such as the right-hand end of an automobile axle.

10-33. Left-hand threads are cut in approximately the same manner as right-hand threads except that the carriage moves toward the tailstock instead of away from it. Also, you move the compound rest to the lett inside of the right to cut external left-hand threads, and to the right instead of the left to cut internal left-hand threads. Figure 155 shows the setup for machining external left-hand Unified threads. You grind the tool for left-hand threads with the relief and rake angles reversed from those on right-hand threading tools.

10-34. An *undercut*, or *groove*, is usually provided as a starting point for the left-hand threading tool. The undercut should be no narrower than the thread pitch and of a depth equal to, or slightly greater than, a single thread depth. The side nearest the thread should be chamfered.

10-35. **Multiple Threads.** A multiple thread is a combination of two or more threads, parallel to each other, progressing around the surface into which they are cut If you think of a single thread as taking the form of a helix; i.e., a string or cord wrapped around a cylinder, you may think of a multiple thread as several cords lying side by side and wrapped around a cylinder. There may be any number of threads, and they start at equally spaced intervals around a cylinder. Multiple threads are used when rapid movement of the nut or other attached threaded part is desired and when any weakening of the thread is to be avoided. For example, the hydraulic quick disconnects on aircraft use multiple threads. A single thread having the same lead as a multiple thread would be very deep compared to the multiple thread. The mating thread must also be a multiple thread. Figure 156 shows the difference in the lead

TABLE 5
TAPER PIPE THREAD DIMENSIONS



FOR ALL DIMENSIONS SEE CORRESPONDING REFERENCE LETTERS IN TABLE

ANGLE BETWEEN SIDES OF THREAD IS 60° TAPER OF THREAD, ON DIAMETER, IS 3/4 INCH PER FOOT

THE BASIC THREAD DEPTH IS 0 8 X PITCH OF THREAD AND THE CREST AND ROOT ARE TRUNCATED AN AMOUNT EQUAL TO 0 033 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			PITCH DIAMETER								
NOMINAL PIPE SIZE	OUTSIDE DIAMETER	NUMBER OF THREADS PER INCH	AT END OF EXTERNAL THREAD	AT END OF INTERNAL THREAD	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
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	1 02
1	1 315	11½	1 21363	1 23863	0.6828	0 400	0.3017	0 06957	0 08696	1 144	1 28

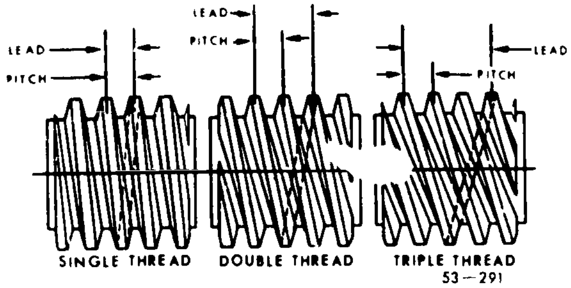


Figure 156. Comparison of single- and multiple-lead threads

of a single-, double-, and triple-lead thread having the same number of threads per inch. The lead of a multiple thread is equal to the pitch of the thread multipled by the number of separate thread leads.

10-36. The tool which is used to cut muitiple threads has the same shape as that of the thread to be cut. It is similar to the tool which is used for cutting a single-lead thread. However, greater side clearance is necessary because the helix angle of the thread increases with the increase in the multiple. The method of cutting multiple threads is about the same as for single-screw threads. The lathe must be geared to the number of single threads per inch, or with reference to the lead of the thread, and not to the pitch. The number of single threads per inch is found by dividing the number of threads per inch by the multiple (number of leads) of the thread. For example, the number of single threads per inch of a double-lead thread having 8 threads per inch is 4. The lathe would be set up to machine 4 threads per inch

10-37. **Pipe Threads.** American Standard Taper Pipe Threads (NPT) are similar in form to the Unified thread; a common feature is that both have an included angle of 60°. Part of the threaded portion of the pipe is machined to a taper of ¾ inch per foot, and the threads are also machined at the same taper. The taper permits a tighter connection and seal between the mating threads than can be obtained with a straight thread. The nominal pipe size is the approximate inside diameter of the pipe. The actual outside diameter (the major diameter) can be found by measuring the pipe or by consulting a machinist publication. Table 5 shows a typical page from such a reference.

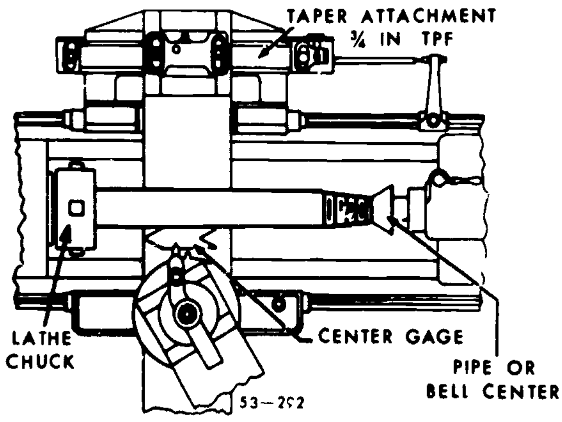


Figure 157. Pipe threading setup.

10-38. The hole for tapered internal threads is straight bored to a diameter equal to, or slightly larger than, the minor diameter of the small end of the pipe. The taper length of both internal and external tapered pipe threads is equal to the length of the normal hand engagement. The thread length is equal to the length of the effective threads plus the length of the imperfect threads. The root of the imperfect threads will be the shape of the threading tool when the threads are cut on a lathe. The illustration in table 5 shows how the thread root would appear if it were cut by a die.

10-39. The most common setup for chasing tapered pipe threads on a pipe is shown in figure 157. but any suitable work setup can be used. Note, however, that the threading tool is positioned with the center gage alined with the straight portion of the pipe and *not* with the taper. The tool setup is identical to the setup for machining Unified threads. The only difference is that the taper attachment is used when tapered pipe threads are machined.

10-40. If the taper attachment requires the cross-slide to be clamped so that it cannot be moved, the threading tool must be withdrawn from the thread groove at the end of the threading cut by means of the compound rest. The spindle speed should be slow enough to insure adequate time to withdraw the tool to prevent damaging the work or the tool.

10-41. The pitch diameter is difficult to measure on tapered threads because of the taper; therefore, the threads are normally machined to fit a taper thread gage or the mating part. The threaded parts should be assembled by hand to the distance equal to the normal engagement by hand, as given in table 5.

11. Machining Special Threads

11-1. You will occasionally be required to cut Acme, square, and buttress threads. While not as common as the Unified thread, they are often used for certain applications. The following information will help you to identify and to machine these types of threads.

11-2. **Acme Threads.** The Acme thread is a heavy-duty thread that is quite often used on feed screws, vises, and similar items. Acme threads are rapidly replacing square threads which were used

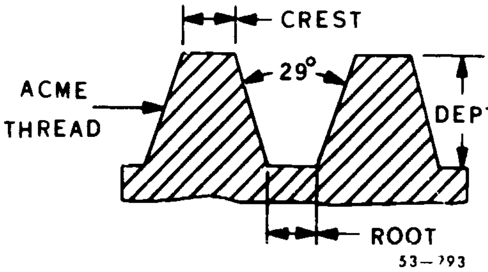


Figure 158. Acme thread.

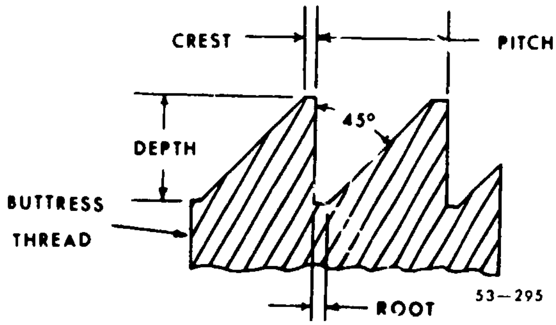
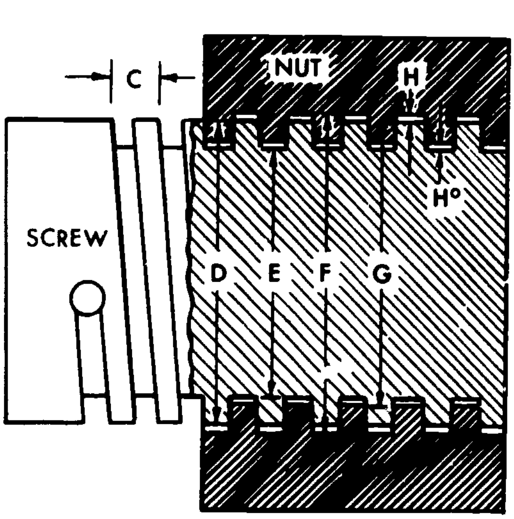


Figure 160 Buttress thread.

extensively at one time because they are easier to machine than square threads. Split nuts having Acme threads can be engaged and disengaged more readily than square-threaded nuts. Acme threads have an included angle of 29°, as shown in figure 158. Split nuts having Acme threads can be adjusted for wear, and backlash can be eliminated by adjusting the nut to seat deeper in the thread groove. This is possible because of the tapered sides of the thread.

11-3. The single depth of General-Purpose Acme threads is equal to one-half the pitch plus an allowance for clearance. The clearance allowance is 0.010 inch for threads of 10 threads per inch and coarser, and 0.005 inch for finer threads. The basic pitch diameter of general-purpose Acme threads is equal to the nominal diameter minus one-half the pitch.

11-4. Stub Acme threads are used when a coarse thread having a shallower depth than the General-Purpose thread is desired. The single depth on a Stub Acme thread equals 0.3 times the pitch, plus the same clearance allowance, of General-Purpose Acme threads. The basic pitch diameter of Stub Acme threads equals the nominal diameter minus 0.3 times the pitch. The exact dimensions of Acme threads are found in reference sources, such as the *Machinery's Handbook*.



C—PITCH
D MAJOR DIAMETER OF SCREW THREAD (BASIC SIZE)
E—MINOR DIAMETER OF SCREW THREAD (BASIC SIZE)
F—MAJOR DIAMETER OF NUT THREAD (D+CLEARANCE)
G—MINOR DIAMETER OF NUT THREAD (E+CLEARANCE)
H AND H°—CLEARANCE 53—294

Figure 159. Sectional view of square thread and nut.

11-5. **Square Threads.** A square thread is a thread whose sides are parallel. The depth of the thread is equal to the width of the thread groove; this space is, theoretically, equal to one-half the pitch. It is necessary, in practice, however, to make the space in the internal threads slightly wider than the thread to permit a sliding fit. External square threads are made exactly according to the theoretical standard. Figure 159 illustrates the terms used with reference to square threads.

11-6. **Buttress Threads.** On a buttress thread the load-resisting side is at a right angle to the thread axis, as shown in figure 160. The load-resisting side or face of the thread is referred to as the *pressure* flank. The pressure flank is often machined at a slight angle to avoid machining difficulties. The American Standard buttress thread pressure flank is machined at an angle of 7° to the perpendicular. Buttress threads are used primarily where the pressure or load is in one direction, such as the screw in a vise.

11-7. Special threads are measured by using wires and a micrometer, a gage, or a mating part. Detailed information pertaining to these threads can be found in machinists' publications, such as the *Machinery's Handbook*.

Special Lathe Operations and Attachments

IN ADDITION to the various operations and attachments which we have already discussed, other lathe operations and attachments make it possible for you to perform a still wider range of work on the lathe. After discussing the filing and polishing, knurling, and spring winding operations, we will cover the center rest, follower rest, micrometer carriage stop, and toolpost grinder attachments. We will conclude the chapter with a discussion of gear blank calculation.

12. Special Lathe Operations

12-1. There are several lathe operations that you must be able to perform in which you do not use a cutting tool. These operations include filing and polishing, knurling, and spring winding. We will discuss filing and polishing—two closely related operations—first.

12-2. **Filing and Polishing.** Filing and polishing are both finishing operations. They are used primarily to improve the finish of the work or to reduce the size of the work slightly.

12-3. *Filing.* When you file work which is revolving in the lathe, hold the tip of the file at an angle of approximately 10° toward the tailstock end of the lathe. Pass the file slowly over the revolving work so that the work turns several revolutions before the stroke is completed. Exert less pressure than for ordinary bench filing, since only a small area of the file and work are in contact when you are filing round work. Release the pressure on the return stroke without lifting the file from the work. Clean the teeth of the file frequently. This prevents the metal that lodges between the teeth of the file from scratching the work or impairing the cutting action of the file. Figure 161 illustrates how to hold the file and how to apply it to the work.

CAUTION: Keep your hands and arms clear of the chuck or other work-driving device to avoid injury. NEVER USE A FILE WITHOUT A FILE HANDLE.

12-4. You will usually use a mill file when you file work in a lathe. Use a bastard cut mill file for rough filing; use a second cut mill file on work requiring less stock removal and a finer finish. Use other types of files, such as round and half-round files, to file radii, fillets, and curved surfaces.

12-5. The spindle speed for filing metals is four to five times faster than the rough turning speed. If the speed is too slow, there is danger of filing the work out of round. If the speed is too fast, the file has a tendency to slide over the work. This causes the file to dull rapidly and glaze the work. The amount of stock left for filing is 0.002 inch to 0.005 inch.

12-6. *Polishing.* Polishing the work with abrasive cloth is done primarily to improve the finish. Since polishing also removes a very small amount of material, it can also be used to accurately fit mating parts, such as shafts and bushings. Usually, 0.00025 inch to 0.0005 inch is allowed for polishing. Use successively finer grades of abrasive cloth to produce a very fine finish. Use abrasive cloth for ferrous metals and sandpaper for nonferrous metals.

12-7. The spindle speeds for polishing are very high. The recommended surface foot speed is 5000 feet per minute. A great deal of heat is generated because of the high speed and the friction of the abrasive cloth against the work. The work expands rapidly because of the heat. To counteract this, you should lubricate and adjust the tailstock center frequently when you are polishing between centers. To determine the spindle speed, use the following formula:

$$\text{Rpm} = \frac{4 \times 5000}{\text{work diameter}}$$

12-8. Figure 162 shows how to hold the abrasive strip when you are polishing. Note that the ends of the strip are separated. This prevents the strip from grabbing and winding around the work, which could pull your hand around with it. Move the abrasive strip slowly back and forth along the work to prevent the formation of polishing rings on the work surface. Polishing rings are a series of

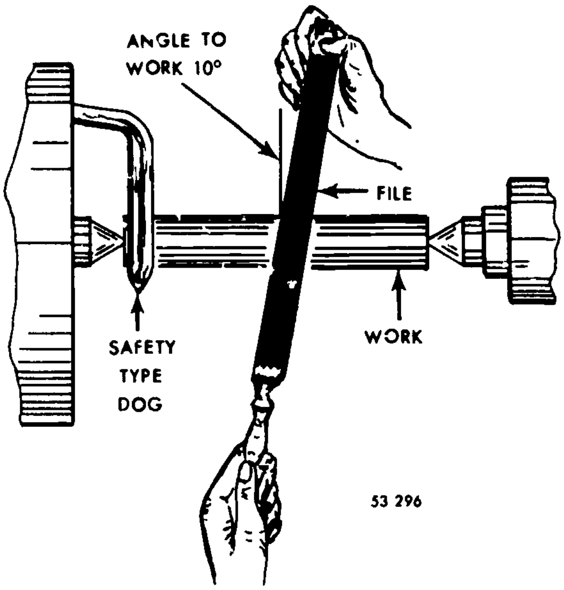


Figure 161. Filing work on a lathe.

closely spaced parallel blemishes in the finish around the circumference of the work. The blemishes are burned and scratched areas caused by the loading of the abrasive strip and the trapping of coarse abrasive particles between the work and the abrasive strip. To produce a bright surface, polish the work dry. To produce a dull satin finish, apply oil as you polish.

12-9. **Knurling.** Knurling is the process of rolling or squeezing impressions into the work with hardened steel rollers that have teeth milled in their faces. Primarily, knurling provides a gripping surface on the work; it is also used for decoration. Knurling increases the diameter of the work slightly.

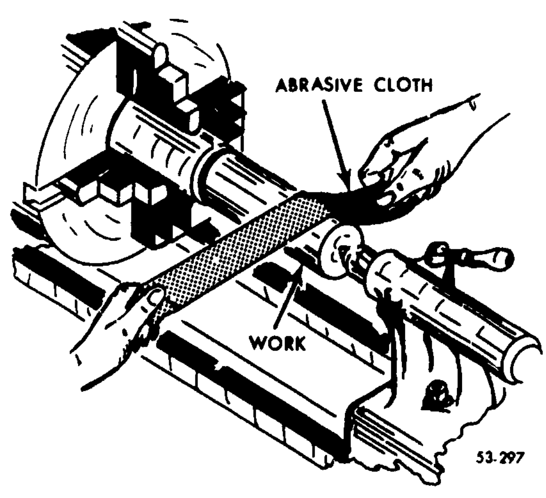


Figure 162. Polishing work on a lathe.

12-10. *Knurling tools.* The three common types of knurling tools are the *knuckle joint,* shown in figure 163,A; the *revolving head,* shown in figure 163,B; and the *straddle,* shown in figure 163,C. The revolving head type has three sets of rolls each with a different pitch, whereas the knuckle and straddle types each have one set of rolls. The straddle type of knurling tool is used primarily to knurl small diameter work. The work revolves between the two rolls and is not distorted because the pressure of one roll counteracts the other.

12-11. There are two patterns of knurls: *diamond* and *straight line;* and three pitches: *fine, medium,* and *course* in each pattern, as shown in figure 164. The diamond is the most common pattern, and the medium is the most common pitch. The coarse pitch is used for large diameter work, and the fine pitch is used for small diameter work.

12-12. *Knurling setup.* The knurling tool is positioned with the faces of the rolls parallel to the work surface to be knurled. The knurling tool is set at center height with the work and the upper and lower rolls equally spaced above and below the work axis, as shown in figure 165. The spindle speed should be approximately half the rough turning speed. It should not exceed the highest speed permitted for shifting the feed reverse mechanism on the lathe. The feed should be between 0.015 and 0.025 inch. The work center holes should be as large as practical to provide as much bearing surface as possible to absorb the pressure of the knurling tool. Work mounted in a chuck should also be supported at the tailstock end with a center to prevent damaging the work or destroying the accuracy of the chuck.

12-13. *Knurling operation.* There are two types of knurling operations: ordinary knurling and knurling between layout lines.

12-14. To perform ordinary knurling, first set up the work and lay out the length of the portion to be knurled. Then swivel the compound perpendicular to the axis of the work, mount the knurling tool in the tool post, and set the lathe for the correct spindle speed and feed. Check and lubricate the knurling tool to insure that the rollers and the revolving head move freely. Apply a cutting lubricant to the work and the knurling tool generously during the knurling operation. Apply the lubricant to the surface of the work opposite the knurling tool with an oilcan (keep the applicator well away from the knurling tool rollers) or with a brush. Keep rags, brushes, and your fingers away from the knurling tool operation.

12-15. Position the carriage so that a third to a half of the face of the rollers extends beyond the end of the work. This reduces the pressure required to start the knurl impression. Force the

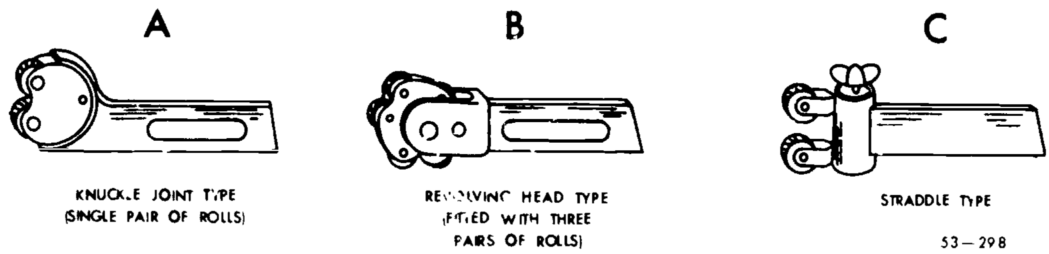


Figure 163. Types of knurling tools.

knurling rollers into contact with the work and engage the spindle clutch. Check the knurl to see if the rollers are tracking properly, as shown in figure 166. This involves disengaging the clutch after the work has revolved three or four times and backing the knurling tool away from the work with the cross-slide. If a double impression has occurred, as shown in figure 166, move the carriage to a new location and repeat the operation.

12-16. You use a slightly different method to start the knurling operation in knurling between layout lines. You set up the work, lathe, and knurling tool the same way as for ordinary knurling, except that the face of the rollers is offset about 5°. This enables the corners of the rollers to contact the work first. This reduces the pressure required to force the rollers into the work.

12-17. To knurl between layout lines, swing the compound to the right 5°. Position the knurling tool slightly the the left of the right-hand layout line and start the knurl by forcing the corner of the rollers into the work surface. If the knurl is tracking properly, move the carriage by hand to extend the knurl to the right-hand layout line. Back the knurling tool away from the work, and position the rollers parallel to the surface of the work. Pick up the knurl by meshing the rollers with the started impression, and knurl to a point just to the right of the left-hand layout line. Mesh the rollers into the existing knurl pattern and force the corner of the rollers into the work. Using hand feed, knurl to the left-hand layout line, then back off the knurling tool.

12-18. Regardless of the method of knurling, do not allow the work to rotate if the knurling tool is contacting it and the carriage travel is stopped, or rings will form, as shown in figure 167. Do not stop the work without relieving the knurling tool pressure. The pressure may distort or spring the work. Remember to keep the knurling tool and the work well lubricated throughout the operation. Check the adjustment of the tailstock center frequently The pressure of the knurling operation may cause the centers to loosen slightly. If the knurl is correctly formed, engage the spindle clutch and carriage feed and move the knurling rollers into contact with the knurled impressions. The rollers will aline themselves. Using the carriage feed, allow the tool to knurl to within $\frac{1}{32}$ inch of the layout line. Disengage the carriage feed and, with the work revolving, feed the carriage by hand until the knurl touches the layout line Force the knurling tool slightly deeper into the work. Re-

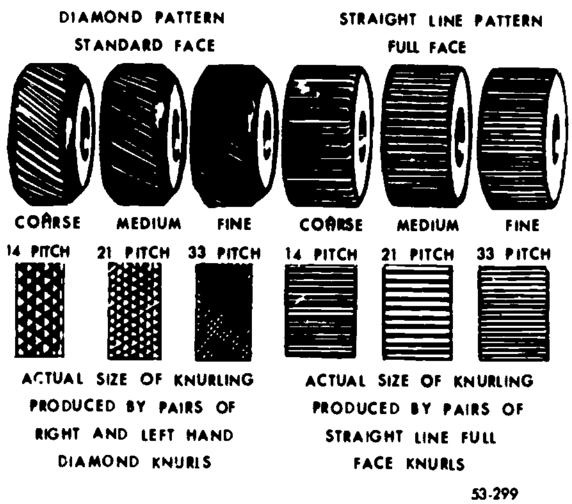


Figure 164. Knurl patterns.

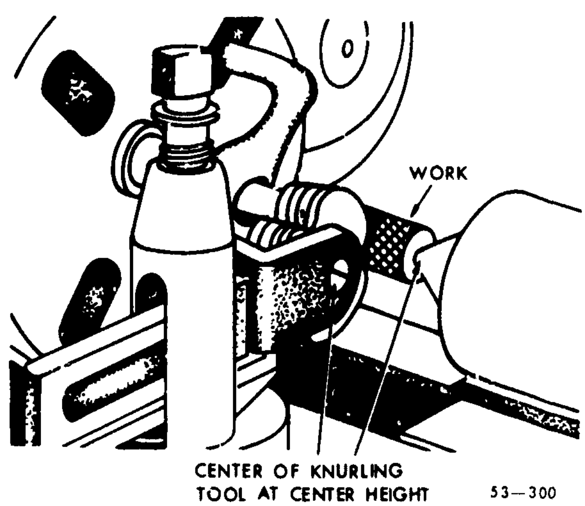


Figure 165. Knurling setup.

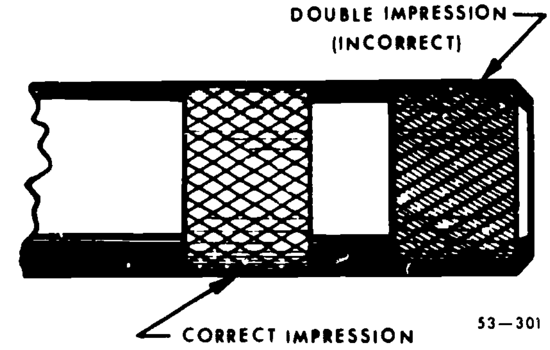


Figure 166. Knurled impressions.

verse the direction of the carriage feed screw, and engage the carriage feed. Allow the knurling tool to feed to the right until approximately half of the rollers extend beyond the end of the work. Never allow the knurling tool to feed entirely off the end of the work. If necessary, repeat the knurling operation until the diamond-shaped impressions converge to a point. If you make additional passes after the pattern has completely formed, you may strip the points away from the surface. Now move the knurling tool away from the work. With the work revolving, clean the knurled work surface with a file brush. Remove any burrs, which may have formed on the tips of the diamond-shaped impressions, by increasing the spindle speed and lightly filing the knurl.

12-19. **Spring Winding.** Both tension and compression coil springs can be wound on a lathe. A *tension spring,* figure 168,A, is one in which the coils lie one against the other, as in a screen door spring. When tension is exerted on the spring, the coils spread apart, and the tendency of the spring, is to pull back to its original form. A *compression spring,* figure 168,B, is one in which the coils are wound a definite distance apart with a fixed space between them. The valve springs of automobiles are compression springs. When force is exerted on the spring, it is compressed, and the coils have a tendency to push or spring back to their original spacing.

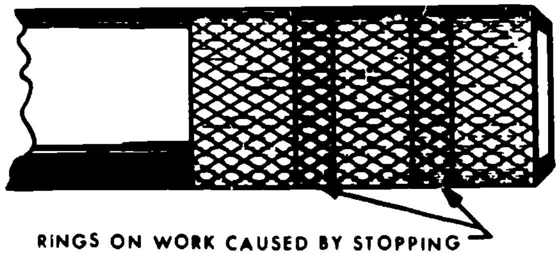


Figure 167. Rings on a knurled surface.

12-20. The size of springs wound on the lathe is controlled by using mandrels of the proper size. Spring wire of the required material and diameter is used. Pass the wire between wood or brass blocks or through a spring wire holder that is held in the toolpost. You can clamp the end of the wire between the chuck jaw and the mandrel. Also, you can drill a hole through the mandrel a few thousandths larger than the wire diameter, and insert the wire into it. The tailstock center supports the other end of the mandrel. Clamp the wire between the blocks or in the wire holder just tight enough to keep it from slipping and still hold a uniform tension on the wire so that it is wound tightly against the mandrel.

12-21. The carriage feed is equal to the diameter of the spring wire in winding a tension spring. It is equal to desired spacing in winding a compression spring. The split nut and lead screw can also be used to move the carriage to obtain the desired spacing. When the spring is wound to the desired length, relieve the tension, clip off the wire, and remove the spring from the mandrel. Information pertaining to the diameter of the spring wire gages which are used by the various manufacturers can be found in reference sources, such as the *Machinery's Handbook.*

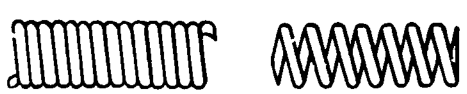


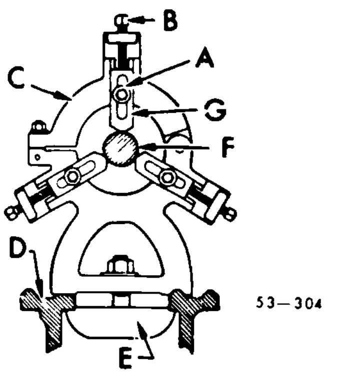
Figure 168. Coil springs.

13. Lathe Attachments

13-1. The center rest, the follower rest, the micrometer carriage stop, and the toolpost grinder are lathe attachments. The variety of work that you can perform on a lathe is increased by the use of attachments.

13-2. **Center Rest.** The center rest consists of a frame and three adjustable jaws which support the work, as shown in figure 169. One purpose of the center rest is to prevent springing or deflection of slender, flexible work; another is to furnish auxiliary support for the work to permit heavy cuts to be made; and a third is to support work for drilling, boring, or internal threading. The overarm containing the top jaw can be unfastened and swung out of the way so that identical pieces can be removed and replaced without adjusting the jaws.

13-3. A bearing surface must be provided for the center rest jaws. The bearing surface is usually



A LOCK SCREW
B. ADJUSTING SCREW
C. OVERARM
D. LATHE WAYS
E. CLAMP
F WORK
G ADJUSTABLE JAW

Figure 169. Center rest.

machined directly on the work, as shown in figure 170. When the work is too small in diameter to machine the bearing surface or shaped so that it would be impractical to machine one, you can use a cathead to provide the bearing surface. The cathead, shown in figure 171, has a bearing surface, a hole through which the work extends, and adjusting screws The adjusting screws fasten the cathead to the work. They are also used to aline the bearing surface so that it is concentric to the work axis. Use a dial indicator to insure concentricity.

13-4. To set up the center rest and reduce friction, you should first polish the portion of the work that is to be used as the bearing surface. Then clean the portion of the ways of the lathe that the center rest will be placed on. Obtain a clean sheet of paper and cut off pieces large enough to be placed between the center rest legs and the ways of the lathe. Position and clamp the center rest so that the adjustable jaws are positioned over the bearing surface of the work, as shown in figure 170. With the work revolving very slowly, loosen the adjustable jaw screw, as shown in figure 169, and adjust the jaws toward the

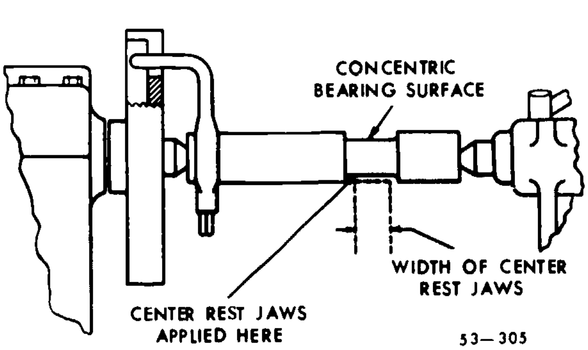


Figure 170. Concentric bearing surface.

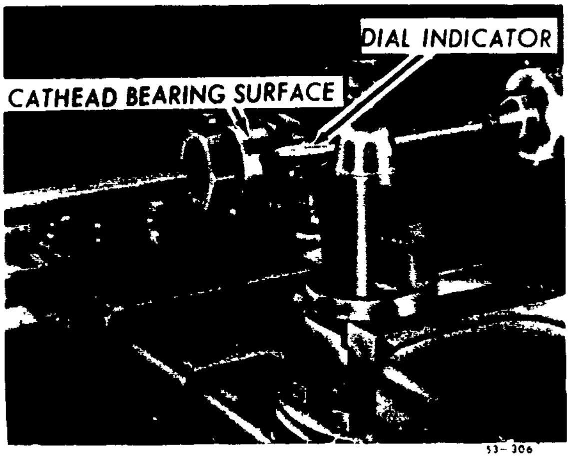


Figure 171. Cathead.

work. Continue to adjust until a slight drag is felt on a strip of paper that has been inserted between the jaws and the work. Then, lightly tighten the adjustable jaw lock screws. Place a piece of paper between the jaws and the work and adjust the jaws until you feel a drag on the paper. The clearance should be .001 inch. Now fully tighten the lock screws. Lubricate the center rest jaws and bearing surface with a mixture of white lead and lubricating oil. Rotate the work slowly, and if any of the jaws rub the lubricating mixture off, readjust the jaws. Then proceed with the machining operation. Keep the jaws well lubricated to prevent scoring the work surface. Copper shims can be fastened over the bearing ends of the adjustable jaws to protect the finish on highly polished work.

13-5. When it is undesirable to hold the work in a chuck, you can machine it with one end supported by the headstock center and the other end supported by a center rest. Use a leather strap or rawhide thong to tie the work to the driveplate and

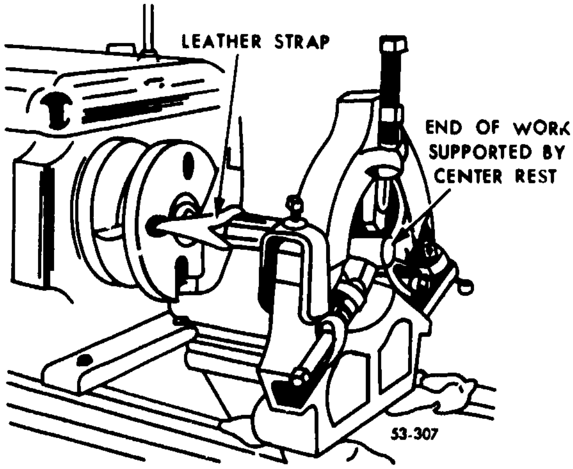


Figure 172. Tying the lathe dog.

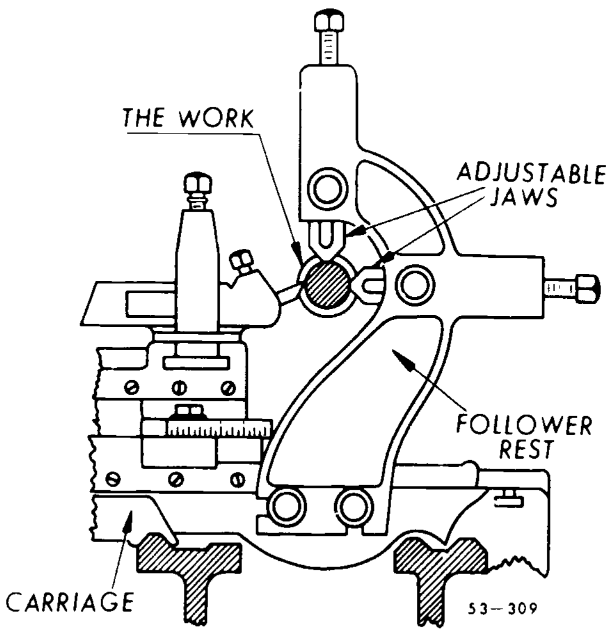


Figure 173. Follower rest.

to prevent it from moving off the headstock center, as shown in figure 172. Mount the work between centers and machine the bearing surface. Set up the center rest. With the work mounted between centers, tie the lathe dog, as shown in figure 172. Then remove the tailstock center and perform the necessary machining.

13-6. **Follower Rest.** Long slender shafts that tend to whip and spring while they are being machined require the use of a follower rest. The follower rest is fastened to the carriage and moves with the cutting tool. The upper jaw prevents the work from climbing the cutting tool. The lower jaw prevents the work from springing away from the cutting tool, as shown in figure 173. The follower rest jaws are adjusted in the same manner as center rest jaws. The follower rest is often used when long, flexible shafts are threaded, as shown in figure 174. At the completion of each threading cut, remove any burrs that may have formed, to prevent them from causing the work to move out of alinement.

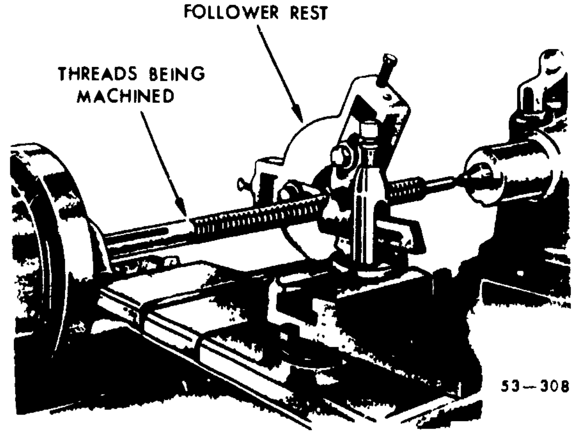


Figure 174. Follower rest used while threading.

13-7. **Micrometer Carriage Stop.** The micrometer carriage stop, shown in figure 175, is used to accurately position the lathe carriage. Move the carriage so that the cutting tool is approximately positioned. Clamp the micrometer carriage stop to the ways of the lathe, with the spindle in contact with the carriage. The spindle of the micrometer carriage stop can be extended or retracted by means of the knurled adjusting collar. The graduations on the collar, which indicate movement in

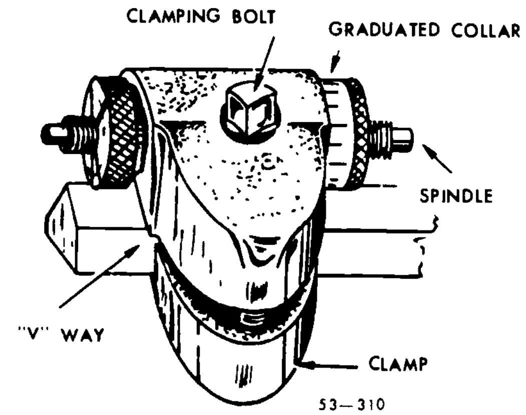
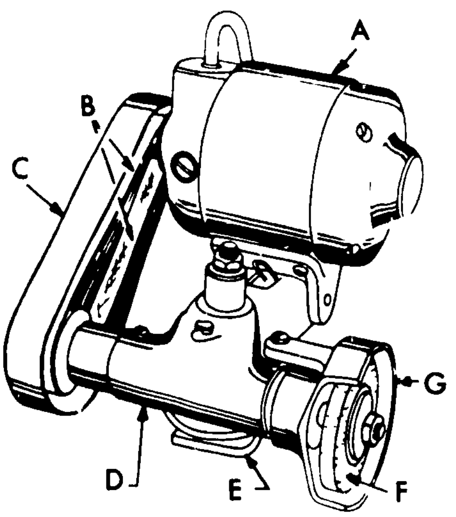


Figure 175. Micrometer carriage stop.

thousandths of an inch, make it possible to set the spindle accurately. Next, bring the carriage in contact with the micrometer spindle again. The carriage can be accurately positioned within 0.001 of an inch. This is very useful when you are facing work to length, machining shoulders to an exact length, or accurately spacing internal and external grooves. After making a cut, bring the tool back to the start of the cut by means of the carriage stop. This feature is very useful when you must remove a tool, such as the internal recessing tool, from the hole to take measurements and then reposition it to take additional cuts. Always bring the carriage into contact with the stop by hand. Use power feed to bring the carriage within $\frac{1}{32}$ inch of the stop. Move the carriage by hand the remaining distance.

13-8. **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



A MOTOR
B BELT
C BELT GUARD
D SPINDLE
E CLAMP
F. GRINDING WHEEL
G WHEEL GUARD

Figure 176 Toolpost grinder

176 shows a typical toolpost grinder. The grinder must be set on center, as shown in figure 177. 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 operations that a cutting tool is capable of performing. 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.

13-9. 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

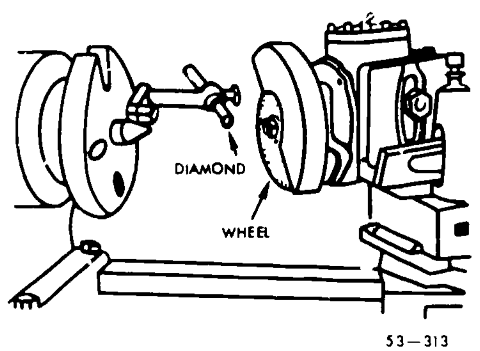


Figure 178 Diamond wheel dresser clamped to a faceplate.

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 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.

13-10. The grinding wheel must be dressed and trued Use a diamond wheel dresser to dress and true the wheel. The dresser is held in a holder that is clamped to the driveplate, as shown in figure 178. 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 179. 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). Remove the diamond dresser holder as soon as the dressing operation is completed. Bring the grinding wheel in contact with the diamond by carefully

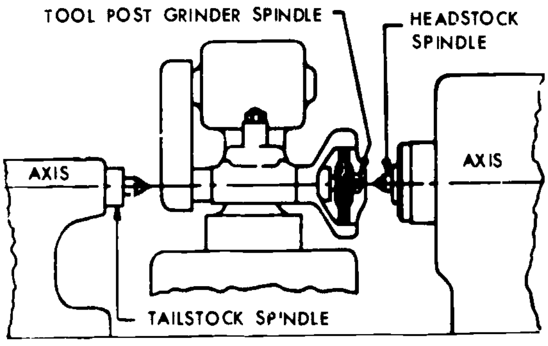


Figure 177 Mounting the grinder at center height.

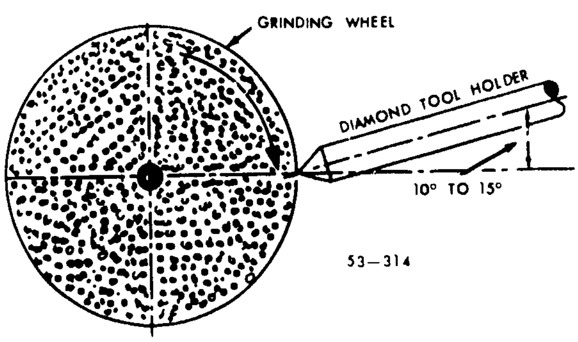


Figure 179. Position of diamond.

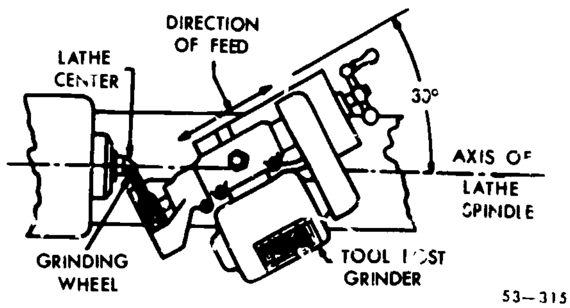


Figure 180. Grinding a lathe center.

feeding the cross-slide in by hand. Move the wheel clear of the diamond and make a cut by means of the cross slide. The maximum depth of cut is 0.002 inch. Move the wheel slowly by hand back and forth over the point of the diamond. Move the carriage if the face of the wheel is parallel to the ways of the lathe. Move the compound rest if the face of the wheel is at an angle. Make the final depth of cut of 0.0005 inch with a slow, even feed to obtain a good wheel finish.

13-11. Rotate the work at a fairly low speed during the grinding operation. The recommended surface foot speed is 60 to 100 feet per minute (fpm). The depth of cut depends upon the hardness of the work, the type of grinding wheel, and the desired finish. Avoid taking grinding cuts deeper than 0.002 inch until you gain experience. Use a fairly low rate of feed. You will soon be able to judge whether the feed should be increased or decreased. Never stop the rotation of the work or the grinding wheel while they are in contact with each other.

13-12. Toolpost grinders are often used to refinish damaged lathe centers. If the lathe is to be used for turning between centers in the near future, grind the tailstock center first, then the headstock center. Leave the headstock center in position for the turning operation. This method provides the greatest degree of accuracy. If you must remove the headstock center in order to perform other operations, marks placed on the headstock center, the sleeve, and the center will enable you to install them in the same position they were in when the center was ground. This will insure the greatest degree of accuracy for future operations involving turning work between centers.

13-13. To refinish a damaged lathe center, you should first install headstock and tailstock centers after insuring that the spindle holes, drill sleeves, and centers are clean and free of burrs. Next, position the compound rest parallel to the ways; there mount the toolpost grinder on the compound rest. Make sure that the grinding wheel spindle is at center height and alined with the lathe centers. Move the compound rest 30° to the right of the lathe spindle axis, as shown in figure 180. Mount the wheel dresser, covering the ways and carriage with rags to protect them from abrasive particles. Wear goggles to protect your eyes.

13-14. Start the grinding motor, by alternately turning it on and off, but letting it run a bit longer each time, until the abrasive wheel is brought up to top speed. Dress the wheel, feeding the grinder with the compound rest. Then move the grinder clear of the headstock center and remove the wheel dresser. Set the lathe for the desired spindle speed, and engage the spindle. Pick up the surface of the center. Take a light depth of cut and feed the grinder back and forth with the compound rest. Do not allow the abrasive wheel to feed entirely off the center. Continue taking additional cuts until the center cleans up. To produce a good finish, reduce the feed rate and the depth of cut to .0005. Grind off the center's sharp point, leaving a flat with a diameter about $\frac{1}{32}$ inch. Move the grinder clear of the headstock and turn it off.

Bibliography

Books

ARMSTRONG, W. H. *Machine Tools for Metal Cutting*. New York: McGraw-Hill Book Co., Inc., 1957.

DOYLE, Lawrence E. *Metal Machining*. New York: Printice-Hall Inc., 1953.

LEGRAND, Rupert (Editor). *The New American Machinist's Handbook*. New York: McGraw-Hill Book Co., Inc., 1955.

OBERG, Erik, and Franklin D. Jones. *Machinery's Handbook*, 18th Edition. New York: The Industrial Press, 1968.

Department of the Air Force Publications

AFM 127–101, *Accident Prevention Handbook*, 26 June 1970.

TO 32–1–151, *Hand, Measuring, and Power Tools*, 19 April 1965.

TO 34–1–10, *Fundementals of Ordnance Corps Machine Tools*, 2 February 1959.

TO 34C2–8–6–62, *Operation and Maintenance, Band Saw Machines, Vertical Metal Cutting (Continental Model Do All ML)*, 2 July 1954.

Resident Course

Training Literature from Block II, ABR53130, Machinist.

Training Literature from Block III, ABR53130, Machinist.

Training Literature from Block IV, ABR53130, Machinist.

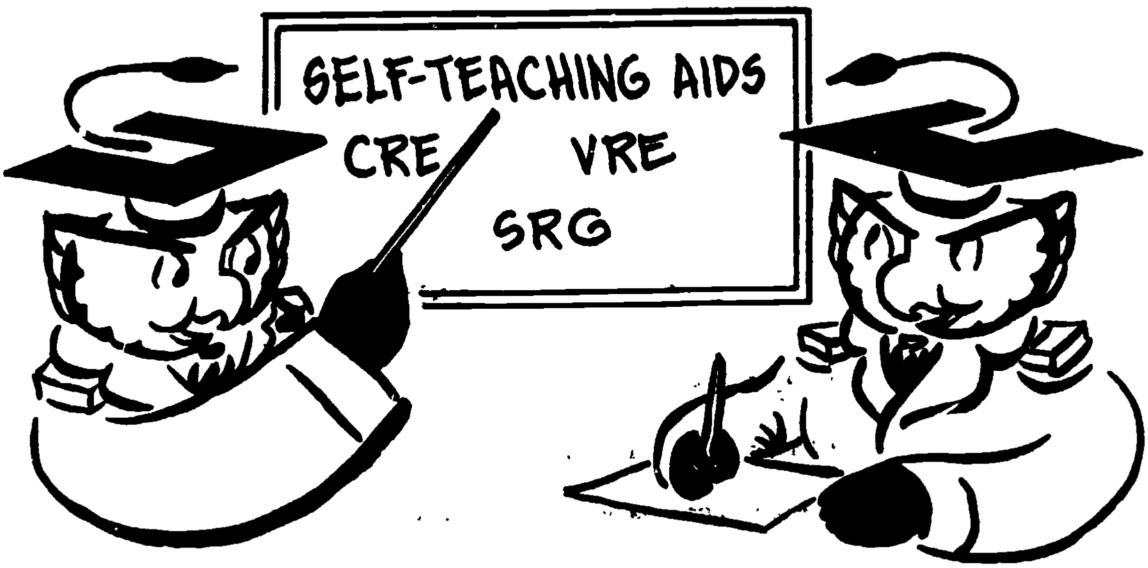
Training Literature from Block VI, ABR53130, Machinist.

NOTE: None of the items listed in the bibliography above are available through ECI. If you cannot borrow them from local sources, such as your base library or local library, you may request one item at a time on a loan basis from the AU Library, Maxwell AFB, Alabama, ATTN: ECI Bibliographic Assistant. However, the AU Library generally lends only *books* and a limited number of *AFM's*. TO's, classified publications, and other types of publications are *not* available. (For complete procedures and restrictions on borrowing materials from the AU Library, see the latest edition of the *ECI Catalog*.)

AU GAFS, ALA (812795)500

53130 02 21
WORKBOOK

Lathe Work



This wc-' ok places the materials you need *where* you need them while you are studying. In it, you will find the Study Reference Guide, the Chapter Review Exercises and their answers, and the Volume Review Exercise. You can easily compare textual references with chapter exercise items without flipping pages back and forth in your text. You will not misplace any one of these essential study materials. You will have a single reference pamphlet in the proper sequence for learning.

These devices in your workbook are autoinstructional aids. They take the place of the teacher who would be directing your progress if you were in a classroom. The workbook puts these self-teachers into one booklet. If you will follow the study plan given in "Your Key to Career Development," which is in your course packet, you will be leading yourself by easily learned steps to mastery of your text.

If you have any questions which you cannot answer by referring to "Your Key to Career Development" or your course material, use ECI Form 17, "Student Request for Assistance," identify yourself and your inquiry fully and send it to ECI.

Keep the rest of this workbook in your files. Do not return any other part of it to ECI.

EXTENSION COURSE INSTITUTE

Air University

TABLE OF CONTENTS

Study Reference Guide

Chapter Review Exercises

Answers For Chapter Review Exercises

Volume Review Exercise

ECI Form No. 17

STUDY REFERENCE GUIDE

1. *Use this Guide as a Study Aid.* It emphasizes all important study areas of this course.

2. *Use the Guide as you complete the Volume Review Exercise and for Review after Feedback on the Results.* After each item number on your VRE is a three digit number in parenthesis. That number corresponds to the Guide Number in this Study Reference Guide which shows you where the answer to that VRE item can be found in the text. When answering the items in your VRE, refer to the areas in the text indicated by these Guide Numbers. The VRE results will be sent to you on a postcard which will list the *actual VRE items you missed.* Go to your VRE booklet and locate the Guide Number for each item missed. List these Guide Numbers. Then go back to your textbook and carefully review the areas covered by these Guide Numbers. Review the entire VRE again before you take the closed-book Course Examination.

3. *Use the Guide for Follow-up after you complete the Course Examination.* The CE results will be sent to you on a postcard, which will indicate "Satisfactory" or "Unsatisfactory" completion. The card will list *Guide Numbers* relating to the questions missed. Locate these numbers in the Guide and draw a line under the Guide Number, topic, and reference. Review these areas to insure your mastery of the course.

Guide Number	*Guide Numbers 200 through 214*
200	Introduction; Construction and Operation; pages 1-6
201	Cutting Tools and Holders; pages 6-12
202	Workholding Devices; Lathe Chucks, Faceplate, Driveplate; pages 12-19
203	Workholding Devices; Mandrels, Mounting Workholding Devices, Dismounting Workholding Devices, Lathe Centers; pages 19-25
204	Introduction; Facing and Straight Turning; pages 26-29
205	Machining Shoulders, Corners, Undercuts, and Grooves; pages 26-32
206	Radii and Form Turning, Taper Turning; pages 32-40
207	Drilling, Boring, Reaming, and Recessing; Drilling, Boring; pages 40-47
208	Drilling, Boring, Reaming, and Recessing; Reaming, Recessing; pages 47-48
209	Introduction; Threads and Thread Measurement; pages 49-54
210	Machining American Standard and Unified Threads; Machining External Threads; pages 54-59
211	Machining American Standard and Unified Threads; Machining Internal Threads, Tap and Die Threading in the Lathe, Left Hand Threads, Multiple Threads, Pipe Threads; pages 59-63
212	Machining Special Threads; pages 63-64
213	Introduction; Special Lathe Operations; pages 65-68
214	Lathe Attachments; pages 68-7?

CHAPTER REVIEW EXERCISES

The following exercises are study aids. Write your answers in pencil in the space provided after each exercise. Immediately after completing each set of exercises, check your responses against the answers for that set. Do not submit your answers to ECI for grading.

CHAPTER 1

Objectives: To understand the construction and operation of a lathe. To be able to identify the various types of lathe cutting tools and toolholders. To recognize the various lathe workholding devices and their uses.

1. What are the three classes of lathes? (1-3)

2. How is the size of a lathe designated? (1-4)

3. What are the ways of a lathe? (1-6)

4. Which type of taper is machined in the spindle nose on most lathes? (1-7)

5. What are some of the purposes of the tailstock? (1-8)

6. What is the primary function of the carriage, and what parts make up the assembly? (1-9)

7. What are the three groups of control levers? (1-12)

8. What are the two groups of spindle controls? (1-13)

9. What determines the ratio of the lead screw rpm to the spindle rpm? (1-17)

10. What does speed refer to in lathe work? (1-24)

11 What CFS value is used to compute rough turning? (1-24)

12 What does feed refer to in lathe work? (1-25)

13. What is depth of cut? (1-26)

14. What is the main objection to using high-carbon steel cutting tools? (2-3)

15. What is red hardness? (2-4)

16. What are the main differences between carbide and high-speed steel tools? (2-5)

17. What is tool geometry? (2-7)

18 What angles are affected by the angle at which a toolholder holds the tool? (2-8, 9)

19 Why is a radius ground on a lathe tool nose? (2-12)

20. What are the advantages of chip breakers? (2-13)

21 What type of tool holder is used most often in straight-turning operations, and why? (2-15)

22 What are the factors involved in selecting workholding devices on a lathe? (3-1)

23 What types of chucks are used most often? (3-2)

24. What type of chuck would be used when extreme accuracy is not essential? To hold a piece of square material? To hold an oval-shaped piece? (3-3, 4, 5)

25. In what way are the jaws of a universal chuck different from those of an independent chuck? (3-5)

26. What type of chuck has the greatest holding power? (3-5)

27. What is the main advantage of using collet chucks? The disadvantages? (3-8)

28. For what purpose is a spindle nose cap used? (3-9)

29. What should you remember about selecting a collet? (3-10)

30. What do the draw bar handwheel and spindle nose handwheel have in common? (3-12, 13)

31. What type of work can be held on a faceplate? (3-15)

32. What is one way of holding an elbow-shaped piece on a lathe? (3-15)

33. What is the main difference between using a faceplate and a driveplate? (3-15-17)

34. What is the main limitation of a solid mandrel? (3-19)

35. Which end of a solid mandrel has the size marked on it? (3-19)

36. If you had several pieces of work to machine to the same diameter, what type of mandrel should you use? (3-23)

37. What are the three types of lathe spindles? (3-25)

38. What could cause a chuck or faceplate to run untrue? (3-26)

39. Why should cradle blocks be used in changing chucks? (3-26, 27, 28)

40. What is the difference between a dead center and a live center? (3-29)

41. What are the two *least* accurate methods of alining centers? (3-32, 33)

42. What are the two *most* accurate methods of alining centers? (3-34, 35)

43. What type of lathe center would you use for most work? For facing work between centers. For heavy duty work when friction would be a problem? (3-36)

44. What is used as a center hole and center lubricant? (3-36*h*)

45 What is the common name of a combination drill and countersink? (3-37)

46. What are the factors affecting the size of center holes? (3-37)

CHAPTER 2

Objectives: To exhibit a knowledge of facing and straight turning. To be able to explain the machining of shoulders, corners, and grooves. To show an understanding of radii and form turning; the different methods of taper turning; and drilling, boring, reaming, and recessing operations on a lathe.

1. Why is work faced? (4-2)

2. How is work faced? (4-3)

3. What is the main advantage of feeding the facing tool toward the center of the work instead of away from the center? (4-3)

4. Why are the amount of overhang and height of the cutting tool important in facing? (4-4, 5)

5. What type of center is used in the tailstock for facing between centers? Why? (4-6)

6. How heavy should roughing cuts be in straight turning? (4-10)

7. On a piece of work that has two or more diameters, why should the largest diameter be turned first? (4-11)

8. How are depths of cut set? (4-13)

9. How is backlash eliminated from the cross feed? (4-16)

10. How should the compound rest and toolpost be positioned for straight turning operations? (4-17)

11. What is the biggest objection to work that has a square shoulder? (5-3)

12. What type of shoulder is stronger, filleted or angular? (5-5, 6)

13. What are the three methods of producing round corners? (5-9)

14. What are the purposes of an undercut? (5-11)

15. Why are grooves machined on shafts? (5-12)

16. What is the best way to cut off tubing into short pieces? (5-13)

17. If the tool starts to gouge during parting, what should you do? (5-15)

18. What type of form tool cuts a concave shape? (6-3)

19. How long should the radius rod that fits between the cross-slide and tailstock be? (6-5)

20. What is the taper per foot in the Morse system? (7-3,*a*)

21. What are the differences between self-holding and self-releasing tapers? (7-3, 4)

22. How are tapers checked for accuracy? (7-5)

23. What limits the length of the taper that can be cut with the compound rest? (7-6)

24. What is the taper per inch (TPI) of a 4-inch piece of work that measures 1.300 on the large end and 0.900 (TPI = $\frac{\text{LD-SD}}{\text{L of T}}$) on the small end? (7-7)

25. What is the tangent of the angle (tan $\angle$ = TPI/2) in question 24? (7-7)

26. How are long, slow tapers cut on a lathe that does not have a taper attachment? (7-9)

27. What will happen to a taper if work of different lengths is turned with the tailstock offset the same amount? (7-11)

28. What are the two *least* accurate methods of offsetting a tailstock for taper cutting? (7-12)

29. What method of offsetting the tailstock is the most accurate? (7-16)

30. What are some disadvantages of the offset tailstock method of turning tapers? (7-17)

31. What are some advantages of using a taper attachment? (7-18)

32. When a taper attachment is used, what would cause a straight portion to be turned instead of a taper? (7-21)

33. How is the guide bar on a taper attachment graduated? (7-22)

34. How are drills held in a lathe? (8-2)

35. How is a taper shank drill whose shank is too small held in a tailstock spindle? (8-4)

36. What are the graduations on the tailstock spindle used for? (8-7)

37. What are some advantages of boring on a lathe? (8-7)

38. Why should the largest boring bar possible be used? (8-10)

39. How does the boring speed compare to the turning speed on material of the same diameter as the hole? (8-11)

40. What are some advantages of using a boring bar and boring bar holder? (8-15)

41. What governs the amount of end relief ground on boring tools? (8-17)

42. What is the cause and remedy for boring a bell-mouthed hole? (8-20)

43. When taper boring a through hole is done, what size should the drill be that is used to provide a hole for the boring bar? (8-23)

44. How are internal tapers checked? (8-25)

45 What are other names for recessing? (8-29)

CHAPTER 3

Objectives: To be able to explain the various types and uses of threads and how they are measured; to exhibit a knowledge of the machining of American Standard and Unified threads; and to be able to explain how to machine Acme, square, and buttress threads.

1. What does the term "threads per inch" mean? (9-2,*b*)

2 What is the difference between a thread major diameter and norminal size? (9-2,*e,g*)

3. What system is replacing the American National Thread System? (9-3, 4)

4. What are the three common series of threads? (9-6)

5. Why are coarse threads used in aluminum? Fine threads on an engine mount? (9-7)

6. Explain the thread designation 1/2 - 20 - UNF - 2A - LH? (9-8, 10)

7. What is the least accurate method of checking threads? The most accurate method? (9-13, 14, 16)

8. Why should you grind two threading tools? (9-18)

9. What factor does the side relief angle of a thread depend upon? (9-19)

10. What side relief angle should be ground on a threading tool? (9-20)

11. How is the form of a threading tool checked? (9-21)

12. How far does the tool move per work revolution in cutting a thread? (10-2)

13. What are the three conditions that must be met prior to machining threads on a lathe? (10-3)

14. In threading work held between centers, why should the tail of the lathe dog be wedged in the driveplate slot? (10-4)

15. What cutting tools may be used to machine the runout groove for the threads on a shaft? (10-6)

16. What is used to set the depth of cut in making threading cuts? (10-7)

17. What causes the carriage to move after the lead screw begins to revolve? (10-13)

18. How do you know when to engage the split nut? (10-13)

19. What checks should you make after finishing the first cut in threading? (10-14)

20. How do speeds for threading compare to those for turning? (10-15)

21. How can you finish both sides of a thread at the same time? (10-17)

What are some of the differences between external and internal threading on a lathe? (10-20)

23. What are the positions of the compound rest in internal threading? (10-23)

24. What spindle speed should be used in power tapping? (10-27)

25. How can pressure be exerted against a die so that it is alined correctly? (10-31)

26. What is the difference between right-hand and left-hand threads? (10-32)

27. What are the difference between chasing left- and right-hand threads? (10-32, 33)

28. Why is greater side clearance required on a tool to be used to chase a multiple thread than for chasing single threads of the same pitch? (10-36)

29. Why are pipe threads tapered? (10-37)

30. What is the main difference between chasing a Unified thread and a pipe thread? (10-39)

31. What are some of the advantages of an Acme thread over a square thread? (11-2, 6)

32. What type of threads could be used in a vise? (11-2, 7)

33. How are special threads measured? (11-8)

CHAPTER 4

Objectives: To be able to explain how to do filing, polishing, and knurling, and spring winding on a lathe. To demonstrate a knowledge of the use of the center rest, follower rest, micrometer carriage stop, and tool post grinder attachments.

1. What are the two finishing operations? (12-2)

2. Why should the file teeth be kept clean? (12-3)

3. What types of files are used in filing work in a lathe? (12-4)

4. What is used to polish ferrous and nonferrous metals? (12 5)

5. Why are the ends of an abrasive strip separated when work is polished in a lathe? (12-8)

6. What is the main reason for knurling? (12-9)

7. What are the two patterns of knurls? (12-11)

8. In what position is the compound placed to perform ordinary knurling? (12-14)

9. Why is the compound swung 5° to the right to start the knurl between layout lines? (12-16, 17)

10. Why should the work rotation not be stopped with the knurling tool engaged? (12-18)

11. How can you tell the difference between a tension and compression spring? (12-19)

12. What are the purposes of a center rest? (13-2)

13. How is a follower rest different from a center rest? (13-6)

14. How close can the carriage be positioned with the carriage stop? (13-7)

15. When would it be necessary to use a tool post grinder? (13-8)

16. What surface foot speed is recommended for the work that is to be ground with a toolpost grinder? (13-11)

17. In refinishing lathe centers, why is the headstock center finished last? (13-12)

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 1

1. Toolroom, engine, and turret.

2. By the maximum diameter that can be swung over the ways, the distance between centers, and the overall length of the bed.

3. The ways are accurately machined surfaces on the bed that provide alinement and a bearing surface for the headstock, tailstock, and carriage.

4. A Morse taper.

5. To support work between centers, to support work extending from a chuck, and to hold cutting tools.

6. The primary function of the carriage is to carry the cutting tool. The carriage contains the apron and saddle, which in turn is made up of the cross slide and compound rest.

7. Spindle controls, lead screw and feed rod controls, and power feed controls.

8. Those that cause the spindle to rotate and those that determine the speed of rotation.

9. The position of the levers located on the gearbox.

10. Spindle rpm.

11. The lowest CFS value for the metal being turned.

12. The distance the tool advances per revolution of the spindle.

13. The distance the tool is fed into the work below the surface.

14. They cannot be used at high cutting speeds because the heat generated breaks down the cutting edge.

15. It is the property of a metal that enables it to retain its hardness and sharp cutting edge while it is red hot during a cutting operation.

16. Carbide-tipped tools are harder and more brittle than high-speed tools. They require special grinding wheels for sharpening.

17. The various angles and radii that form the shape of the tool, enabling it to cut.

18. End relief and back rake.

19. To strengthen the tip, assist in radiating heat away from the cutting tip and provide a good finish.

20. They prevent long, dangerous chips from forming, reduce heat in the tool, and permit better coolant flow.

21. The left-hand toolholder is used most often because it allows the tool to cut closest to the chuck or lathe dog.

22. The shape of the object, the operation to be performed, and the degree of accuracy required in alinement.

23. Three- or six-jaw universal and four-jaw independent.

24. A universal chuck is used when extreme accuracy is *not* required, a four jaw independent is used for the square- and oval-shaped objects.

25. The universal chuck jaws cannot be removed and reinserted in reverse to handle larger than normal material, but the jaws on an independent chuck can.

26. A four-jaw independent chuck.

27. The main advantage is that they are very accurate when they are used with the correct sized material, thus eliminating having to true each piece. The disadvantages are that they do *not* have as strong a gripping power, are generally available only in standard, common sizes, and the work cannot be of a rough finish.

28. To protect the threads on the spindle nose and to remove the collet adapter.

29. The collet should not be more than .005 larger, and *never* smaller, than the work.

30. Both are turned clockwise to tighten the collet around the work.

31. Large, irregularly shaped work; thin, flat work; and work that requires offset holes.

32. By mounting it on an angle plate that has been attached to a faceplate.

33. A faceplate can have work mounted on it, whereas a driveplate should not have work mounted on it.

34. A solid mandrel can be used only in a hole that has been accurately machined to a standard size.

35. The large end.

36. A gang mandrel.

37. The cam lock, tapered, and threaded.

38. A chip or burr caught between the spindle nose and the mounting surface of the chuck or faceplate.

39. To avoid possible accidents, such as dropping the chuck on your hands or the ways of the lathe, or straining yourself trying to hold chuck in place as it is fastened to the spindle.

40. The nonrotating center mounted in the tailstock is called a dead center, whereas the center mounted in the spindle is called a live center.

41. The cricket mark and visual methods.

42. The dial indicator and test bar method, and the cut and try method.

43. Male center. Half center. Ball bearing center.

44. A mixture of white or red lead and lubricating oil.

45. A center drill.

46. Work diameter, work hardness, depth of cut, operation being performed, and shape of the work.

CHAPTER 2

1. To cut it to the desired length and to obtain a surface from which measurements can be taken.

2. Hold it in a chuck, with a minimum of overhang; and, normally, finish feed the tool from the outside to the center of the work.

3. Feeding toward the center enables you to observe both the layout line and cutting tool in starting the cut.

4. Too much overhang can cause vibration and too little can lead to interference between the tool and the chuck; also, if the tool is not at center height, a stub of metal will be formed as the tool passes over or under the center of the work.

5. Insert a half center, because it will allow the facing tool to finish the entire end of the work and not damage the center.

6. As heavy as the work, machine, and cutting tool can withstand.

7. This will enable you to save the work if the largest diameter is accidentally cut undersize. Turn the work end for end and machine from the other end.

8. By using the graduated collars on the cross-slide or compound rest.

9. Turn the cross-feed handle the reverse of the direction you want to go until the cross-slide starts to move; then turn the handle in the desired direction to the setting you want.

10. The compound rest is positioned 30° to the right of the cross-slide axis with the tool post on the left side of compound rest T-slot.

11. The work cannot be subjected to excessive pressure, or it will fail.

12. Filleted.

13. Freehand manipulation of cross-slide and carriage, using a file, or using a radius form tool.

14. It helps reduce the weight of a shaft and provides clearance or "runout" for toher machining operations.

15. To provide runout, gripping surfaces on tools, to allow clearance for assembly of certain parts, and lubrication channels.

16. Parting on a lathe.

17. Decrease the feed.

18. Convex.

19. Equal to the radius to be cut.

20. Approximately 5/8 inch per foot.

21. A self-holding taper has a very small ($2°$-$3°$) angle and can usually withstand any torque that is applied to it without assistance. A self-releasing taper is much steeper ($16°$) and requires a locking device to keep it from slipping.

22. With a protractor, tapered ring gages, or micrometer.

23. The travel of the compound rest.

24. The TPI is .100.

25. Tan $\angle$ = .050.

26. By offsetting the tailstock.

27. Each piece of work will have a different taper on it.

28. Using the graduation marks on the tailstock base and using a machinist's rule.

29. The cut and try method.

30. Variations in work length or center hole size can lead to variations in taper when duplicate pieces are machined. Center holes get distorted, and centers become damaged. The tailstock has to be alined before straight turning, and only external tapers can be cut.

31. Lathe centers are in alinement, keeping down wear on centers and center holes, and the same taper can be cut on different lengths of work without changing the taper setting.

32. Failing to remove the backlash from the crossfeed screw before beginning the cut.

33. Taper per foot on one end and degrees of taper on the opposite end.

34. In a drill chuck or a tailstock spindle, floated on the dead center, or mounted in the headstock spindle.

35. By inserting the drill in a socket reducer of the correct size, and then inserting the socket reducer in the spindle.

36. For regulating the depth of the hole.

37. Boring on a lathe produces round and concentric holes, and it can be used to produce odd-sized holes.

38. The boring bar is more rigid and it decreases the chance of tool chatter.

39. They are the same.

40. The overhang can be adjusted and the cutting tool can be removed for sharpening without disturbing the setup.

41. The diameter of the hole.

42. It is caused by spring in the boring bar, which can be caused by a dull tool or too heavy a cut, and can be eliminated by feeding the tool at the same setting as the previous cut, sharpening the tool, or reducing feed or depth of cut.

43. A few thousandth smaller than the small diameter of the taper.

44. With a taper plug gage or the mating part.

45. Channeling and chambering.

CHAPTER 3

1. It is the number of threads per inch measured parallel to the thread axis.

2. The major diameter is the largest diameter of an internal or external thread, whereas the nominal size is the one used for identification.

3. The Unified Thread System.

4. The coarse, fine, and extra fine thread series.

5. Coarse threads are used in aluminum because of the better gripping power of the deeper threads, whereas fine threads are used on an engine mount because vibration won't loosen them easily.

6. ½ = ½" nominal size
   20 = 20 threads per inch
   UNF = Unified National Fine series
   2A = 2, class of thread; A, external thread
   LH = left-hand thread

7. The least accurate method uses the mating part, and the most accurate uses go-no-go gages and thread micrometers.

8. One for roughing cuts and the other for finishing cuts.

9. The helix angle of the thread.

10. 3° to 6° more than the helix angle.

11. With a 60° center gage.

12. A distance equal to the lead of the thread.

13. Proper work preparation, correct tool setup, and properly arranged lathe gearing.

14. To prevent damaging the threads or tool if the tool grabs the work.

15. A roundnose tool or parting tool with rounded corners.

16. The compound rest.

17. Engaging the split nut.

18. By observing the thread-chasing dial.

19. Stop the lathe and check the side relief angle of the tool and the actual pitch being cut.

20. Slower than regular turning speeds for the type of metal being machined.

21. By using a tool ground to 60° and setting the compound at 29°

22. Internal threading, unlike external threading, has to take into account clearance restrictions, tool problems, and the different cutting angle that the tool approaches the work.

23. The normal position is 30° to the left of the perpendicular position; but if clearance between the compound rest and the work occur, then position it 180° from normal position.

24. 10 to 30 rpm.

25. With a drill pad center, the front of the tailstock spindle, or the front of the drill chuck.

26. Right-hand threads cause a bolt or screw to advance when it is rotated clockwise, whereas a left-hand thread causes a bolt or screw to advance when it is rotated counterclockwise.

27. For left-hand chasing, the carriage moves toward the tailstock, the compound is swiveled 30° to the left for external threads, an undercut is usually required to provide a starting area for the tool, and the tools are ground just the reverse of right-hand threading tools.

28. Because the greater the number of leads, the greater the helix angle required.

29. This permits tighter connections and helps to seal these connections.

30. The taper attachment has to be used to chase pipe threads and the compound rest is used to withdraw the tool at the end of the cut.

31. It is easier to machine, easier to engage and disengage, and can be adjusted for wear more easily than a square thread.

32. Acme and Buttress.

33. By using wires and a micrometer, a gage, or mating part

CHAPTER 4

1. Polishing and filing.

2. To prevent any metal that has lodged between the teeth from scratching the work or impairing the file's cutting action.

3. Bastard and second cut mill files for straight surfaces, and round and half round for curved surfaces

4. Abrasive cloth is used to polish ferrous metals, and sandpaper is used on nonferrous metals.

5. This prevents it from grabbing and winding around the work, possibly pulling your hands with it.

6. To provide a nonslip gripping service.

7. Diamond and straight line.

8. Perpendicular to the axis of the work.

9. Permits the corners of rollers to contact the work first, reducing the pressure required to force the rollers into the work.

10. The pressure may distort or spring the work.

11. The coils of a tension spring lie alongside one another, but a compression spring has space between each coil.

12. It prevents springing or deflection of slender work and supports work for internal operations, such as boring or threading.

13. A follower rest is fastened to the carriage and moves with it, whereas a center rest is bolted to the ways of the lathe and cannot move.

14. .001 inch.

15. When the work is too hard to machine with normal cutting tools or when a very fine finish is required.

16. 60 to 100 fpm.

17. Because it need not be removed when it is completed, thus maintaining its accuracy.

S T O P - 1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER.
2. USE NUMBER 2 PENCIL ONLY.

EXTENSION COURSE INSTITUTE
VOLUME REVIEW EXERCISE

53130 02 21

Carefully read the following:

DO'S:

1. Check the "course," "volume," and "form" numbers from the answer sheet address tab against the "VRE answer sheet identification number" in the righthand column of the shipping list. If numbers do not match, take action to return the answer sheet and the shipping list to ECI immediately with a note of explanation.
2. Note that item numbers on answer sheet are sequential in each column.
3. Use a medium sharp #2 black lead pencil for marking answer sheet.
4. Write the correct answer in the margin at the left of the item. (When you review for the course examination, you can cover your answers with a strip of paper and then check your review answers against your original choices.) After you are sure of your answers, transfer them to the answer sheet. If you have to change an answer on the answer sheet, be sure that the erasure is complete. Use a clean eraser. But try to avoid any erasure on the answer sheet if at all possible.
5. Take action to return entire answer sheet to ECI.
6. Keep Volume Review Exercise booklet for review and reference.
7. If mandatorily enrolled student, process questions or comments through your unit trainer or OJT supervisor. If voluntarily enrolled student, send questions or comments to ECI on ECI Form 17.

DON'TS:

1. Don't use answer sheets other than one furnished specifically for each review exercise.
2. Don't mark on the answer sheet except to fill in marking blocks. Double marks or excessive markings which overflow marking blocks will register as errors.
3. Don't fold, spindle, staple, tape, or mutilate the answer sheet.
4. Don't use ink or any marking other than a #2 black lead pencil.

NOTE: **STUDY REFERENCE GUIDE NUMBERS ARE USED ON THE VOLUME REVIEW EXERCISE.** In parenthesis after each item number on the VRE is the Study Reference Guide Number where the answer to that item can be located. When answering the items on the VRE, refer to the Study Guide Numbers indicate by these numbers. The VRE results will be sent to you on a postcard which will list the actual VRE items you missed. Go to the VRE booklet and locate the Study Reference Guide Numbers for the items missed. Go to the text and carefully review the areas covered by these references. Review the entire VRE again before you take the closed-book Course Examination.

Multiple Choice

Chapter 1

1. (200) The three classes of lathes are the

   a. engine, turret, and bench.
   b. toolroom, bench, and turret.
   c. bench, toolroom, and engine.
   d. turret, engine, and toolroom.

2. (200) The handscraped, finished surfaces on the heavy framework of a lathe are known as the

   a. beds.
   b. ways.
   c. vee grooves.
   d. alinement grooves.

3. (200) The formula D = 1.414 x S is used to compute the diagonal (D) of a square when the length of a side (S) is known. Which of the following is closest to the maximum size of square material that can be turned in a 10" capacity lathe?

   a. 5 inches square.
   b. 6 inches square.
   c. 7 inches square.
   d. 10 inches square.

4. (200) The hole in the nose of a spindle on most lathes is provided with

   a. a keyway
   b. coarse threads.
   c. a Morse taper
   d. cam-lock clamps.

5. (200) The lateral adjustment of the tailstock is made after first

   a. tightening the spindle clamp.
   b. loosening the sliding spindle clamp.
   c. backing the spindle away from the work.
   d. unclamping the tailstock from the ways.

6. (200) The primary function of the carriage is to carry the

   a. apron.
   b. saddle.
   c. cutting tool.
   d. compound rest.

7. (200) Among other things, the spindle control group controls the

   a. feed rod and lead screw.
   b. rate of spindle rotation.
   c. power feed and lead screw.
   d. spindle speed and power feed.

8. (200) The levers that control the ratio of the lead screw rpm or feed rod rpm to the spindle rpm are located on the

   a. apron.
   b. gearbox.
   c. carriage.
   d. headstock.

9. (200) The split-nut lever is engaged when performing

   a. thread cutting.
   b. taper turning.
   c. straight turning.
   d. facing.

244

10. (200) In the formula $Rpm = \frac{4 \times CFS}{\text{work diameter}}$, the work diameter refers to the

a. largest diameter of the work.
b. smallest diameter of the work.
c. average diameter of the work.
d. diameter actually being machined.

11. (201) Clearance angle is another term for

a. relief angle.
b. rake angle.
c. wedge angle.
d. cutting edge angle.

12. (201) The biggest objection to the use of carbide cutting tools is the

a. high initial cost.
b. difficulty in sharpening.
c. low resistance to shock.
d. requirement for special toolholders.

13. (201) Too large a nose radius ground on a lathe tool would result in

a. better heat radiation.
b. chatter marks on the work.
c. increased power requirement.
d. rapid breakdown of the cutting edge.

14. (201) The type of lathe tool that a chip breaker is most often found on is a

a. facing tool.
b. threading tool.
c. finishing tool.
d. roughing tool.

15. (201) The type of toolholder used for most straight-turning operations using a right-hand tool is the

a. left-hand toolholder.
b. right-hand toolholder.
c. straight toolholder.
d. special toolholder.

16. (202) The quickest and easiest type of chuck to use if 30 pieces of round material are to be faced off and centerdrilled is the

a. solid steel collet.
b. three-jaw independent.
c. three-jaw universal.
d. four-jaw independent.

17. (202) The type of chuck that would hold a piece of hexagonal stock with greater holding power than any other is the

a. solid steel collet.
b. three-jaw combination.
c. three-jaw universal.
d. four-jaw independent.

18. (202) The type of chuck that incorporates the features of an independent jaw chuck and a universal chuck is the

a. three-jaw independent.
b. three-jaw universal.
c. four-jaw independent.
d. four-jaw combination.

19. (202) The *least* desirable chuck for holding work to be threaded is a

a. collet chuck.
b. three-jaw universal chuck.
c. four-jaw universal chuck.
d. four-jaw combination chuck.

20. (202) Standard collet sets consist of collets designed to fit

a. decimal sizes.
b. fractional sizes.
c. decimal and fractional sizes.
d. round, square, and hexagonal shapes.

21. (202) The front of a collet adapter is provided with

a. threads.
b. a keyway.
c. a conical opening.
d. an untapered opening.

22. (202) The number of rubber flex collets required to hold work whose diameter ranges from $^1/_{16}$ to $1^3/_8$ inches is

a. 7.
b. 11.
c. 26.
d. 263.

23. (202) The characteristic that faceplates and driveplates do *not* share is the

a. slots machined in their faces.
b. ability to drive a lathe dog.
c. ease of installation and removal.
d. provision for mounting work on their faces.

24. (203) By accident, a standard size reamer reams a hole several thousandths larger than intended. The type of mandrel that should then be used is

a. a solid mandrel.
b. a special mandrel.
c. an expansion mandrel.
d. an eccentric mandrel.

25. (203) The amount of eccentricity that any particular eccentric mandrel is capable of depends upon the

a. taper of the mandrel.
b overall length of the mandrel.
c. number of center holes in each end.
d. distance between offset and original center holes.

26. (203) The best mandrel to use if 200 washers of the same size have to be made is

a. a gang mandrel.
b. a nut mandrel.
c. an eccentric mandrel.
d. an expansion mandrel.

27. (203) A cradle block is *not* necessary when mounting or dismounting a

a. three-jaw universal chuck.
b four-jaw independent chuck.
c. 6-inch-diameter driveplate.
d. faceplate with a fixture and the work attached.

28. (203) A live center does not require and lubricant because

a. it rotates with the work.
b. the pyramid shape helps drive the work.
c. it has less area in contact with the center hole.
d. the point is harder than a dead center point.

29. (203) When you aline centers using the dial indicator and test bar on the cut and try method, make sure that the test bar or trial piece is the same

a. diameter as the work.
b. length as the work.
c. material as the work.
d. hardness as the work.

30. (203) The type of lathe center that is least likely to be used in the headstock is the

a. male center.
b. half center.
c. female center.
d. self-driving center.

Chapter 2

31. (204) When facing work from the outer surface toward the center, you should use a

a. left-hand finishing tool and right-hand toolholder.
b. left-hand finishing tool and left-hand toolholder.
c. right-hand finishing tool and right-hand toolholder.
d. right-hand roughing tool and straight toolholder.

32. (204) When should a half center be replaced by a standard male center when work is faced between centers?

a. After the first roughing cut.
b. Just prior to the finishing cut.
c. When ferrous metals are machined.
d. When facing operations are complete.

33. (204) In what position is the compound set when it is used to face work to length?

a. Parallel to the axis of the work.
b. Perpendicular to the work axis.
c. Perpendicular to the ways of the lathe.
d. At a 30° angle to the cross slide.

34. (204) Roughing cuts used to remove considerable amounts of metal should be

a. limited to a maximum depth of cut of 0.250 inch.
b. as heavy as lathe, cutting tool, and work can take.
c. restricted to under 100 rpm on the spindle.
d. made with a carbide tool.

35. (204) In straight turning, which of the following actions must be taken when the roughing tool is replaced with a finishing tool?

a. Cut must be picked up.
b. Tailstock must be alined.
c. Centers must be adjusted.
d. Cutting tool must be resharpened.

36. (204) Cross-slide and compound rest graduations usually provide a micrometer adjustment for moving the tool

a. 0.0005 or 0.001 per graduation.
b. 0.0005 or 0.002 per revolution.
c. 0.001 or 0.002 per graduation.
d. 0.001 or 0.005 per revolution.

37. (204) The lost motion between a feed screw and the nut it engages is called

a. wear.
b. tolerance.
c. clearance.
d. backlash.

38. (204) With reference to the work, the position of a roughing tool should be

a. at center height.
b. 5^0 below center height.
c. 5^0 above center height
d. 3/64" below center height.

39. (205) The surface that joins two different diameters is called

a. a groove.
b. a corner.
c. a shoulder.
d. an undercut.

40. (205) Which is the strongest type of shoulder?

a. Square.
b. Angular.
c. Undercut.
d. Filleted.

41. (205) The type of corner which is best produced by using a form tool is the

a. round.
b. square.
c. angular.
d. chamfered.

42. (205) Cutting tool runout for later machining operations is provided by machining

a. fillets and grooves.
b. undercuts and fillets.
c. grooves and undercuts.
d. shoulders and undercuts.

43. (205) The length of the cutting portion of a parting tool should be equal to

a. half the work diameter.
b. just over half the work diameter.
c. two-thirds the work diameter.
d. the work diameter.

44. (206) The most proctical method of manufacturing several dozen small spherical-shaped knobs involves the use of a

a. radius rod.
b. convex form tool.
c. concave form tool.
d. template and pointer.

45. (206) The two classes of standard machine tapers are the

a. friction and drive fit.
b. slow taper and fast taper.
c. American Standard and Jarno.
d. Self-holding and self-releasing.

46. (202) The type of self-holding taper used exclusively on twist drill shanks is the

a. Morse taper.
b. Jarno taper.
c. Brown and Sharpe taper.
d. American Standard taper.

248

47. (206) The only restriction placed on taper turning with the compound rest concerns the

a. work length.
b. work diameter.
c. included angle.
d. amount of compound travel.

48. (206) What is the taper of a taper pin?

a. 1/4 inch per inch.
b. 1/4 inch per foot.
c. 0.600 inch per foot.
d. 3/4 inch per foot.

49. (206) Self-releasing tapers differ from self-holding tapers in that they have a

a. positive locking device.
b. self-holding tang drive.
c. taper of 3/4 inch per foot.
d. taper of $2\frac{1}{2}$ inches per foot.

50. (206) When the large and small diameters and length of taper are known, you must calculate for the angle of taper to

a. find the TPI.
b. determine the TPF.
c. set the compound.
d. position the cutting tool.

51. (206) If the tailstock offset stays constant and the length of work decreases, the effect on the taper would be to

a. eliminate the taper.
b. decrease the taper.
c. increase the taper.
d. keep the taper constant.

52. (206) In the offset tailstock method of taper turning, the primary factors in determining the degree of taper that can be turned are the maximum

a. work diameter and work length.
b. size of center holes and work diameter.
c. amount of offset and work length.
d. amount of offset and maximum work diameter.

53. (206) Once it is set up, the main advantage of cutting tapers using the taper attachment is that this method

a. reduces wear on centers.
b. keeps the centers in alinement.
c. eliminates center hole distortion.
d. cuts the same taper on varying lengths of work.

54. (206) A straight portion turned on work that should be tapered is caused by the

a. centers being out of alinement.
b. wrong setting of the taper attachment.
c. spring in the work when the cut is started.
d. failure to eliminate backlash from the crossfeed screw.

55. (206) When the taper attachment draw bar has locked the crosslide to the guide block, depths of cut are controlled by using the compound rest set

a. 60° to the ways.
b. parallel to the ways.
c. perpendicular to the ways.
d. at any convenient angle.

56. (207) A method that is *not* normally used to hold a drill in a lathe is

a. mounting in the headstock spindle.
b. holding in a drill chuck.
c. floating on a live center.
d. inserting into the tailstock spindle.

57. (207) When the shank of a taper-shank drill is too small to fit into the tailstock spindle, you should use a

a. lathe dog.
b. socket reducer.
c. drill holder.
d. dead-center float.

58. (207) Drilling a hole to a predetermined depth is regulated by using the graduations on the

a. tailstock spindle.
b. crossfeed collar.
c. dial indicator
d. compound rest.

59. (207) Prior to boring, the drilled hole diameter should be

a. 1/64 inch smaller than the final size.
b. 1/32 inch smaller than the final size.
c. 1/32 inch larger than the diameter of the boring bar
d. equal to the boring bar diameter.

60. (207) Compared with external turning, boring a hole to size requires

a. the same speed and feed.
b. a decrease in speed and feed.
c. a decrease in speed and the same feed.
d. the same speed and a decrease in feed.

61. (207) An important characteristic that a boring bar and a solid boring tool have in common is that both

a. have an adjustable overhang.
b. have the same degree of rigidity.
c. are held on the compound rest by the toolholder.
d. can be easily removed without disturbing the holder position.

62. (207) The end relief of a boring tool varies with the

a. diameter of the hole.
b. size of the boring bar.
c. type of finish desired.
d. depth of the hole

63. (207) The position of the boring tool in relation to the work for straight boring is

a. 5° above center for straight and on center for taper boring.
b. 5° above center for both straight and taper boring.
c. on center for straight and 5° above for taper boring.
d. on center for both straight and taper boring.

64. (207) Spring in a boring bar often leads to

a. chatter marks.
b. oval-shaped holes.
c. bell-mouth holes.
d. break down of the tool cutting edge.

65. (207) When the taper attachment is used to bore tapered holes, the compound rest is positioned

a. parallel to the ways.
b. perpendicular to the ways.
c. at the same angle as the taper.
d. perpendicular to the angle of the taper.

66. (208) The speed and feed for machine reaming as compared to drilling should be

a. same speed and 1/2 the feed.
b. 1/2 the speed and same feed.
c. less speed and greater feed.
d. greater speed and same feed.

Chapter 3

67. (209) Changing the fit between a thread being machined and its mating thread is dependent upon a change in the

a. pitch.
b. nominal size.
c. pitch diameter.
d. major diameter.

68. (209) The thread system established by the United Kingdom, United States, and Canada is known as the

a. National System.
b. Unified System.
c. Coordinated System.
d. Commonwealth System.

69. (209) The three most common thread series in the United States are the

a. fine, extra fine, and coarse.
b. fine, special, and coarse.
c. coarse, fine, and special.
d. coarse, fine, and pipe.

70. (209) Classes of thread fits are designated by a

a. title.
b. letter.
c. number.
d. symbol.

71. (209) The part of a thread designation that refers to size is actually the thread

a. major diameter
b. nominal diameter.
c. basic diameter.
d. actual diameter.

72. (209) What is the *least* accurate method of checking threads?

a. Mating part
b. Thread calipers
c. Thread micrometers.
d. Go and no-go gages.

73. (209) A thread micrometer measures which diameter?

a. Major.
b. Minor.
c. Pitch.
d. Actual.

74. (209) The amount of relief angle ground on a threading tool depends upon the

a. diameter of the work.
b. helix angle of the thread.
c. series of thread to be cut
d. type of material to be threaded

75. (209) Threading tools for coarse threads require more side relief than tools for fine threads because the

a. thread is deeper.
b. diameter is larger.
c. included angle changes.
d. helix angle is greater.

76. (210) Engaging the split nut with the lead screw causes

a. movement of the carriage.
b. rotation of the feed rod.
c. rotation of the lead screw.
d. disengagement of the gear train.

252

77. (212) What type of thread should be used when the pressure during use is in only one direction?

a. Acme
b. Straight
c. Buttress
d. Unified.

78. (210) When you chamfer work prior to a threading operation, the depth of the chamfer should be

a. equal to the helix angle.
b. equal to the root diameter.
c. slightly less than the thread depth
d. slightly more than the thread depth

79. (210) During threading, successive depths of cut are taken with the

a. feed rod.
b. split nut.
c. crossfeed.
d. compound rest.

80. (210) The range of thread leads that can be cut on a lathe can be increased by

a. engaging the lead screw gear.
b. disengaging the spindle gear.
c. changing the gear box levers.
d. changing the spindle stud gear.

81. (210) The use of the thread-chasing dial is unnecessary when the number of threads per inch being cut is

a. any even number.
b. an odd number.
c. divisible by five or ten.
d. a multiple of the lead screw thread.

82. (210) The depth of cut during thread chasing should

a. remain the same.
b. not exceed .015.
c. decrease with succeeding cuts.
d. increase with succeeding cuts.

83. (210) A threading tool ground with a 60° included angle and fed into the work with the compound set at 29° is a method used for finishing

a. both sides of the general thread pitch range simultaneously.
b. each side of the general thread pitch range individually.
c. both sides of a very coarse thread simultaneously.
d. each side of a very coarse thread individually.

84. (211) Prior to chasing an internal thread, the work should be bored to the

a. pitch diameter of the thread.
b. minor diameter of the thread.
c. major diameter of the thread.
d. nominal diameter of the thread.

85. (211) During internal thread chasing operations, the compound rest is positioned

a. 30° to the left of the perpendicular position.
b. 30° to the right of the perpendicular position.
c. 180° to the left of the parallel position.
d. 180° to the right of the perpendicular position.

86. (211) If the recommended clearance between an internal and external thread is not available for the hole that is to be threaded, it is customary to cut the thread

a. .002 to .005 inch larger than the major diameter.
b. .005 to .010 inch larger than the major diameter.
c. .005 to .015 inch per inch of the major diameter.
d. .010 to .015 inch per inch of the major diameter.

87. (211) In hand tapping work in a lathe, the center can be removed from the tap as soon as

a. all the chips are removed from the tap flutes.
b. enough threads are cut to maintain tap alinement.
c. the tap begins to emerge from other side of work.
d. the compound rest begins to interfere with the tap wrench.

88. (211) The size of the groove machined in a shaft as the starting point for a left-hand threading tool should be

a. two thread pitches wide and as deep as the pitch diameter.
b. one thread pitch wide and as deep as the double depth of thread.
c. equal to one thread pitch wide and slightly deeper than the single thread depth.
d. a maximum of one thread pitch wide and slightly shallower than the single thread depth.

89. (211) One complete turn of a nut on a 1-16 UNC double-lead threaded bolt will cause the nut to advance

a. 1/32 inch.
b. 1/16 inch.
c. 1/8 inch.
d. 1/4 inch.

90. (211) You are preparing a lathe to cut 6 threads per inch, double lead. The number of threads per inch you would set the lathe to cut is

a. 2.
b. 3.
c. 6.
d. 12.

91. (211) You are given a requirement to chase threads on various pieces of pipe. What should the taper attachment setting be and how is the thread cutting tool alined?

a. 1/16 taper per inch and perpendicular to the tapered surface.
b. 3/8 taper per foot and perpendicular to the work axis.
c. 3/4 taper per foot and perpendicular to the work axis.
d. 3/4 taper per foot and perpendicular to the tapered surface.

Chapter 4

92 (214) Grinding wheel surface foot speeds are controlled on the toolpost grinder by changing

a. the rpm of the work.
b. belts on the grinder.
c. drive and wheel pulleys.
d. types of grinding wheels.

93. (21_) In filing work in a lathe, the file tip should point toward the

a. headstock at about a 10° angle.
b. tailstock at about a 10° angle.
c. headstock at about a 20° angle.
d. tailstock at about a 20° angle.

94. (213) The primary purpose of a polishing operation is to

a. improve the finish of the work.
b. remove high spots from the work.
c. reduce the dimensions of the work.
d. aid in accurately fitting mating parts.

95. (213) The main purpose for moving the abrasive strip slowly back and forth along the work in polishing a a piece of steel is to prevent the

a. removal of too much metal.
b. generation of excessive heat.
c. abrasive strip from gouging the work
d. formation of scratch rings on the work surface.

96. (213) What is the most common knurling pitch and pattern?

a. Fine diamond.
b. Medium diamond.
c. Fine straight line.
d. Medium straight line.

97. (213) The maximum knurling speed should *not* exceed

a. half the rough turning speed.
b. half the finish turning speed.
c. the highest spped that shifting the speed into reverse permits.
d. the highest speed that shifting the feed into reverse permits.

98. (213) In ordinary knurling, the compound should be positioned

a. perpendicular to the work axis.
b. parallel to the work axis.
c. 30° to the work axis.
d. 45° to the work axis.

99. (213) What is the most probable cause of rings in a knurled surface?

a. Tailstock center coming loose.
b. Center holes of work too small.
c. Work rotation without carriage travel.
d. Pressure too high for size of work.

100. (213) The feed setting on the quick change gearbox in winding a tension spring should be equal to

a. one-half the wire diameter.
b. the diameter of the wire.
c. twice the wire diameter.
d. the desired spacing between the coils.

101. (213) The main difference between the tension and compression spring winding operations on a lathe is in the

a. carriage feed.
b. spindle speed.
c. degree of skill required.
d. manner of holding the spring ends.

102. (213) When springs are wound in the lathe, the diameter of the spring is controlled by the

a. method used to hold the mandrel.
b. pressure placed on the wire during winding.
c. size of the mandrel used to wind the spring.
d. type of wire material used to wind the spring.

103. (214) A principal advantage of the center rest overarm is that

a. vertical adjustment of the work is easier.
b. clearance between the jaws and work is more closely controlled.
c. It compensates for slight irregularities on the surface of the work.
d. It enables duplicate pieces to be removed and replaced without jaw readjustment.

104. (214) When you use a center rest, a concentric bearing surface is machined on the work to

a. true the work in the clamp
b. prevent the work from springing.
c. provide clearance for the center rest jaws
d. provide a bearing surface for the center rest jaws.

105. (214) The adjustable jaws of a center rest should be adjusted to the work while it is

a. stationary.
b. rotated by hand.
c. revolving at a very slow speed.
d. revolting at a rough cutting speed.

106. (214) Grinding with a toolpost grinder is more efficient than machining with a cutting tool for finishing

a. ferrous metals.
b. work shoulders.
c. tapered surfaces.
d. extremely hard metal.

107. (214) When a follower rest is set up, you mount it on the

a. carriage.
b. cathead.
c. lathe ways.
d. compound rest.

108. (214) The attachment that should be used to help accurately position an internal recessing tool after taking some measurements is the

a. compound rest lock.
b. thread chasing dial.
c. micrometer carriage stop.
d. micrometer cross-slide stop.

256

STUDENT REQUEST FOR ASSISTANCE

PRIVACY ACT STATEMENT

AUTHORITY 44 USC 3101. PRINCIPAL PURPOSE(S) To provide student assistance as requested by individual students. ROUTINE USES: This form is shipped with every ECI course package. It is utilized by the student, as needed, to place an inquiry with ECI. DISCLOSURE: Voluntary. The information requested on this form is needed for expeditious handling of the student's need. Failure to provide all information would result in slower action or inability to provide assistance.

SECTION I: CORRECTED OR LATEST ENROLLMENT DATA: MAIL TO ECI, GUNTER AFS ALA 36118

1. THIS REQUEST CONCERNS COURSE
2. TODAY'S DATE
3. ENROLLMENT DATE
4. PREVIOUS SERIAL NUMBER
5. SOCIAL SECURITY NUMBER
6. GRADE/RANK
7. INITIALS LAST NAME
8. OTHER ECI COURSES NOW ENROLLED IN
9. ADDRESS: (OJT ENROLLEES - ADDRESS OF UNIT TRAINING OFFICE/ALL OTHERS - CURRENT MAILING ADDRESS)
   ZIP CODE
10. NAME OF BASE OR INSTALLATION IF NOT SHOWN ABOVE:
11. AUTOVON NUMBER
12. TEST CONTROL OFFICE ZIP CODE/SHRED

SECTION II: Old or INCORRECT ENROLLMENT DATA

1. NAME:
2. GRADE/RANK:
3. SSAN:
4. ADDRESS:
5. TEST OFFICE ZIP/SHRED:

SECTION III: REQUEST FOR MATERIALS, RECORDS, OR SERVICE

(Place an "X" through number in box to left of service requested)

ADDITIONAL FORMS 17 available from trainers, OJT and Education Offices, and ECI. The latest course workbooks have a Form 17 printed on the last page.

1	EXTEND COURSE COMPLETION DATE. (Justify in Remarks)
2	SEND VRE ANSWER SHEETS FOR VOL(s): 1 2 3 4 5 6 7 8 9 - ORIGINALS WERE: NOT RECEIVED, LOST, MISUSED
3	SEND COURSE MATERIALS (Specify in remarks) - ORIGINALS WERE: NOT RECEIVED, LOST, DAMAGED.
4	COURSE EXAM NOT YET RECEIVED. FINAL VRE SUBMITTED FOR GRADING ON (Date):
5	RESULTS FOR VRE VOL(s): 1 2 3 4 5 6 7 8 9 NOT YET RECEIVED. ANSWER SHEET(s) SUBMITTED ON (Date):
6	RESULTS FOR CE NOT YET RECEIVED. ANSWER SHEET SUBMITTED TO ECI ON (Date):
7	PREVIOUS INQUIRY (ECI FORM 17, LTR, MSG) SENT TO ECI ON:
8	GIVE INSTRUCTIONAL ASSISTANCE AS REQUESTED ON REVERSE:
9	OTHER (Explain fully in remarks)

REMARKS: (Continue on Reverse)

OJT STUDENTS must have their OJT Administrator certify this request.

ALL OTHER STUDENTS may certify their own requests.

I certify that the information on this form is accurate and that this request cannot be answered at this station. (Signature)

ECI FORM 17 JUN 77 PREVIOUS EDITIONS MAY BE USED

BEST COPY AVAILABLE

SECTION IV: REQUEST FOR INSTRUCTOR ASSISTANCE

NOTE: Questions or comments relating to the accuracy or currency of textual material should be forwarded directly to preparing agency. Name of agency can be found at the bottom of the inside cover of each text. All other inquiries concerning the course should be forwarded to ECI.

VRE ITEM QUESTIONED: MY QUESTION IS

Course No

Volume No. ______

VRE Form No ______

VRE Item No. ______

Answer You Chose (Letter) ______

Has VRE Answer Sheet been submitted for grading?

☐ YES ☐ NO

REFERENCE
(Textual support for the answer I chose can be found as shown below)

In Volume No: ______

On Page No: ______

In ______ (Left) ______ (Right) Column

Lines ______ Through ______

Remarks:

258

AU GAFS, AL(900833)400

53130 03 1071

CDC 53130

Apprentice Machinist

Volume 3

Shaper and Contour Machine Work

Extension Course Institute

Air University

Preface

VOLUME 3 OF THIS CDC deals with shaper and contour machines. You will not be expected to go directly into the shop and operate these machines after studying this volume. However, knowledge of tool and work setups, as well as of the machining operations, will make your work in the shop a lot easier. The first two chapters cover shaper work and planing operations, while the last two concern contour machines and the operations that can be performed on them. To understand the scope of this volume. you will find it helpful to leaf quickly through the pages of each chapter and note the numbered headings.

Code numbers appearing on the figures are for preparing agency identification only.

If you have questions on the accuracy or currency of the subject matter of this text, or recommendations for its improvement, send them to Tech Tng Cen (TTOC) Chanute AFB, IL 61868.

If you have questions on course enrollment or administration, or on any of ECI's instructional aids (Your Key to Career Development, Study Reference Guides, Chapter Review Exercises, Volume Review Exercise, and Course Examination), consult your education officer, training officer, or NCO, as appropriate. If he can't answer your questions, send them to ECI, Gunter AFB, AL 36118, preferably on ECI Form 17, Student Request for Assistance.

This volume is valued at 15 hours (5 points).

Material in this volume is technically accurate, adequate and current as of June 1971.

Contents

		Page
	Preface	iii
Chapter		
1	SHAPER WORK	1
2	PLANNING OPERATIONS	i7
3	CONTOUR MACHINI. WORK	26
4	SAWING, FILING, AND POLISHING OPERATIONS	40
	Bibliography	49

Shaper Work

ARE ALL manufactured items cylindrical in shape? Of course not. The shape of many items is rectangular, square, or hexagonal, or a combination of several geometric figures. In other words, many objects have several flat surfaces that intersect. One machine which is designed primarily to produce flat surfaces is the shaper. Since you will machine objects with flat surfaces, you must understand its operation. In this chapter, we will discuss the construction and operations of shapers, workholding devices, and cutting tools and toolholders.

1. Construction and Operation

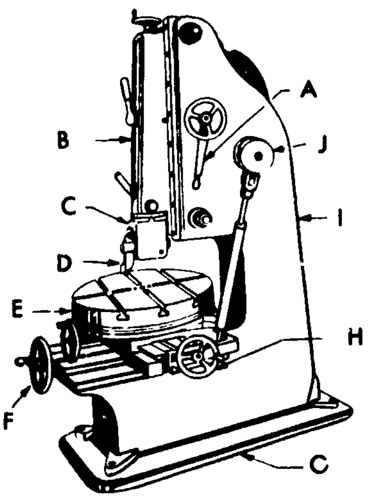
1-1. In this section we will discuss the types of shapers and also the major parts of a shaper and their functions and operating features. Alinement of the table, vise, and head in relation to one another is required for most shaper operations.

1-2. **Construction.** The *shaper* uses a reciprocating ram to carry the cutting tool. The cutting action takes place during the forward stroke of the ram. The feed takes place during the return stroke. The speed of the forward stroke is much slower than that of the return stroke. The toolmarks made by a shaper are parallel and evenly spaced. They produce a surface that you can scrape or polish to a high degree of finish. The work is mounted on a movable table which feeds the work at a right angle to the movement of the ram. The size of a shaper is designated by the maximum length of its stroke. For example, a 24-inch shaper is capable of machining an object 24 inches long. Shapers are used to machine small and medium size work. Extremely large work is machined with a planer, which is somewhat similar to a shaper. The cutting tool on a planer remains stationary, and the work reciprocates (moves back and forth). The opposite is true of shapers.

1-3. *Types of shapers.* There are two major types of shapers: *vertical* and *horizontal*. The ram on a vertical shaper reciprocates vertically, while the ram on a horizontal shaper (the most common type) reciprocates horizontally.

1-4. The *vertical shaper,* shown in figure 1, is sometimes called a *slotter*. It is especially adapted to slot angles up to 5°. The head can also be swiveled through 180° in a horizontal plane. The worktable is supported on a fixed knee and can be moved in or out and longitudinally by hand or power. When cylindrical or semicylindrical forms are being shaped, it can be rotated by hand or by power. The work may be held in a vise, clamps, or special fixtures.

1-5. *Horizontal shapers* are classified as *universal* and *plain* shapers. The table on universal shapers can be swiveled in a vertical plane on a trunnion, and one surface can be tilted. In contrast, the table on a plain shaper cannot be tilted or swiveled. This is the only difference between a universal shaper and a plain shaper.



53—320

A. Clutch lever
B. Vertical ram
C. Tool head
D. Cutting tool
E. Rotary table
F. Transverse feed handwheel
G. Base
H. Longitudinal feed handwheel
I. Column
J. Feeding mechanism

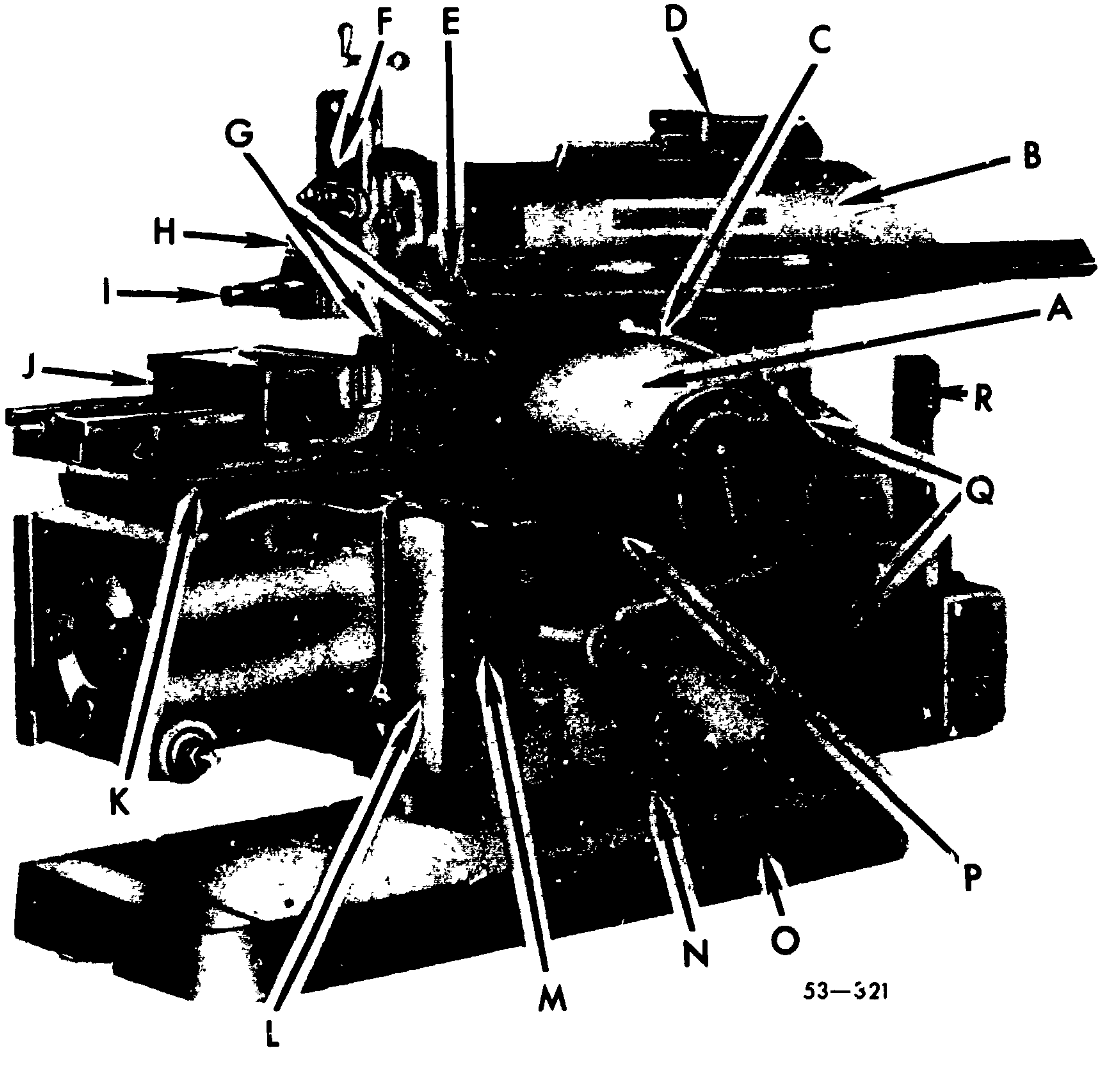
Figure 1. Vertical shaper.

1-6. *Major parts of a shaper.* In order to fully understand the operation of a shaper, you must know the names of the major parts of a shaper and the purposes for which they serve. Figure 2 will help you to identify the parts.

a. The *column* is mounted on the base. Both the column and the base are designed to reduce vibration to a minimum. Accurately machined and scraped dovetail ways at the top of the column serve as a bearing surface for the ram. The front of the column has vertical ways upon which the crossrail slides up or down.

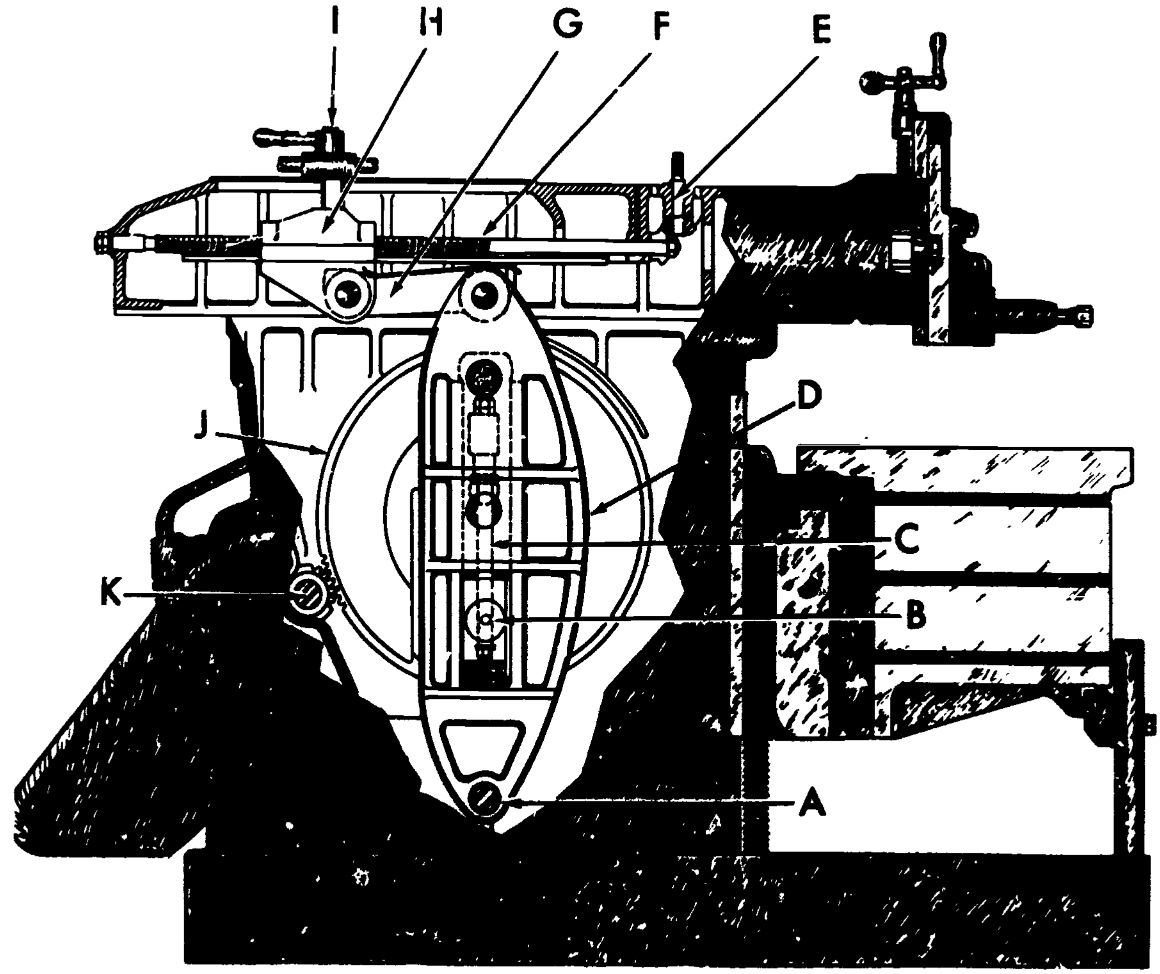
b. The *ram* is heavily ribbed inside to give strength and rigidity. It is actuated back and forth horizontally in the dovetail ways by means of the rocker arm. The ram contains a stroke position mechanism.

c. The *crossrail* is attached to the vertical ways of the base by means of plates, gibs, and bolts. The top surface and the front surfaces are accurately machined and scraped to provide smooth bearing surfaces for the trunnion apron as it is fed back and forth. The crossrail contains the table elevating mechanism as well as the table traverse mechanism.



A. Column
B. Ram
C. Clutch lever
D. Ram positioning shaft
E. Dovetail ways
F. Tool slide
G. Column vertical ways
H. Clapper box
I. Tool post
J. Vise
K. Tilting surface
L. Trunnion apron
M. Crossrail
N. Hand traverse crank (crossfeed screw)
O. Base
P. Rapid traverse lever
Q. Speed-change levers
R. Motor switch

Figure 2. Universal shaper.



53—322

A. Rocker arm shaft
B. Crank pin
C. Sliding block
D. Rocker arm
E. Ram positioning shaft
F. Ram adjustment screw
G. Link
H. Ram adjustment nut
I. Ram clamp handle
J. Crank gear
K. Crank gear pinion

Figure 3. Drive mechanism.

d. The *trunnion apron* is an accurately machined casting which is gibbed to the crossrail. The front part of the casting is machined cylindrically and is called the *trunnion.*

e. The *table* is an accurately machined casting of boxlike construction with a machined hole through it, which fits around the trunnion.

f. The *table feed mechanism* is powered from a cam through a ratchet and gearing system which causes the table feed screw to rotate and move the table. The automatic feed moves the table a certain distance, usually in thousandths of an inch, for each stroke of the ram. The feed mechanism also has a rapid traverse for repositioning the table and the work quickly after the cutting stroke.

g. The *driving mechanism* or *bull wheel,* shown in figure 3, is a large gear which is mounted inside the shaper column and is driven by a pinion. Anchored to the center of the bull wheel is a radial slide which carries a sliding block into which the *crankpin* fits. The position of this pin is controlled by a small lead screw which is connected to the operator's side of the machine by means of bevel gears and a square-ended shaft. The location of the sliding block and pin with respect to the center of the bull wheel governs the length of the stroke. As the bull wheel and crankpin turn, the sliding block moves up and down in a rocker arm, causing the rocker arm to move back and forth. The top end of the rocker arm is connected to the ram by a link, and the movement of the rocker arm imparted through this link gives the ram a reciprocating motion. The number of strokes that the ram travels per minute is governed by the change gearbox on the operator's side of the machine.

1-7. **Operation.** An explanation of the function and operation of some of the shaper parts will help you to understand how adjustments are made and how shaping operations are performed. Figures 2 and 3 will help you to locate and identify the various shaper parts.

1-8. *Length of Stroke.* You change the *length of the stroke* by turning the stroke adjusting shaft with a handcrank. A dial indicates the length of stroke. You should normally use a stroke that is 1 inch longer than the length of the work. The excess length of stroke is the *overstroke.* CAUTION: Always remove the adjusting crank as soon as you have made the adjustments.

1-9. *Position of ram.* You position the ram by turning the ram-positioning shaft with the handcrank. Loosen the clamp handle before you position the ram, and *tighten it immediately afterward.* Position the ram so that the cutting tool extends beyond the end of the work approximately ⅛ inch to permit the chip to break off. The remainder (the greater part at the rear of the work) of the overstroke permits the clapper box to seat before the cutting stroke and the feed take place. Before you position the ram, set the length of the stroke and turn the motor ON. Extend the ram on the forward stroke by jogging the clutch lever until the cutting tool is beyond the work. CAUTION: Always check to insure that the tool clears the work and that the toolslide clears the dovetail ways before you engage the ram activating clutch.

1-10. *Speed.* The *speed* of a shaper is expressed in strokes per minute The number of strokes per minute depends upon the cutting foot speed recommended for the metal and the length of the stroke. Use the following formula to convert the recommended cutting foot speed for shapers, shown in chart 1, to strokes per minute:

$$N = \frac{CFS \times 7}{L}$$

when:

N = Number of strokes per minute
CFS = Cutting foot speed
L = Length of stroke in inches (including the 1-inch overtravel)

Example: Assume that you are to rough machine a piece of tool steel 11 inches long, using high-speed tools. At what speed should you operate the shaper? By consulting a machinist's handbook, you will find that the recommended CFS is 40. Then:

$$N = \frac{40 \times 7}{12} \quad \text{(the length of the work plus the 1-inch overtravel)}$$

$$N = \frac{280}{12}$$

$$N = 23\tfrac{1}{3}$$

Thus, you would set the speed-change levers to the nearest available speed as indicated on the plate which shows the lever positions for various strokes per minute. NOTE: To prevent damage to the shaper, the ram must be stopped (on most shapers) before the speed-change levers are moved.

1-11. Take the following factors into consideration when you determine shaper speed:

- The hardness of the material. Usually, you machine hard material at slower speeds than soft or ductile material.
- Tool shape. Use low speeds for form tools because of the increased area of contact.

CHART 1

RECOMMENDED SHAPER CUTTING SPEEDS

Materials To Be Machined	Carbon Steel Tools		High Speed Steel Tools	
	Cutting Speed (feet per minute)			
Cast Iron	Roughing 30	Finishing 20	Roughing 60	Finishing 40
Mild Steel	25	40	50	80
Tool Steel	20	30	40	60
Brass and Bronze	75	100	150	200
Aluminum	75	100	150	200

53-416

• Tool material. Use lower speeds for carbon steel tools than for high-speed steel tools.

• Power, design, and condition of the machine. Use higher speeds on heavy, rigidly constructed machines and on machines that are in good condition than on machines that are intended for light-duty applications. CAUTION: NEVER OPERATE THE SHAPER AT ITS HIGHEST SPEED AND LONGEST LENGTH OF STROKE AT THE SAME TIME. DOING SO MAY DAMAGE THE SHAPER OR CAUSE IT TO MOVE OUT OF ALINEMENT.

1-12. *Feed.* The tool may be fed into the work by means of the toolslide in the same manner as that which applies in using the lathe compound rest to machine short taper. However, in shaper work, feed is normally understood to mean the rate of crossfeed travel. This is the distance the table moves the work toward the tool before each cutting stroke, as shown in figure 4. You select the desired rate of feed by positioning the feed selection lever as shown on a graduated plate. The available feeds on a shaper are much coarser than those on a lathe. They range from 0.010 inch on most shapers to approximately 0.170 inch. The feed that you select depends upon the finish you desire and on the depth of cut. You should take into consideration three factors that determine the speed you select: the tool, the material, and the machine.

1-13. The directional feed control lever engages the power feed and also determines the direction of the table travel. If you turn the control lever toward the shaper, the table will feed toward the shaper. If you turn it away from the shaper, it will feed away from the shaper. The power disengages when you place the feed control lever in the center (neutral) position.

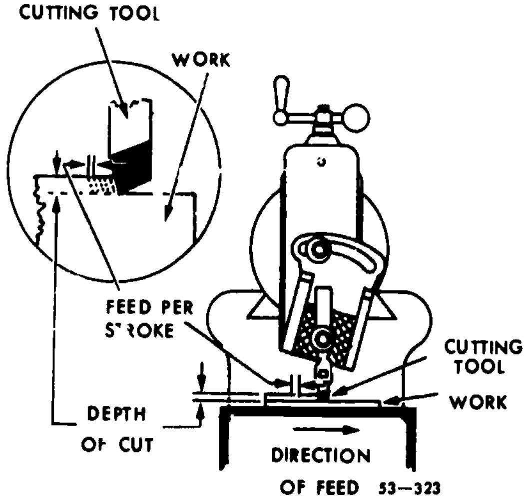


Figure 4. Shaper feed and depth of cut.

1-14. You can move or feed the table by hand by placing the directional feed control lever in the neutral position and turning the crossfeed screw with the hand crank. This safety type crank is held on the crossfeed screw by a spring-loaded ball that engages a groove which is machined around the end of the crossfeed screw shaft. The ball and groove hold the crank on the shaft but permit the crank to hang stationary when the shaft rotates. The top of the crank is fastened to the portion of the shaft that contains the spring-loaded ball by means of a hinge. You engage the crank with the crossfeed screw by pushing the bottom portion of the crank toward the screw. This permits the teeth on the crank, which are normally kept from contact by spring pressure, to engage the teeth on the feed screw shaft and allows the crank to turn the crossfeed screw.

1-15. *Rapid traverse.* The *rapid traverse lever* permits you to move the table rapidly by power. You use the rapid traverse to move work back to its original location before you take another cut, or when you are setting up the work or tool. Do not use it while cutting operations are in progress. Point the directional feed control in the direction in which the table is to move and engage the rapid traverse by lifting up on the rapid traverse lever. Do not operate the rapid traverse with the ram in motion. The table may not stop exactly where you intend for it to stop and it may coast into the tool. CAUTION: Remove the hand traverse crank from the crossfeed screw before you engage the rapid traverse. The crank may bind on the shaft and rotate, and you might get hurt.

1-16. *Toolhead.* The *toolhead,* which is located at the front of the ram, consists of the *toolslide* and the *clapper box.* The base of the toolhead is graduated in degrees, as shown in figure 5. Loosening the toolhead locknuts permits you to swivel and position the toolhead at any desired angle. You generally use the toolhead in the vertical position.

1-17. The *toolslide* is part of the toolhead. You use it primarily to control the depth of cut, which is shown in figure 4. Turning the toolslide handcrank raises or lowers the toolslide. A collar, which is graduated in thousandths of an inch, as shown in figure 5, and which is held on the toolslide screw by a friction spring, indicates the travel of the toolslide. The friction spring allows you to position the collar at zero. The toolslide serves the same purpose as both the lathe compound rest and cross slide, and you use it in the same manner. A lock screw permits you to secure the toolslide at any setting you desire.

1-18. Support the toolslide as much as possible by the toolhead to insure a rigid tooling setup.

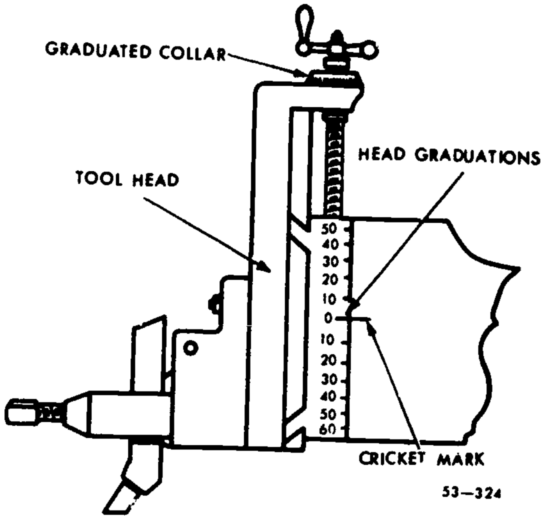


Figure 5. Toolhead graduations.

Figure 6 shows proper and excessive toolhead overhang. You can raise the table by loosening the table locknuts and rotating the table elevating screw with a wrench to position the work near th toolhead in order to avoid excessive toolhead overhang. Be sure to tighten the table locknuts after you have elevated the table.

1-19. The depth of cut you select depends upon (1) the operation being performed, (2) the finish desired, (3) the amount of material that is to be removed, (4) the type of tool used, (5) the type of metal being machined, (6) the rate of table feed, and (7) the type and condition of the shaper. You should plan the work so that equal amounts of material are removed from both sides ot the work to avoid leaving surface imperfections on one side of the work. Your calculations for the roughing cuts should take into consideration the material to be left for th finishing cut, which is normally from 0.010 inch to 0.020 inch. Finish cuts made with a shear tool should not exceed 0.005 inch in depth. The roughing cuts may be as deep as the tool, work, and setup permit. Do not make a cut deeper than 0.125 inch until you are thoroughly familiar with the shaper.

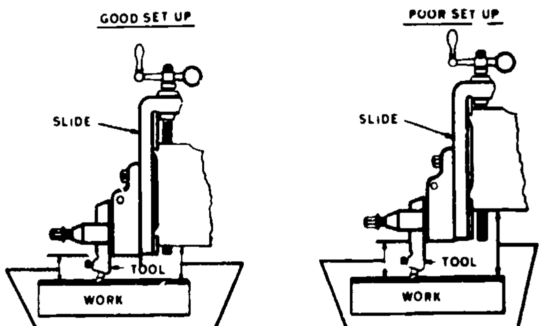


Figure 6. Tool and toolhead overhang.

1-20. Let us assume that you are shaping a piece of metal 1.500 inches thick. It is to be finish machined to a thickness of 1.300 inches. How deep should the roughing cut be? You should calculate the depth of the cut in the following manner:

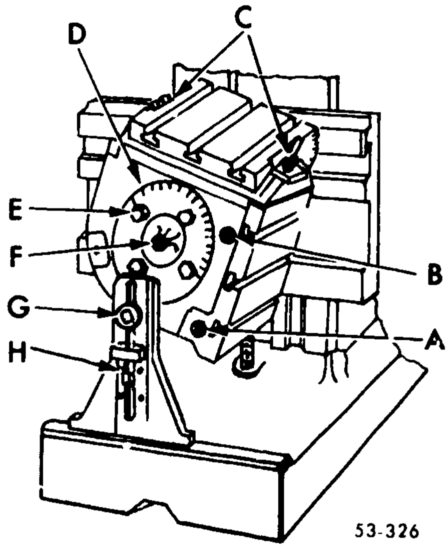
Original size of material	1.500 inches
Finished size of object	−1.300 inches
Stock to be removed	0.200 inch
Material left for finishing cuts (0.015 each side)	−0.030 inch
Stock to be removed by roughing cuts	0.170 inch
Stock to be removed from each side by roughing cuts	$\frac{.170}{2} = 0.085$ inch

1-21. The *clapper box* is mounted on the toolslide. The toolpost is mounted in the clapper box. The clapper box is hinged at the top to allow the cutting tool to raise slightly during the return stroke. You can tilt the clapper box to the left or to the right, or you may leave it vertical. The operation you are performing and the direction of the cut determine the direction in which you tilt the clapper box. Position the top of the clapper box opposite the direction of the cut. Doing so permits the cutting tool to swing away from the cut during the return stroke, prevents marring the w rk, and also avoids unnecessary wear on the cutting edge of the tool. Before moving the clapper box, loosen the locknuts. NOTE: Do not forget to tighten the locknuts after you have moved the clapper box.

1-22. *Table, Vise, and Toolhead Alinement.* You can machine work surfaces that are parallel and perpendicular to each other if the table, vise, and toolhead are properly alined in relationship to the ram.

1-23. There are two types of tables: the plain and the universal. The plain type can only be moved in vertical and horizontal directions. The universal type not only can be moved in these directions but also can be swiveled and tilted, as shown in figure 7.

1-24. Table alinement, to a large extent, determines the accuracy of shaper work. Before alining the table, clean the surface and remove any burrs with an oilstone. Check the alinement with dial indicator. Readings taken at the four corners of the table, as shown in figure 8, must be identical. If the readings are not identical, the table must be alined horizontally from left to right and from front to rear.



A. Swivel control
B. Tilting control
C. Tilting surface clamps
D. Graduated front of trunnion
E. Swivel table clamping nuts
F. Front hand feed control (when furnished)
G. Table support clamp
H. Table support lock

Figure 7. Universal shaper table.

1-25. To aline the solid face of the table, you should first loosen the swivel table clamping nuts. Next, turn the swivel control shaft to aline the cricket mark on the table with the zero graduation on the front of the trunnion, as shown in figure 8. Because this gives only an approximate alinement, you must use a dial indicator for accurate alinement. Mount the dial indicator in the tool post. Then move the toolslide down until the indicator plunger contacts the table surface and the dial reads zero. Now, move the table with the hand traverse crank, as shown in figure 2, until the opposite side of the table is positioned under the indicator. NOTE: Lift the plunger of the dial indicator with your finger to prevent it from dropping into the T-slots. Allow the plunger to gently contact the surface of the table, and then read the indicator. If the reading is zero, the table is properly alined from left to right; if the reading is not zero, you must continue alining the table. Remember the indicator reading, and swivel the table until you obtain an indicator reading that is halfway between this reading and zero. Again, set the indicator to zero, move the table back to the opposite side, and take a second reading. Repeat this process until you obtain identical readings on both sides of the table. The table is now alined horizontally from left to right, but not from front to rear. To check that alinement, first place the indicator plunger in contact with the table near the back edge and set the dial at zero. Move the ram forward by rotating the ram positioning shaft, as shown in figures 2 and 3. Take a reading near the front edge of the table. The readings should be identical at both locations. If they are not identical, you should check the bearing surfaces on the column front and the crossrail for cleanliness and the gibs for proper adjustment. Now tighten the table clamp nuts. To make sure that the table did not move out of alinement while the clamp nuts were being tightened, take indicator readings at the four corners.

1-26. You aline the *tilting surface* horizontally from left to right in the same way in which you aline the solid face. Make the front to rear alinement by tilting the movable surface. Loosen the tilting surface clamps, shown in figures 7 and 9. Tilt the surface by rotating the tilt control shaft until you obtain zero-zero readings at the front and rear when the ram is extended. Figure 9 shows the alinement of the tilting surface. After you have become experienced in operating a shaper, you can use power to move the ram while you check the front-to-rear alinement. When you use power, set the length of the stroke to slightly less than the length of the shaper table. Then, position the ram so that the indicator plunger will not travel beyond the edges of the table and operate the shaper at a very slow speed.

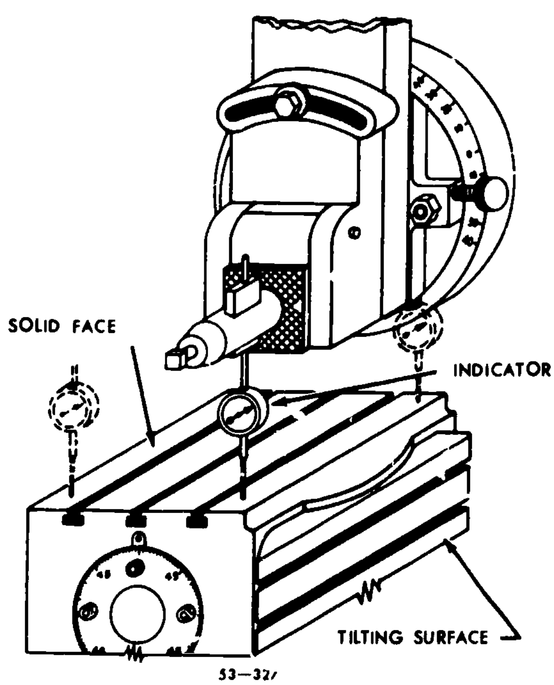


Figure 8 Alining solid face of universal shaper table.

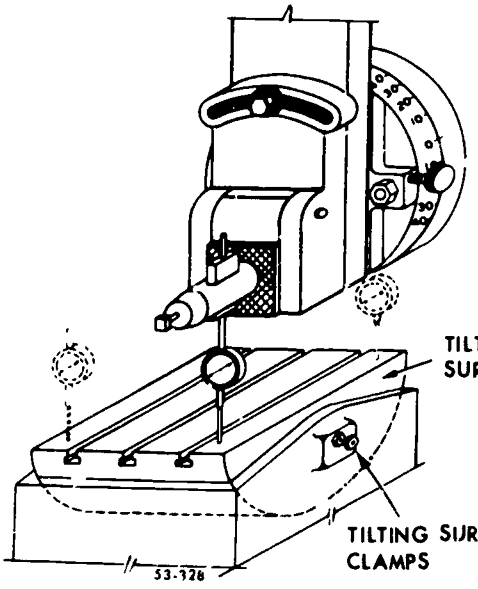


Figure 9. Alining tilting surface.

1-27. Accurate vise alinement is required to produce accurately squared surfaces when work is held in a vise. The bottom surface of the work-holding portion of the vise must be parallel to the stroke of the ram. Check the bottom surface of the vise and aline it by the same method that is used in alining the surface of the table. Before mounting the vise on the table, clean and deburr the table surface and the surface of the vise base that contacts it. Mount the vise on the table and swivel it to the position in which it will be used. Some jobs require the vise jaws to be perpendicular to the stroke. Figure 10 shows the dial indicator position for alining the jaws parallel and perpendicular to the travel of the ram. Use the *crossfeed screw,* as shown in figure 2, to move the table when you are checking alinement in the perpendicular position. Use the *ram-positioning shaft* to move the ram when you are

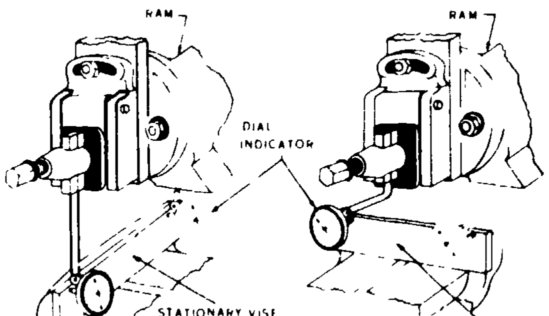


Figure 10 Alining vise jaw

checking alinement in the parallel position. Make necessary adjustments by swiveling the vise. CAUTION: Most shaper vises are extremely heavy and awkward to handle. Do not attempt to lift them by yourself. Clamp a suitable piece of stock in the vise to serve as handles to lift the vise.

1-28. Proper toolhead alinement requires that the toolhead be perpendicular to the surface of the table or to the inner face of the stationary vise jaw. This will insure that the graduations on the toolhead micrometer collar indicate the exact amount of vertical travel. It will also insure that vertical surfaces produed by feeding the tool with the toolslide are perpendicular to the table or vise. To aline the toolhead, the table surface or the vise bottom surface should first be alined in the manner just discussed. Next, clamp a box parallel, angle plate, or other object that has accurate perpendicular surfaces to the table or the bottom of the vise. Place the toolhead is the vertical position. Use the graduated scale on the toolhead and the cricket mark on the ram for an approximate alinement, as shown in figure 5. Now mount a dial indicator in the toolholder with the plunger contacting the vertical surface of the box parallel near the top edge. Set the dial to zero reading and crank the toolslide down. The dial readings at the top and bottom of the vertical surfaces should be identical. If the readings are not zero-zero, swivel the toolslide slightly in the necessary direction until they are.

2. Workholding Devices

2-1. The shaper is used to machine work that varies greatly in size, shape, and machining requirements. A knowledge of some of the workholding devices will help you determine the correct work setups to use. Work being machined in a shaper may be held or clamped by different methods. It may be held in a vise or between shaper index centers. It may be clamped to an adjustable or nonadjustable angle plate or fastened directly to the shaper table with one of several clamping devices. Work may also be held in special fixtures.

2-2. **Vises.** There are two common types of *shaper vises:* the double-screw and the single-screw, as shown in figure 11. The *double-screw vise* is stronger and has greater holding power than the single-screw vise. Slightly tapered workpieces may be held in the double-screw vise without using extra jaws or shims. The *single-screw vise* has a faster clamping action. Most shaper vises may be swiveled or rotated in a horizontal plane. This makes it possible for you to position the jaws either parallel or perpendicular to the

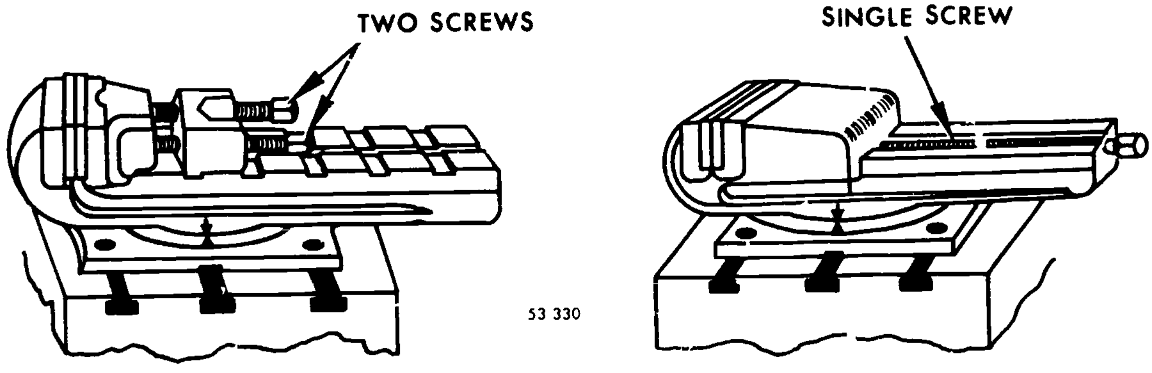


Figure 11. Double and single screw vises.

stroke of the ram, or to any desired angle in between. The base of the vise is graduated in degrees. These graduations indicate the angle to which the vise is swiveled. You can swivel the vise without removing it from the machine.

2-3. **Clamps.** You may use a variety of *clamps* to fasten work to the table of a shaper. The type of clamp that you use will depend upon the shape and size of the work to be clamped. Use enough clamps to insure that the work will not move during the machining operation. Figure 12 illustrates and identifies the most common clamps. Figure 13 illustrates good and bad clamping practices.

2-4. **Parallels.** *Parallels* are bars of hardened tool steel whose opposite surfaces are parallel to each other. You use them to support the work at a convenient height while you maintain parallelism between the bottom of the work and the shaper table or vise. Figure 14 shows the most common type of parallel, a *solid parallel.* Figure 15 shows an *adjustable parallel.* You can vary the height of the adjustable parallel by loosening the lockscrew and sliding the upper portion in the dovetail slot. *Degree parallels,* shown in figure 16, are similar to solid parallels except that one side is machined at an angle.

2-5. **Angle Plates.** There are two common types of *angle plates:* the standard (nonadjustable) and the adjustable. Angle plates are fastened to the table of the shaper, and the work is fastened to the angle plate. The *standard angle plate,* shown in figure 17, has two outer surfaces perpendicular to each other which permit you to mount work at an angle of 90° to the shaper table. You can raise the table on an *adjustable angle plate,* shown in figure 18, and lock it in the horizontal position, the vertical position, or any position in between. You can also rotate the adjustable angle plate on its base. This permits you to perform light machining operations on two axes.

2-6. **Holddown Straps.** You use *holddown straps* to hold thin objects flat against the table or parallel, as shown in figure 19. You generally use them when you want to remove a small amount of material from the surface of the work and when other clamping methods would be impractical. The contacting edges of holddown straps are machined at a slight angle, which causes them to force the work down tightly against the table surface or the parallels, as shown in figure 19.

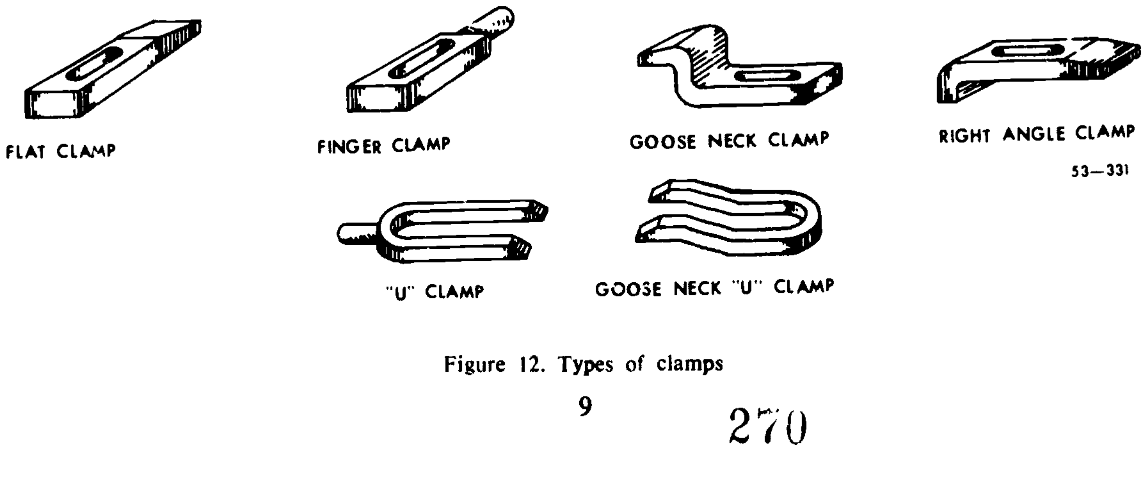


Figure 12. Types of clamps

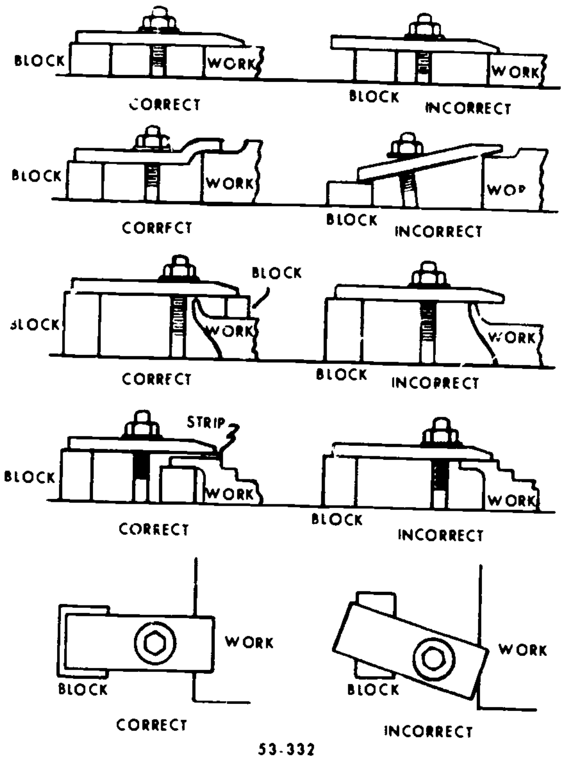


Figure 13. Clamping work.

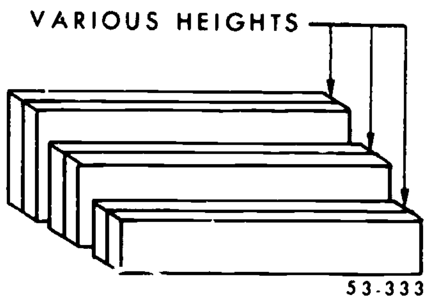


Figure 14 Common parallel

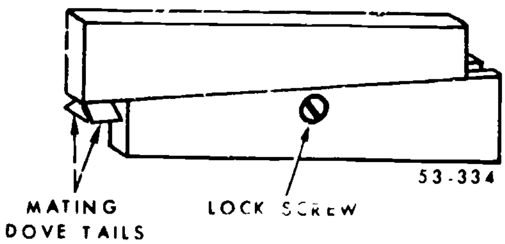


Figure 15 Adjustable parallel

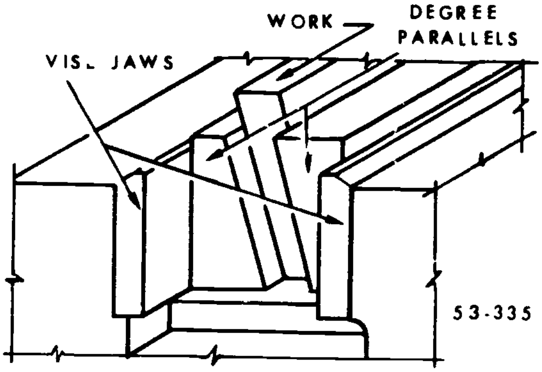


Figure 16. Degree parallels.

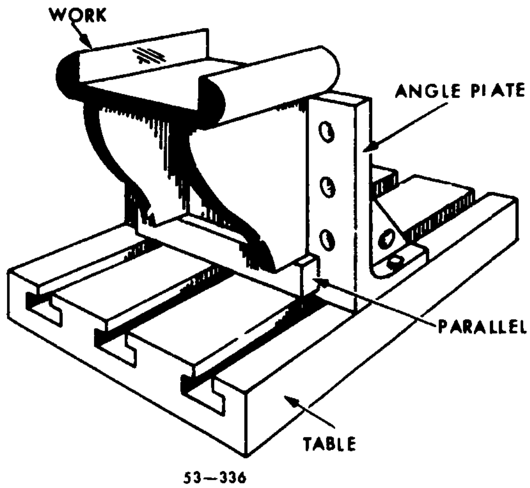


Figure 17 Nonadjustable angle plate.

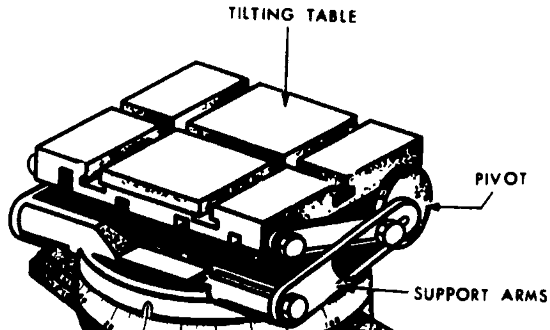


Figure 18. Adjustable angle plate

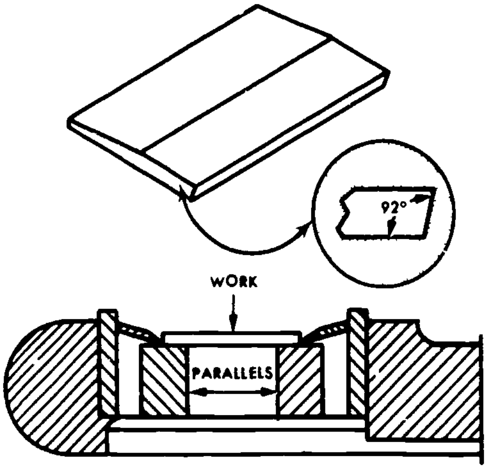


Figure 19. Holddown straps

2-7. **Bunters and Toe Dogs.** You use *bunters*, shown in figure 20, to prevent work from shifting sideways during machining operations. Insert the hooked portion of the bunter in the T-slot of the table and use the threaded screw to lock the work and bunter in position. You can use *toe dogs*, shown in figure 21, with bunters to hold thin work on the table. Figure 22 shows correct and incorrect ways to use toe dogs.

2-8. **Jacks.** *Jacks*, such as the one shown in figure 23A, are used to level work and to support portions that could spring down while they are being machined. Figure 23B shows the jack adjusting screw. You can use extension bases, figures 23C and D, to extend the effective height of the jack. Adapters, as shown in figures 23E and F, permit you to use the jack on work of varying contours.

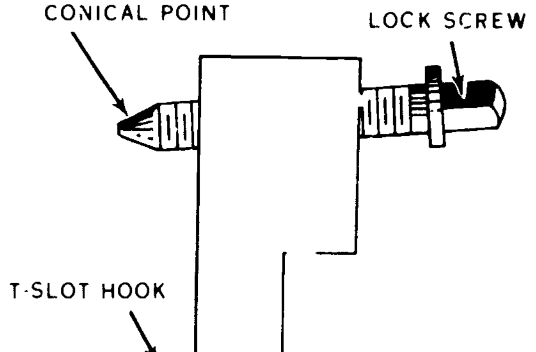


Figure 20. Bunter

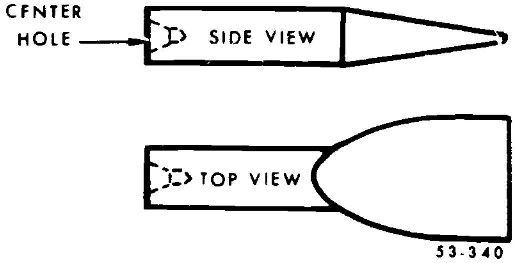


Figure 21. Toe dogs.

3. Shaper Cutting Tools and Toolholders

3-1. Many types of cutting tools are required to perform work on a shaper. We will now describe different cutting tools and different types of toolholders.

3-2. **Shaper Tools.** *Shaper tools* are ground from the same tool steels that are used for lathe tools, but they are usually ground from 3⁄8-inch- by 3⁄8-inch-(or larger) tool blanks. Shaper tools usually have less effective end relief and side relief than lathe tools. You can grind shaper tools as either left- or right-hand tools.

3-3. We will discuss below some of the common types of shaper tools.

a. Roughing. The *roughing tool*, also called a *bullnose roughing tool*, is intended for heavy-duty cutting. It will withstand the pressures of heavy feeds and depths of cut. Figure 24 shows the various angles used for a left-hand roughing tool. Left- and right-hand roughing tools can be ground with a smaller nose radius and used for finishing vertical surfaces, as shown in figure 25.

b. Steel shear. Steel can be more highly finished with a *shear tool* than with any other shaper tool. Use the steel shear tool, shown in

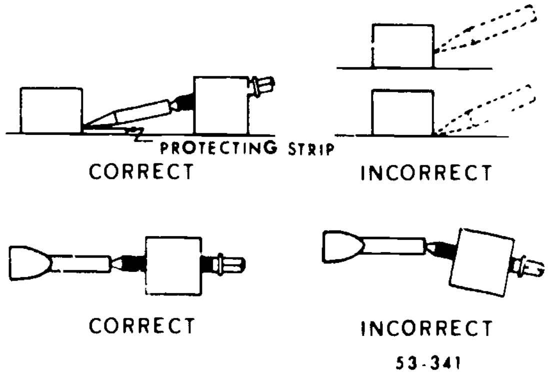


Figure 22 Use of toe dogs

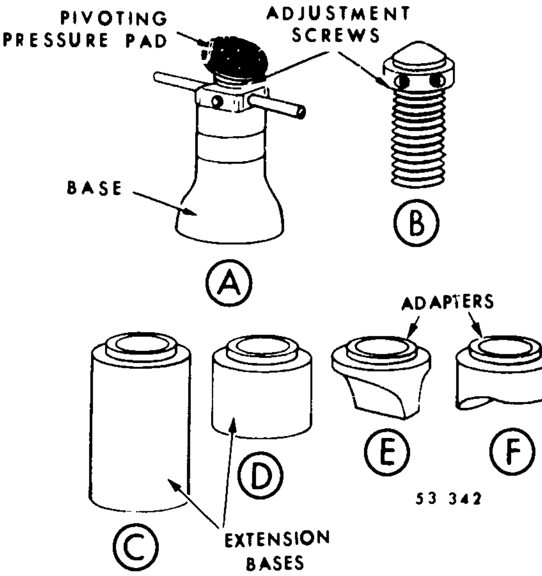


Figure 23. Jack and extension bases.

figure 26, with a depth of cut less than 0.005 inch and the finest possible feed for the best results. Figure 27 shows the cutting action of a shear tool.

c. Aluminum shear. Use the *aluminum shear tool*, shown in figure 28, for finishing cuts on aluminum.

d. Roundnose. The *roundnose tool*, shown in figure 29, has 0° side rake and cutting edges on both sides. These enable it to cut in both directions. You can use it to rough out metal between two shoulders, as shown in figure 30.

e. Squaring. The *squaring tool*, shown in figure 31, has sharp corners on the ends of the cutting edge which allow it to produce 90° corners. You use it primarily to finish the bottom and sides of shoulders, keyways, and grooves, as shown in figure 32.

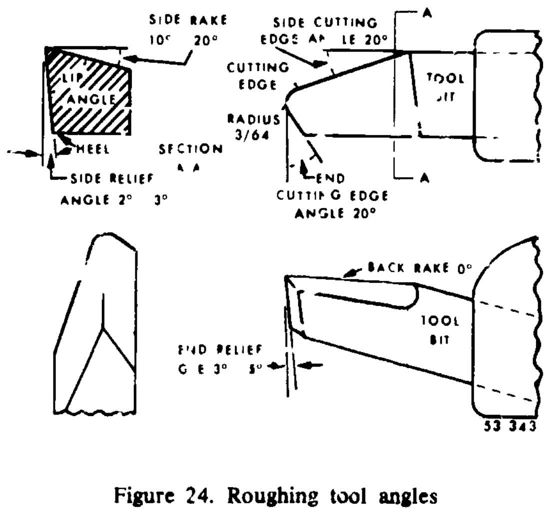


Figure 24. Roughing tool angles

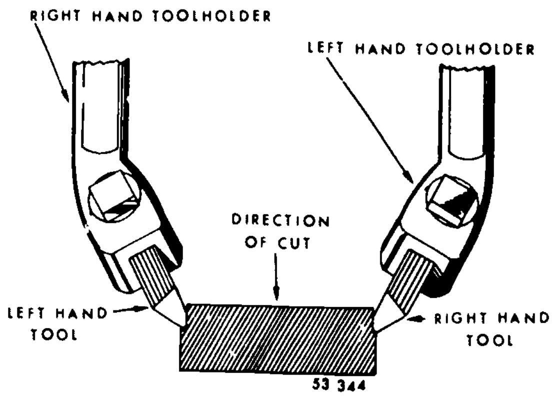


Figure 25. Down cutting tools.

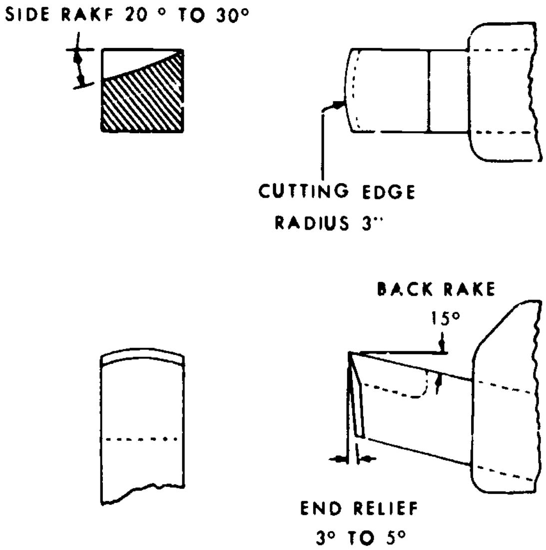


Figure 26. Shear tool for steel.

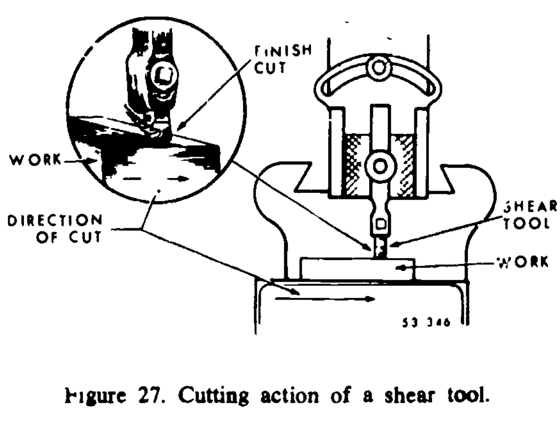


Figure 27. Cutting action of a shear tool.

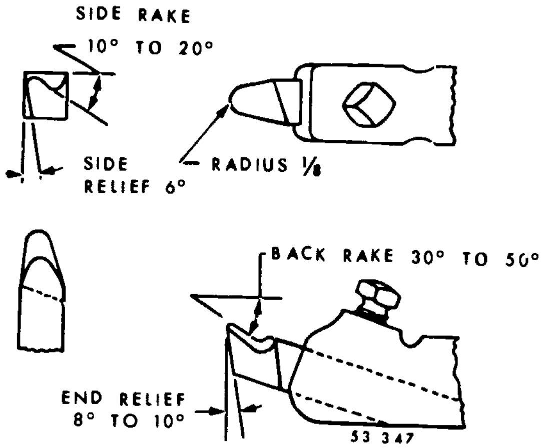


Figure 28 Aluminum shear tool

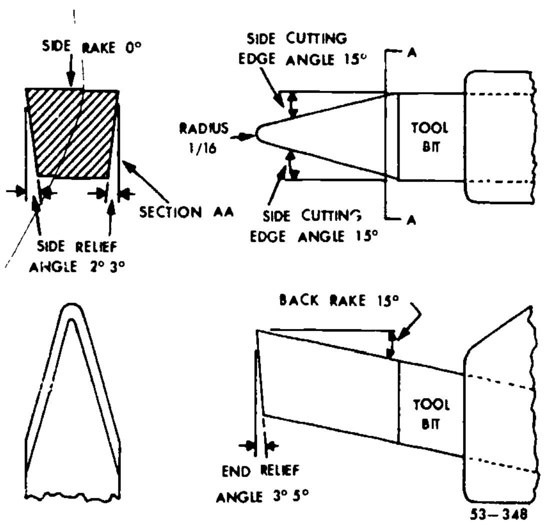


Figure 29 Roundnose tool angles

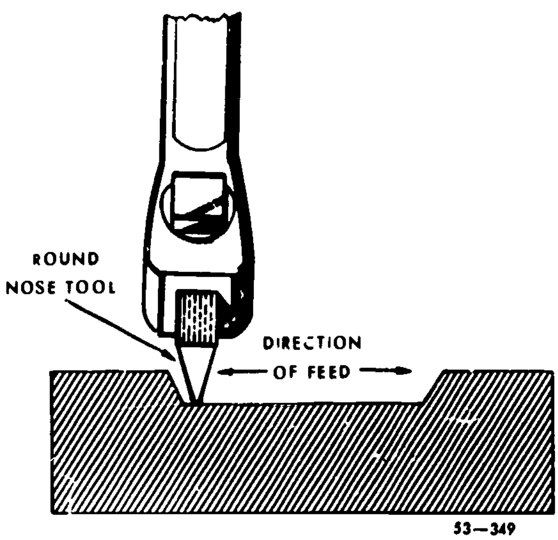


Figure 30. Roundnose tool.

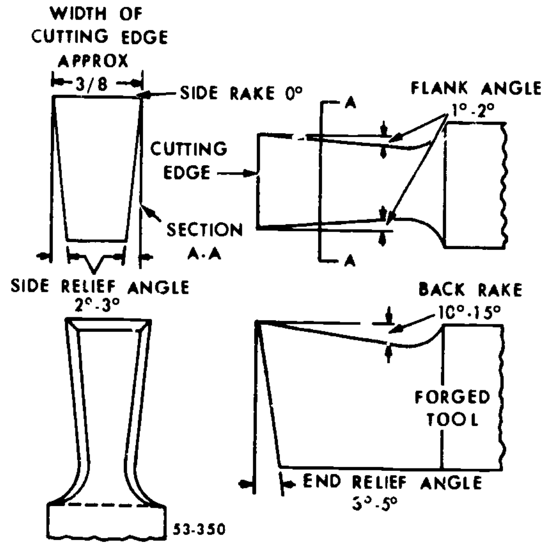


Figure 31 Squaring tool angles.

f. Shovelnose. The *shovelnose tool* is similar to the squaring tool, except that the ends of the cutting edge are rounded slightly and the back rake angle is reduced. You can use it either as a right-hand or left-hand tool. You can use it to machine vertical surfaces, as shown in figure 33.

g. Parting. The *parting tool* is similar to the squaring tool, except that it is much narrower. You use it to part work, as shown in figure 34.

h. Side finishing. The *side-finishing tools*, shown in figure 35, are ground as right-hand and left-hand tools and are used to finish vertical surfaces. You can also use them for cutting or finishing small horizontal shoulders after you have taken a vertical cut. Figure 36 shows how to position right-hand and left-hand side-finishing tools.

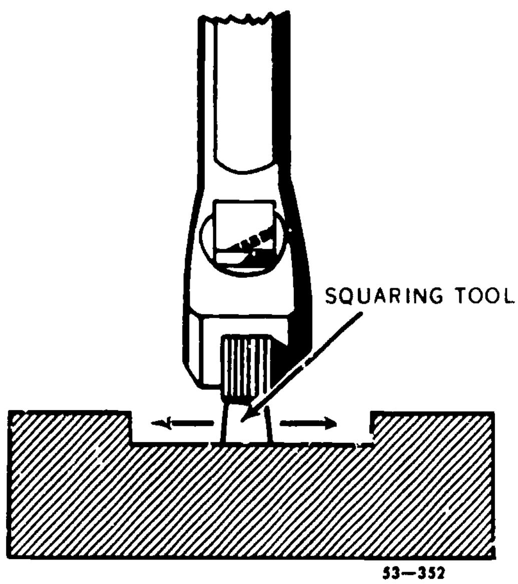


Figure 32. Squaring tool.

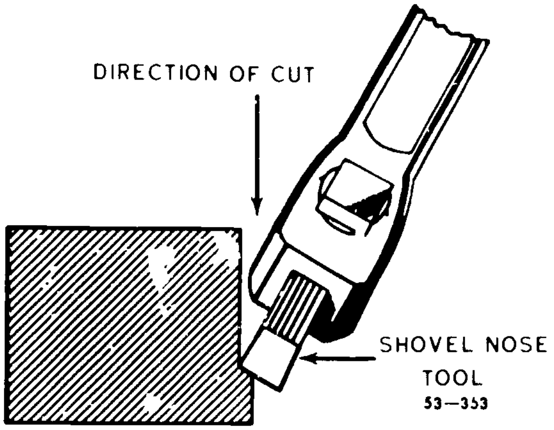


Figure 33 Shovelnose tool

i. Angle cutting. *Angle cutting tools* are used to finish angular surfaces, such as dovetails, shown in figure 37. You use them after you have roughed out the angle with a roundnose tool. You can grind them as right- or left-hand tools, as shown in figure 37.

j. Form. *Form tools* may be used to cut a specific shape, such as concave or convex radii, V-grooves, etc. They are usually ground with 0° back and side rake. You should rough out the form with a roundnose tool and use the form tool to finish the form.

k. Gooseneck. The *gooseneck tool,* shown in figure 38, is forged so that the cutting edge is behind the back side of the tool shank. This allows the tool to spring away from the work slightly, reducing the tendency of a tool to gouge or chatter when you are machining cast iron. The cutting edge should have a slight back rake and should be slightly rounded at the corners.

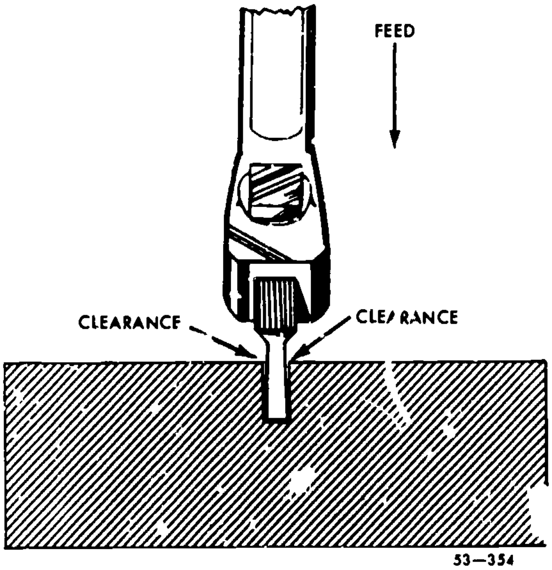


Figure 34. Cutting off tool.

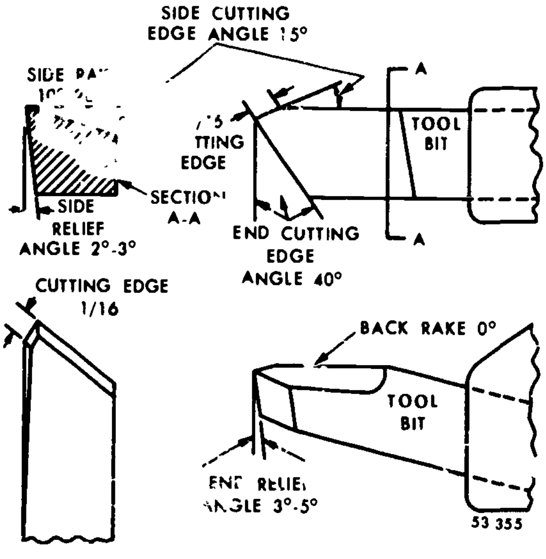


Figure 35. Side finishing tool angles.

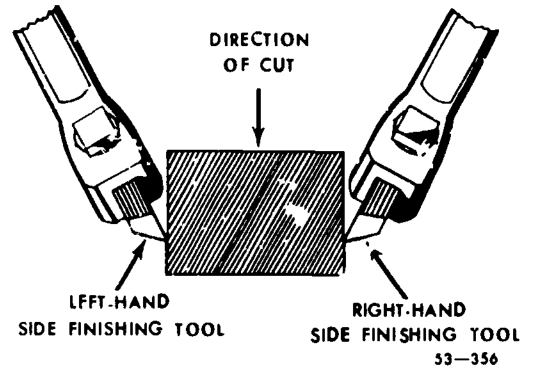


Figure 36. Side finishing tools

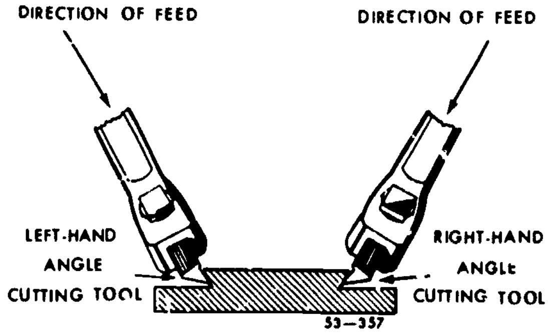


Figure 37. Angle cutting tools.

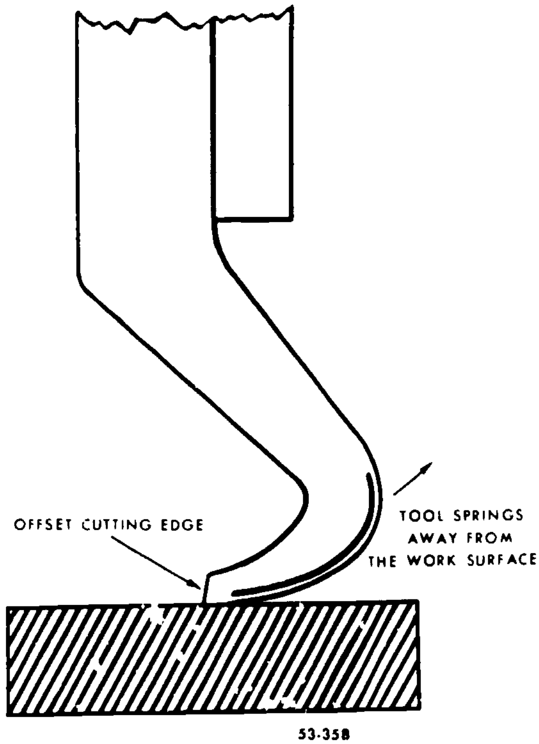


Figure 38 Gooseneck tools

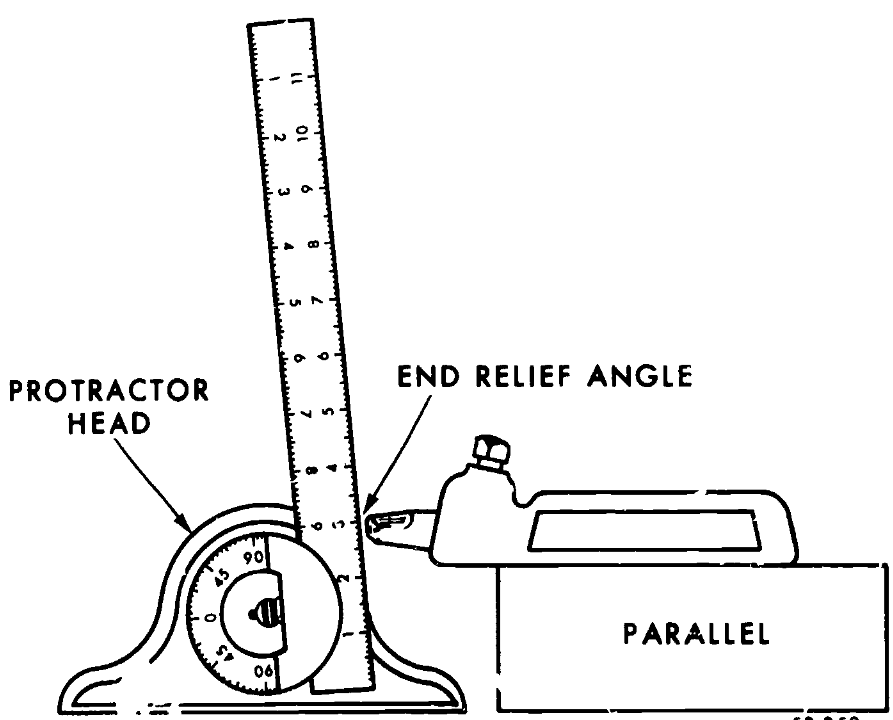


Figure 39. Checking end relief angle.

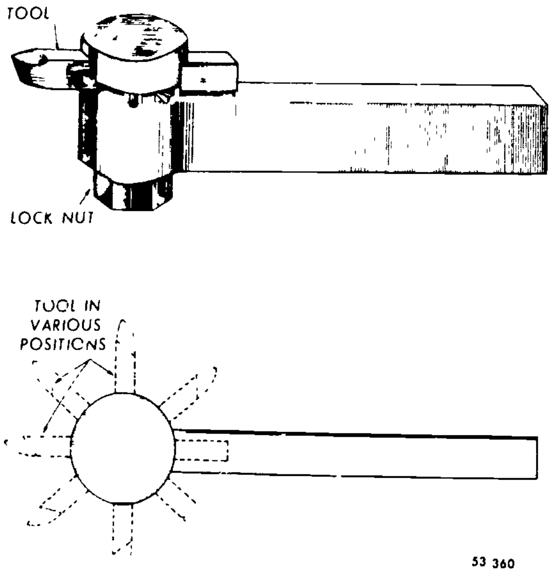


Figure 40. Swivel head toolholder.

3-4. **Toolholders.** A variety of toolholders are used for shaper work. First, we will discuss some of the common toolholders and then some of the general tooling setup recommendations.

3-5. The most common types of toolholders used in shaper work are the straight, the left-hand, and the right-hand toolholders that are identical to the common lathe toolholders. The straight toolholder is used more frequently in shaper work than in lathe work. The tool is

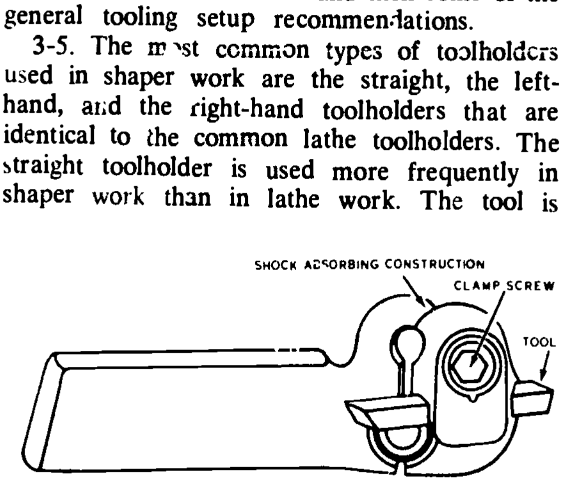


Figure 41 Spring toolholder

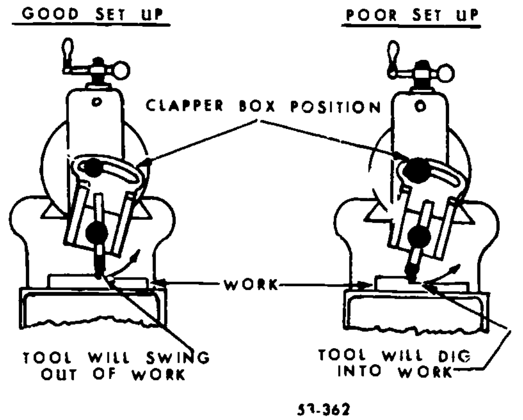


Figure 42. Positioning the tool and clapper box.

held in the toolholder at an angle of approximately 14° Do not forget to compensate for this angle when you are grinding the end relief and the back rake angles. Figure 39 shows how you can measure the effective end relief angle with a protractor head and blade by placing the toolholder on a parallel.

3-6. *Swivel head* toolholders, such as the one shown in figure 40, can be used as a straight, right-hand, or left-hand toolholder. *Spring* toolholders, such as the one shown in figure 41, are designed to absorb vibration. They are used with forming tools since they reduce the tendency of the tool to chatter. You can also use them when you machine cast iron for the same reason. *Extension* toolholders are used when you machine internal splines, keyways, etc. They are similar in construction and use to the boring bars used in lathe work.

3-7. The overhang of the tool and the toolslide should be held to a minimum to insure the most rigid tooling setup possible. This is shown in figure 6. Whenever possible, position the cutting edge of the tool directly below the vertical centerline of the toolpost, as shown in figure 42. This will permit the tool to swing away from the surface that you are machining if the toolholder should move.

Planing Operations

ALTHOUGH THE CUTTING stroke of the shaper is restricted to one direction of travel, there are few restrictions on the types of work that can be done on a shaper. All styles of flat and contour surfaces can be produced. as well as special shapes such as slots and keyseats. In fact, the only restrictions on shaper work are the maximum length of stroke and the capacity of workholding devices. The first section of this chapter covers the machining of flat and contoured surfaces; the second covers the machining of special shapes.

4. Machining Horizontal, Vertical, Angular, and Contoured Surfaces

4-1. In this section, we will discuss the machining of horizontal, vertical, angular, and contoured surfaces. It is important to understand the work and tool setups involved. We will explain these as we discuss the operations.

4-2. **Horizontal Surfaces.** Horizontal surfaces are surfaces that are machined in a horizontal plane. The surfaces may or may not be in a horizontal position in actual use. You machine a horizontal surface by feeding the work laterally below the ram, as shown in figure 4. The direction of the feed is usually from left to right, as shown in figure 4, to permit an unobstructed view of the work surface and the cutting action of the tool. CAUTION: Eye protection should always be worn when you are operating a shaper.

4-3. Use a roughing tool and roughing feed to remove excess stock. Take the finish cut with a roughing tool, using a finishing feed, or with a shear tool, using a finishing feed. Use the cutting lubricant that is recommended for the metal that you are machining during all cutting operations. Apply the lubricant to the work surface with a brush or an oilcan. Do not permit the brush or the oilcan spout to get caught between the cutting tool and the work.

4-4. *Machining surfaces square and parallel.* You can machine surfaces that are square and parallel to each other by machining them horizontally in the proper sequence. The following information will help you to understand how to machine work so that the opposite surfaces are parallel and the adjacent surfaces are square. Figure 43 shows the position of the work as the various sides are machined.

4-5. You should first aline the vise so that the stationary jaw and bottom of the vise are parallel to the ram travel. Obtain the proper size and type of material and mount it in the vise on the correct size parallels. The work should extend above the vise jaws far enough to permit removal of the necessary stock without damage to the vise.

4-6. If the workpiece has a rough surface, use soft copper or brass shim stock between the work and vise jaws, and the work and parallels. The shim stock protects the vise jaws and the parallels from the rough material. It also compensates for uneveness in the work and gives the vise greater holding power. On work that has smooth or machined surfaces, shim stock is required only between the work and the movable vise jaw. This permits the work to seat on the parallels and also protects the finished surface. To seat work on the parallels, tap it lightly with a soft hammer *after* you have tightened the vise jaws. If you tighten the vise jaws after you have seated the work on the parallels, the work may be unseated. Burrs and sharp edges left by planing should be filed off before removing the work for planing the next surface.

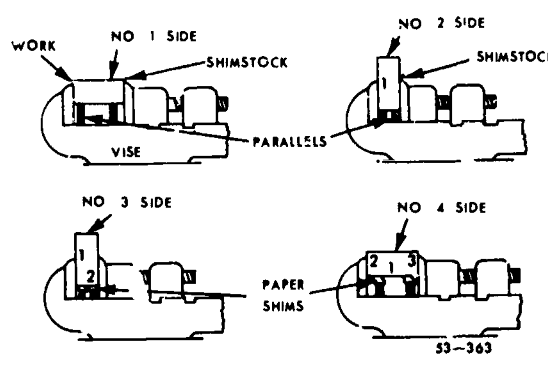


Figure 43. Squaring a block.

4-7. Tilt the clapper box and mount the roughing tool and the toolholder in the toolpost. Calculate the amount of stock to be removed from side 1. Set the length of the stroke, position the ram, and set the rate of feed and speed Rough machine side 1. If an exceptionally good finish is not required, you can use the roughing tool for the finish cut. Otherwise, replace the roughing tool with the shear tool. The shaper should be set for finish speed and feed no matter which tool is used. Finish machine side 1. Remove the work, and reposition it in the vise with side 1 in contact with the stationary jaw and side 2 uppermost, as shown in figure 43. Determine the amount of stock to be removed from side 2. Then rough and finish machine side 2. Again, remove the work. Reposition it in the vise with side 1 in contact with the stationary jaw and side 2 in contact with the parallels. Use paper shims to insure seating the work on the parallels, as shown in figure 43. Rough machine side 3, leaving enough stock for the finish cut. Finish machine side 3 so that the distance between sides 2 and 3 is correct. Now place the work in the vise wtih side 1 in contact with the parallels, and side 2 in contact with the stationary jaw, as shown in figure 43. Finally, rough and finish machine side 4 so that the dimension from side 1 to side 4 is correct.

4-8. *Machining ends of work square and to length.* You finish work to length by machining the ends square with the sides. To do this, you should first position the vise so that the jaws are 90° to the stroke of the ram. Next, insert the work upright in the vise. Use a machinist's square to aline the work vertically, as shown in figure 44. Position the ram and set the length of stroke and speed and feed. Determine the amount of stock to be removed from the first end. Rough and finish machine end 1. Remove the parallel and aline it in the vise, as just explained, but

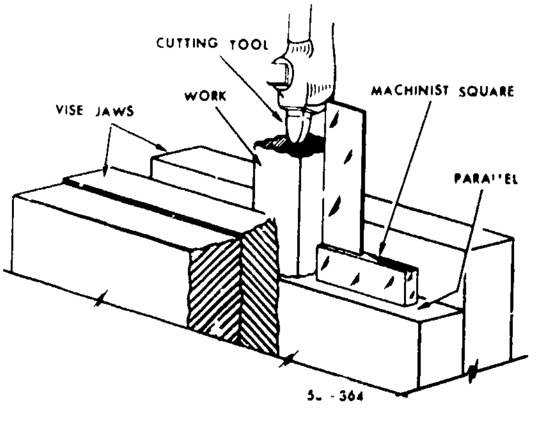


Figure 44 Positioning a short piece in the vise.

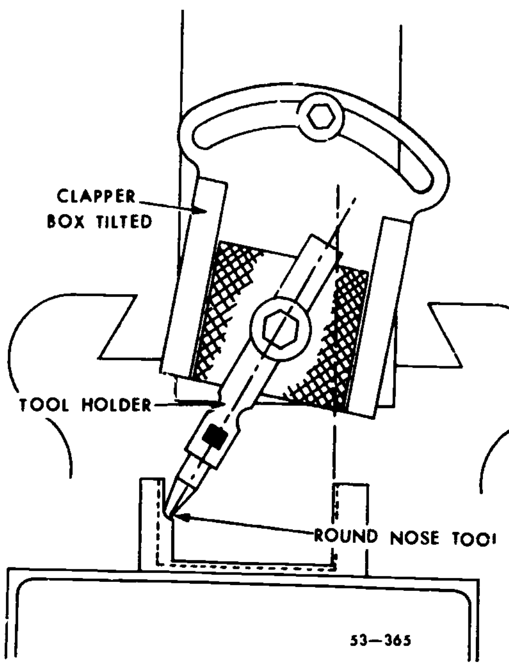


Figure 45. Tilting clapper box for down cutting

with end 2 uppermost. Rough machine end 2. Leave a finishing allowance, and then finish machine end 2 until the desired length is reached.

4-9. **Vertical Surfaces.** When you machine *vertical surfaces,* such as the vertical surfaces of shoulders or the ends of work, position the toolhead square with the table or vise. Position the vise jaws square or parallel to the travel of the ram, depending upon the location of the surface that you are machining. When great accuracy is not required, you can aline the toolhead and the vise by means of their graduations. When greater accuracy is required, use the dial indicator. Hand feed the tool with the toolslide crank and make the depth of cut by moving the work toward the cutting tool. You can use the crossfeed dial graduations to determine the exact depth of cut. Tilt the clapper box away from the surface being machined, as shown in figure 45. You can use a roundnose tool for both the roughing and the finishing cuts when average finishes are permitted. Use side-finishing tools when finer finishes are needed and when you face deep vertical surfaces on shoulders and corners. Use squaring tools to finish the vertical surfaces on shallow shoulders and corners and to finish the sides of deep slots or grooves.

4-10. Generally, you machine vertical surfaces with a roundnose tool. After alining the toolhead and squaring the vise with the table, mount the work in the vise. Swivel the clapper box and mount a roundnose tool in the toolholder. The toolholder should be mounted at a

30° to 45° angle with the side of the work, as shown in figure 46. Position the ram and stroke length and set the machine for proper speed. Pick up the cut and set the tool for depth of cut. The depth of cut can range from $1/32$ to $1/16$ inch, depending on the rigidity of the setup. Start the machine and hand feed the toolslide down a few thousandths of an inch per stroke. The tool should be fed at the end of the return stroke. Machine the vertical surface to the required depth.

4-11. Another method of machining vertical surfaces is to use a side finishing tool. This type of tool is used after the vertical surface has been roughed out with a roundnose tool, leaving material for finishing cuts. The top of the clapper box is swung away from the surface that is to be finished. Then mount a left-hand side finishing tool in a straight toolholder, if you are to finish a left-hand shoulder. The toolholder should be positioned in the toolpost so that the $1/16$ inch portion of the tool's cutting edge, as shown in figure 35, is parallel to the surface to be finished. The best way to position the cutting edge is with a machinist's square, as shown in figure 47. Set the machine to the proper length of stroke Reduce the CFS to about 30 feet per minute to protect the small cutting edge. With the tip of the tool touching the bottom surface of the corner or shoulder, set the vertical feed dial at zero. NOTE: Omit this step when you use the side-finishing tool to finish the ends of the work. To plane wide, deep grooves, leave 0.005 inch

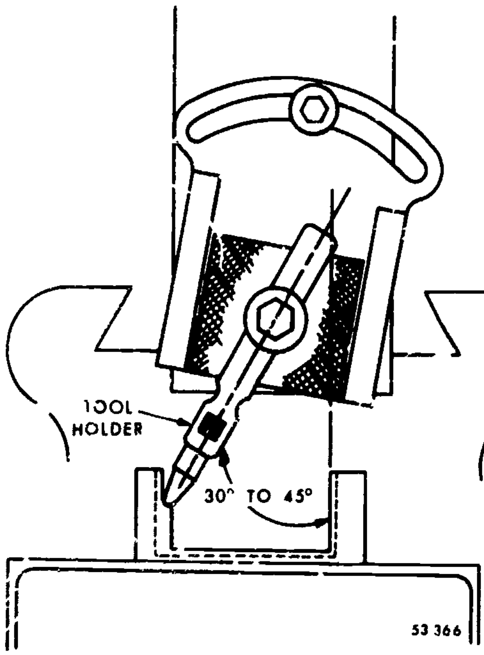


Figure 46. Positioning toolholder for internal vertical surfaces.

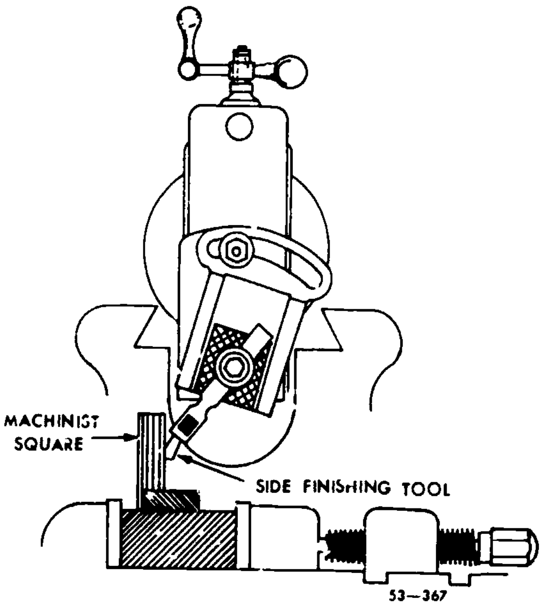


Figure 47 Alining cutting edge with a machinist square.

on the bottom surface for finishing with a squaring tool. Pick up the vertical surface of the work, using a strip of paper. Use vertical cuts to finish the shoulder or end of work to the correct size. Do not exceed a depth of cut of .004 and a feed of .002.

4-12. **Angular Surfaces.** You can machine angular surfaces by any one of five methods: (1) swiveling the vise, (2) swiveling the toolhead, (3) swiveling or tilting the table, (4) mounting the work on an adjustable angle plate, or (5) holding work in a fixture.

4-13. *Swiveling the vise.* If the surface to be machined is perpendicular to the surface of the table (or to the bottom of the vise) and at an angle to the stroke of the ram, you can machine it by simply swiveling the vise. Swivel the vise until the angular surface is parallel to the stroke of the ram. Make the depth of cut by moving the table until the work contacts the cutting tool. Feed the tool vertically by means of the toolslide handcrank. Use the same tooling setup that you use to machine vertical surfaces.

4-14. *Swiveling the toolhead.* The toolhead can be used to machine surfaces that are parallel to the stroke of the ram but at an angle to the vertical centerline of the toolhead. Swivel the toolhead to the required angular setting and set up the tool as you would for horizontal planing. Feed the tool by hand by means of the toolhead crank. Use the toolhead graduations, shown in figure 5, for angles requiring average accuracy. Aline the toolslide with a dial indicator and a sine bar, as shown in figure 48, when you are machining angles that must be highly accurate.

4-15. *Swiveling the table.* You can position the work at the angle you desire by swiveling the table on the trunnion or by tilting the tilting surface. Use the graduations on the table for average accuracy. Use a sine bar, as shown in figure 49, for extreme accuracy. Set up the tool and power feed the work in the same manner you use to machine a horizontal surface. You can machine work on three axes, as shown in figure 50, by swiveling the vise, the table, and the tilting surface.

4-16. *Using an adjustable angle plate.* You can mount work on an adjustable angle plate, such as the one shown in figure 18, in order to machine the work at the angle, or combination of angles, that you desire. You can machine the same angles using an adjustable angle plate that you can machine by swiveling the vise, the table, or the tilting surface. However, you should use the adjustable angle plate for light-duty applications only.

4-17. *Using a fixture.* You can mount work on a fixture, as shown in figure 51, in order to machine an angular surface in the same manner you use to machine a horizontal surface. You can manufacture fixtures that will hold work at any angle you desire. However, the time required to manufacture a fixture is not justified unless the work is of a recurring nature, or unless several identical items are to be machined.

4-18. **Contoured Surfaces.** In addition to flat surfaces which are machined horizontally, vertically, or at an angle, you can also machine irregular or contoured surfaces with a shaper. While horizontal, vertical, or angular surfaces may connect the curved portions of an object, only the curved areas are considered as being contoured. The contour may consist of a single radius, or it may have several curves, such as the contoured surface shown in figure 52.

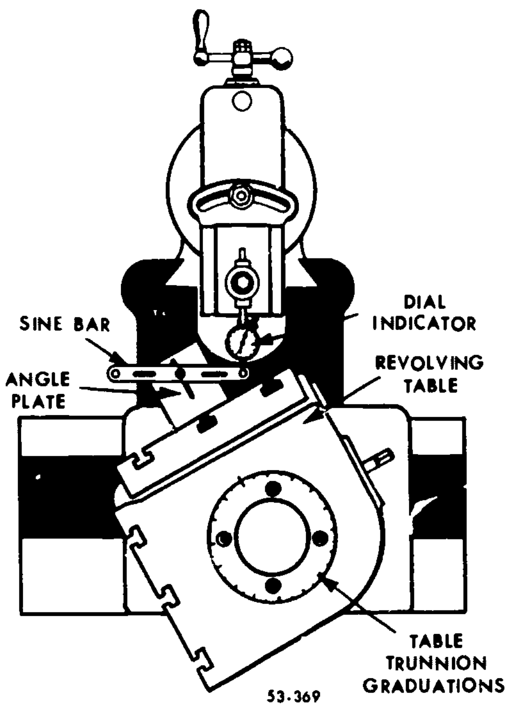


Figure 49. Alining table with a sine bar.

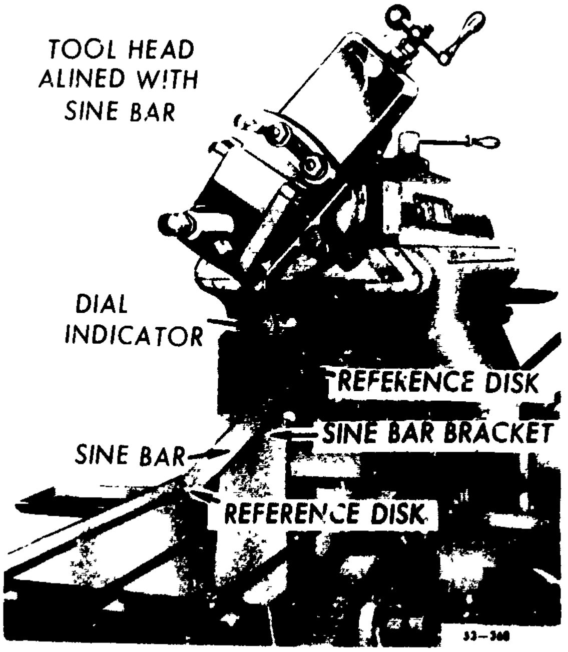


Figure 48. Alining toolhead with a sine bar.

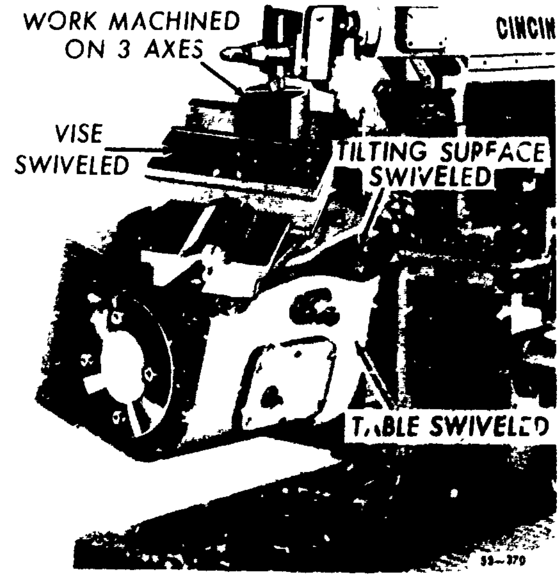


Figure 50. Angular planing on three

281

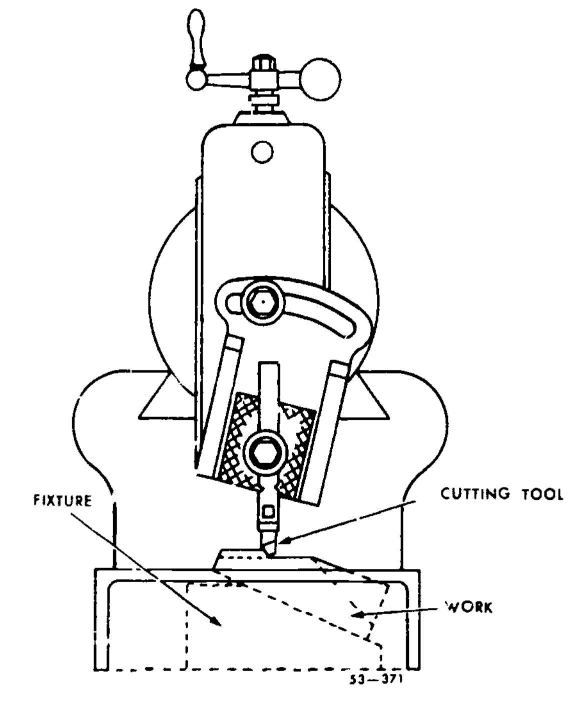


Figure 51. Angular planing with a fixture.

4-10. You can machine contoured surfaces with a shaper by first laying out the contour on the end of the work blank and then cutting the contour with form tools. You can also do it by moving the work and cutting tool at the same time. You normally use form tools to machine several identical items. If the contour is too large to be formed by one tool, you can grind tools to the shape of portions of the contour and then machine the contour by sections. You will obtain the best results by roughing out the contour with a roundnose tool before using the form tools. Contours that are not practical to produce with form tools can be machined by moving the work and the tool simultaneously so that the tool follows the layout line. You can feed both the work and the tool by hand, which requires a great deal of skill and experience, or you can feed the work by power and the tool by hand. You can machine a fairly accurate and uniform contour with this method, since it enables you to concentrate mainly on feeding the tool. Use a roundnose tool to machine the contour. NOTE: Place the clapper box in the vertical position when you machine a contour. If necessary, you can file and polish the contour after machining it to improve its shape and finish.

5. Machining Shoulders, Slots and Grooves, and Keyseats

5-1. In this section, we will discuss the machining of shoulders, slots and grooves, and keyseats. We will also discuss the work and tool setups involved in these operations.

5-2. **Shoulders.** Square shoulders are machined by removing excess material with a roundnose tool. The surfaces are then finished with a side-finishing tool or a squaring tool.

5-3. *Removing excess material.* You can remove excess material by making a series of horizontal cuts toward the layout line or shoulder. When you are roughing out material between two shoulders, feed the work in each direction of successive cuts. This will save time, since it will not be necessary for you to return the work to a starting point for each cut. When you are roughing between shoulders, the clapper box should be positioned centrally and the straight toolholder set vertically, as shown in figure 53. Set up the tool as shown in figure 54 when you are machining to one shoulder. Make each succeeding cut slightly shorter than the preceding cut, leaving a stepped fillet in the corner. Tilt the toolholder to permit machining closer to the shoulder and to keep the toolholder from interfering with the cut.

5-4. After mounting a roundnose tool in a straight toolholder, set the machine for the length and position of stroke and the correct speed. Make sure that the ram and tool slide clear the work and vise. Pick up the cut with a piece of paper, and set the toolslide dial at zero. With the feed set for roughing, begin the cut approximately $\frac{1}{32}$ inch from the vertical layout line. The depth of cut should not exceed 0.075 inch. You can use the power feed to machine within $\frac{1}{8}$ inch of the layout line and then feed by hand until the tool is within $\frac{1}{6}$ of the preceding cut. CAUTION: *Be sure that the toolholder continues to clear the work as the depth*

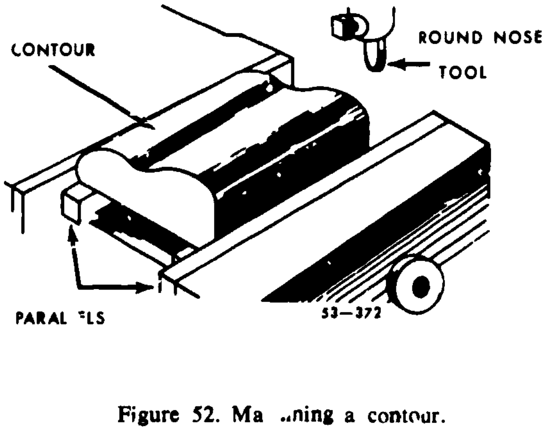


Figure 52. Machining a contour.

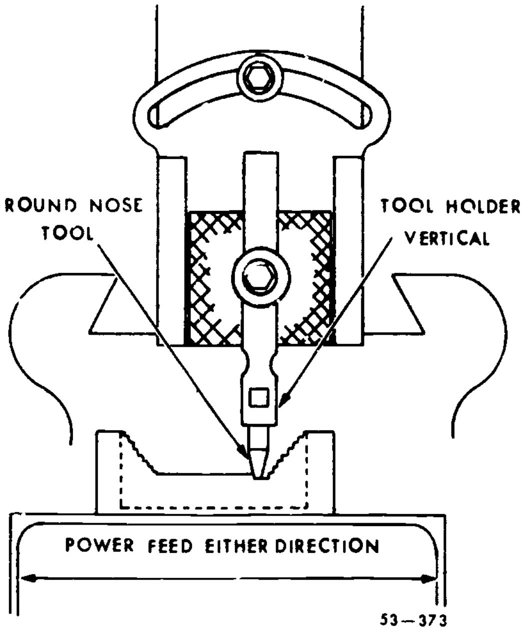


Figure 53 Roughing excess between two shoulders.

of the shoulder increases. Repeat this process until you obtain the depth you desire

5-5. You can measure the height of the shoulder with a depth micrometer. Be sure that you remove all burrs from the shoulder before you make the measurement. Leave approximately 0.015 inch for additional cuts on the horizontal surface. Now swing the clapper box away from the shoulder and position the tool at a 30° to

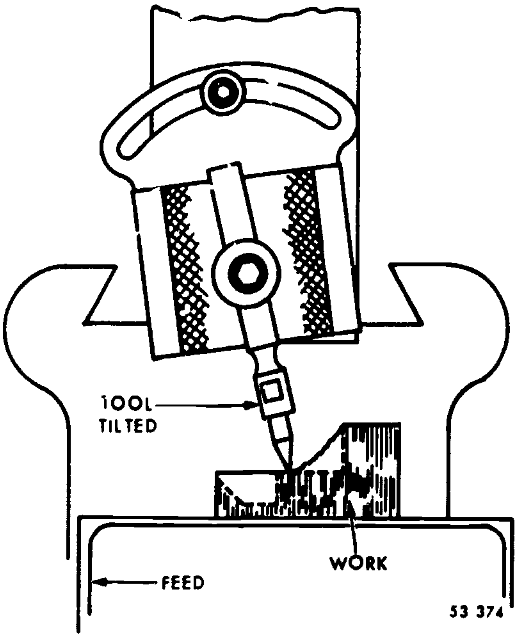


Figure 54. Roughing excess to one shoulder.

40° angle to the shoulder. Pick up the vertical surface of the shoulder and set the crossfeed graduated collar at zero Position the work and the tool with the crossfeed and toolslide so that the tool contacts the horizontal surface of the work. Then set the toolslide graduated collar at zero. Rough out the stepped material next to the shoulder to within 0.005 inch of the vertical layout line. You do this by making a series of vertical cuts. Use the zero setting of the toolslide and crossfeed graduated collars as reference points for each cut. Move the work for the depth of cut (do not exceed 0.050 inch) with the crossfeed. Then move the tool downward with the toolslide for the feed. Feed the tool downward to within 0.005 inch of the horizontal layout line and feed the work away from the tool by engaging the power feed.

5-6. You have now machined both the vertical and horizontal surfaces to within 0.005 inch of the layout line. If the fillet formed by the nose radius on the tool and the finish produced by the roundnose tool are acceptable, go ahead and use a roundnose tool. Machine the shoulder to the finished dimensions. If you require a better finish and a square shoulder, machine the vertical surface with a side-finishing tool and the horizontal surface with a squaring tool. Or machine both the vertical and horizontal surfaces with a squaring tool. You obtain the best finish by using both the side-finishing tool and the squaring tool.

5-7. *Finishing with a side-finishing tool.* To machine the vertical surface of a shoulder with a side-finishing tool, you should swing the top of the clapper box away from the surface that is to be machined. Next, mount the side-finishing tool in a straight toolholder. Aline the cutting edge with a machinist's square. A properly positioned side-finishing tool is shown in figure 55. After setting the machine for proper stroke length, position, and speed, pick up the bottom surface of the shoulder with the point of the tool. Then set the vertical feed dial at zero. When the side intersects a horizontal surface, 0.005 inch may be left on the bottom surface to permit a finishing cut with a squaring tool. Pick up and machine the vertical surface in the manner previously described for machining vertical surfaces. Measure the shoulder width with a micrometer and machine it to the desired size. To machine the horizontal surface, you must use a squaring tool.

5-8. *Finishing with a squaring tool.* Another method of finishing a vertical surface is with the squaring tool. After placing the clapper box in the vertical position, mount the squaring tool in a straight tool holder. Next, position it in the toolpost with the cutting edge parallel to the horizontal surface. Set the cutting edge parallel to the horizontal surface by positioning the tool until a

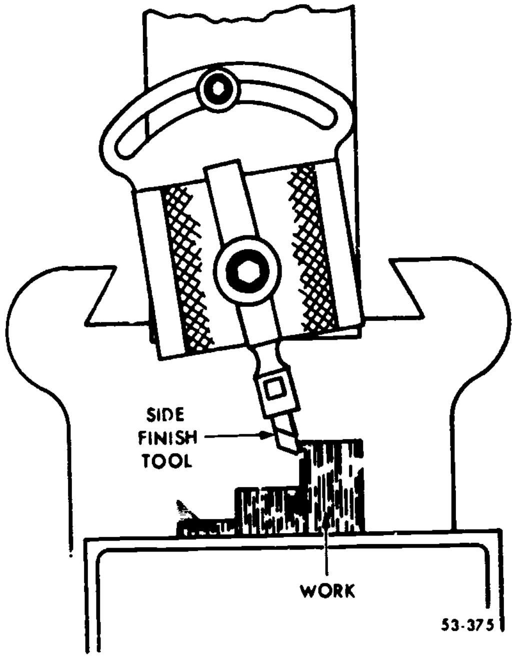


Figure 55. Position of side finishing tool.

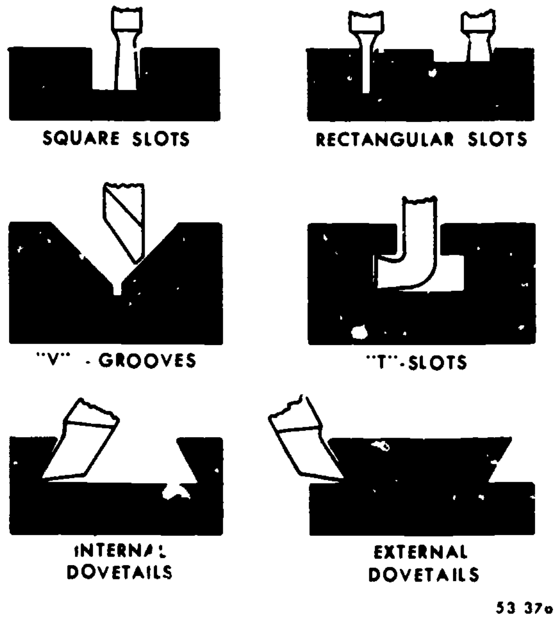


Figure 56. Slots and grooves.

drag is felt on paper placed between the corners of the cutting edge and a parallel. The parallel is placed in the vise or on the table. After picking up the vertical surface of the work, feed the tool vertically. Machine the shoulders to the correct width and pick up the horizontal surface with the tool. Move the tool away from the shoulder and take a shallow cut on the horizontal surface. Now measure the height of the shoulder from the surface that you have just machined. Take additional cuts and feed the work horizontally until you reach the correct depth. NOTE: Feed by power to within 1/8 inch of the vertical surfaces, and feed by hand the remaining distance. Do not exceed a 0.005-inch depth of cut and a 0.010-inch feed.

5-9. **Slots and Grooves.** You can machine small slots and grooves with form tools, as shown in figure 56. Or you can rough out large slots and grooves with a roundnose tool and finish machine them with squaring tools, as shown in figure 57. Use square-nose tools to finish square and rectangular slots. Use left- and right-hand side finishing tools to finish machine V-slots. You can cut off or part work with a shaper by simply machining a narrow slot deep enough to separate the work. Use a shaper parting tool, such as the one shown in figure 34, for parting. Grind shaper parting tools identical to the squaring tool shown in figure 31, except for the width of the cutting edge. The cutting edge on shaper parting tools should be from 1/8 inch to 3/16 inch wide. CAUTION: Do not attempt to part work or machine deep grooves parallel with vise jaws. The pressure of the vise jaws may cause the groove to close slightly and bind on the cutting tool.

5-10. **Keyseats.** A square or rectangular keyseat is an external or internal slot or groove that is cut parallel to the axis of a shaft or hole and which is fitted with a key. This key prevents slippage between a shaft and its mating part. The terms "keyway" and "keyseat" are often used

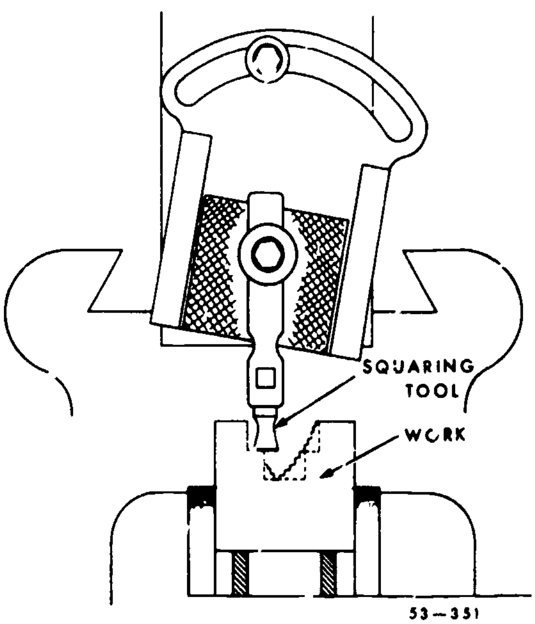


Figure 57. Machining a wide groove.

interchangeably; however, they are not the same. A keyway consists of two grooves or slots, called keyseats, machined in a shaft and in a part going on the shaft which are in alinement with each other. The cutting tool that you use to machine a keyseat is similar to the square-nose tool, shown in figure 31, except for the width of the cutting edge. The cutting edge should be the same width as the key that will be used in the keyway. Before machining a keyseat, you should accurately lay out its location, width, depth, and centerline. Extend the centerline of the keyseat down the end and through the axis of the workpiece. The extended centerline will help you aline the work and cutting tool. A slow operating speed and a depth of cut of less than 0.010 inch will help keep the tool from springing. Both external and internal keyseats can be machined with a shaper.

5-11. *External keyseats.* If the keyseat that you are to machine does not extend the full length of the shaft, drill a hole at the point where the keyseat will terminate, as shown in figure 58A. The diameter of the drill should be equal to the width of the keyseat. The depth of the drilled hole, *excluding* the conical point of the drill, should be equal to the depth of the keyseat. The hole prevents chips from building up in front of the cutting tool and permits machining the keyseat to its full length. If both ends of the keyseat that you are to machine terminate on the shaft, drill holes at both ends of the keyseat, as shown in figure 58B. Drill two adjacent holes on the keyseat where the cut will originate, as shown in figure 59. Remove the metal between the holes by chiseling and filing. The elongated hole will permit the cutting tool to drop into position. Grind away the back portion of the tool, as shown in figure 59, to provide additional clearance between the tool and the sides of the holes. Position and set the length of the stroke carefully. If the length and location of the stroke are incorrect, the tool or the work could be damaged, or *you* could be injured, so *be careful!*

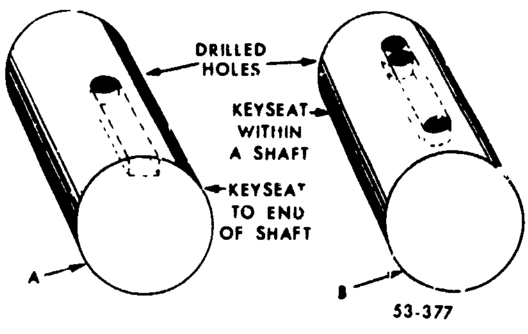


Figure 58. Drilled holes for keyseats.

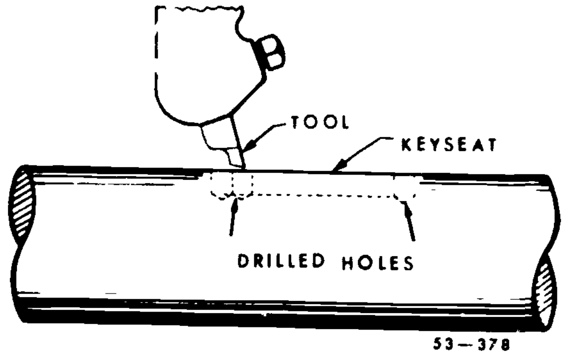


Figure 59. Keyseat slotting tool.

Measure the depth of the keyseat along the side from the bottom to the edge formed by the intersection of the side and the circumference of the shaft. You can find the recommended dimensions of keyseats for shafts of various diameters in machinists' publications such as the *Machinery's Handbook.*

5-12. To machine an external keyseat, first lay out the keyseat, extending the centerline over the end of the shaft. Drill holes at the ends of the keyseat, equal in diameter to the width of the key. Mount the work in the shaper vise. Aline the centerline with a machinist's square, as shown in figure 60. Place the clapper box in the vertical position. Mount the toolholder and cutting tool so that the tool-cutting edge is alined horizontally. Position the keyseat under the cutting tool. Set the machine for proper speed and length of stroke. Carefully position the stroke so that the tool will not overrun the drilled holes. Pick up the top surface of the shaft and machine the keyseat to the required depth.

5-13. The total depth of cut may be determined by the formula:

$$D = \frac{W}{2} + f$$

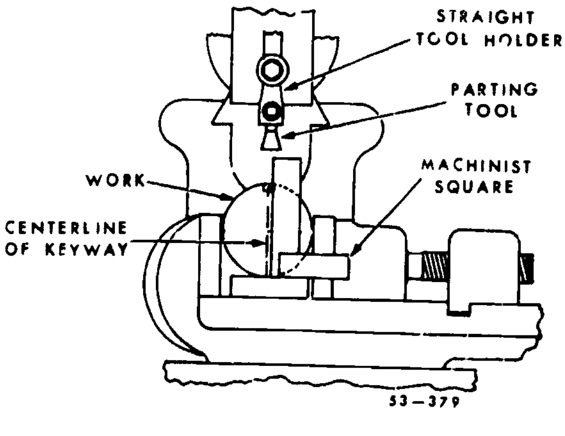


Figure 60. Alining work with a machinist square.

when

D = depth of cut
W = width of key
f = height of arc

(The height of arc may be found in a chart in such machinists' publications as the *Machinery's Handbook.*) Insert a key of the proper size in the keyseat and measure over the key and the diameter of the shaft to insure that the keyseat is the required depth. The micrometer reading over the key and the shaft may be determined by the formula:

$$M = D + \frac{W}{2} - f$$

when

M = micrometer reading
D = diameter of shaft
W = width of key
f = height of arc

Deburr the keyseat and remove the work from the shaper.

5-14. *Internal keyseats.* Internal keyseats are machined with a setup, such as the one shown in figure 61. Note that the clapper box is in the vertical position and is locked to keep it from moving. The tool is held in an extension toolholder and fed upward for the depth of cut.

5-15. Machine an internal keyseat by first laying out the keyseat centerline on one end of the work. Then mount the work in a vise. Aline the centerline with a machinist's square, as shown in figure 61. Mount the cutting tool in an extension toolholder and the toolholder in the clapper box with a lock on it to keep it from swinging. Aline the tool-cutting edge horizontally. Set the machine for the proper speed and length of stroke. Position the stroke and aline the tool with the keyseat. CAUTION: *Check to insure clearance between the cutting tool and the hole and the toolholder and the hole. DO NOT engage the ram movement until you are absolutely sure of the proper clearance.* Pick up the cut on the surface of the bore and set the graduated toolslide collar at zero. Machine the keyseat to the required depth through use of the collar graduations.

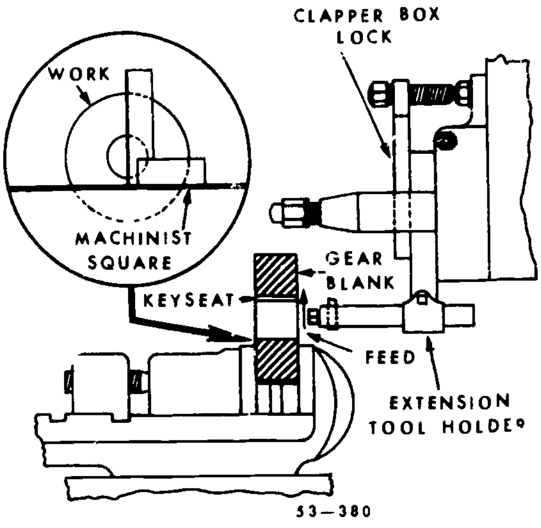


Figure 61. Alining gear and positioning extension toolholder.

5-16. The total depth of cut may be determined by the formula:

$$D = \frac{W}{2}$$

when

D = depth of cut
W = width of key

Before removing the work from the vise or dismantling the tool setup, measure the keyseat for the proper depth with an inside caliper. Measure from the bottom surface of the keyway to the surface of the hole directly opposite. Measure the inside caliper setting with an outside micrometer. The correct micrometer reading for an internal keyseat may be determined by the formula:

$$M = D + \frac{W}{2} + f$$

when

D = diameter of hole
W = width of key
f = height of arc

Deburr the keyseat and remove the work. The keyseat should be wide enough to permit the key to be pressed easily in position, but it should hold the key in position without support.

Contour Machine Work

OF ALL THE machines you will operate, the contour machine will probably be the greatest challenge to your resourcefulness and skill. Unlike the lathe and shaper, the contour machine depends almost entirely upon the skill of the operator for the precision and quality of the work it produces. In this chapter, we will discuss the construction and operation of a contour machine and essential features of saw, file, and polishing bands.

6. Construction and Operation

6-1. In this section we will discuss the main parts of a contour machine and their functions. We will also cover the controls and the operation of a contour machine.

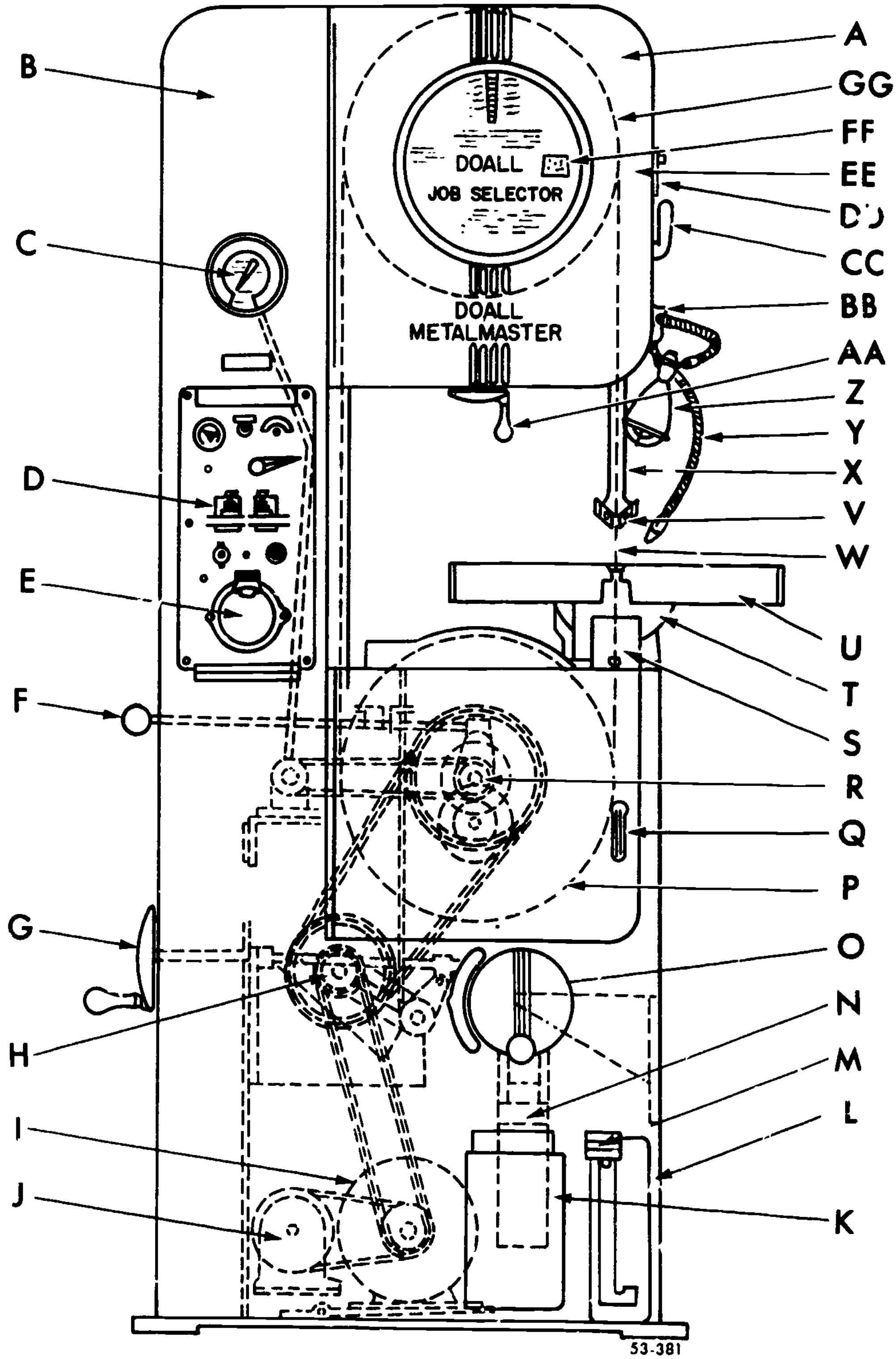
6-2. **Construction.** You must be familiar with the names and the uses of the major parts of a contour machine in order to set up and operate one. Contour machines are made in a variety of sizes and models by several manufacturers. The size of a contour machine is determined by the throat depth, which is the distance from the saw band to the column. All contour machines are similar in construction and operation. In this chapter, the description of the parts, the setups, and the operations pertain to one of the models that is most widely used in the Air Force. You should have no difficulty in applying the information to contour machines made by different manufacturers. Figure 62 will help you locate and identify the major parts.

6-3. *Head.* The head is the large unit at the top of the contour machine that contains the upper guide wheel. The job selector dial mounted on the upper door gives you complete information for sawing, filing, and polishing more than fifty different materials. The drive motor switch is located on the front of the head. It is equipped with voltage overload protection. The tension adjustment handwheel is located directly below the upper guide wheel. A flexible air line is mounted on the front of the head. The air line supplies a constant blast of air at the cutting point to keep the work layout lines free from chips. The adjustable post supports the upper saw guide and can be adjusted close to the work to guide the saw as it passes through the kerf (the slot produced by the saw band).

6-4. If the machine is bolted rigidly to a solid, level floor, it seldom springs out of alinement. If the saw guides are in alinement and the table is square with the post, the machine will cut true. You can check the table for alinement by checking the squareness of the table and the post with a square. If the table is not square with the post, you can tilt the table until it is.

6-5. *Column.* The column is the portion of the machine that supports the head. The speed indicator or tachometer is mounted on the column. It indicates speeds in feet per minute (fpm). It is driven by a separate cable from the transmission. It has a "low range" from 0 fpm to 400 fpm and a "high range" from 0 to 1600 fpm. The resistance type butt welder is used to join new bands and to join bands which have been cut for internal sawing. A grinding wheel for grinding saw bands is mounted directly below the butt welder. The wheel and guard are designed so that bands may be ground on the side as well as on the face of the wheel. CAUTION: *Always wear safety glasses or face shield when you are using the grinding wheel.*

6-6. *Base.* The base, which is the lower portion of the contour machine, contains the lower, or drive, wheel, the motor, and the transmission. The transmission has two speed ranges. The low range gives speeds from 50 fpm to 375 fpm. The high range gives speeds from 200 fpm to 1500 fpm. A shift lever on the back of the column can be placed in the HIGH, LOW, or NEUTRAL position. You use the NEUTRAL position when you are tracking saw and file bands manually on the guide wheel and when the machine is not in use. LOW is recommended for all speeds under 275 fpm. To shift from one range to the other, you must *always* reduce the speed to 50 fpm or less. The shift lever remains locked until the speed has been reduced.



A. Head
B. Column
C. Speed indicator
D. Butt welder
E. Grinder
F. Shift lever
G. Speed change handwheel
H. Variable speed unit
I. Drive motor
J. Air pump
K. Chip box
L. Base
M. Power feed foot pedal
N. Power feed weight
O. Power feed adjustment handwheel
P. Drive wheel
Q. Lower wheel door
R. Gear transmission
S. Chip baffle
T. Graduated plates for table tilt
U. Table
V. Upper saw guides
W. Saw band
X. Post
Y. Air jet
Z. Table lamp
AA. Band tension handwheel
BB. Magnifier outlet
CC. Door latch
DD. Drive motor switch
EE. Upper door
FF. Job selector
GG. Upper saw wheel

Figure 62. Contour machine.

6-7. *Variable speed unit.* The variable speed unit is located within the base of the machine. This unit consists of two V-type pulleys which are mounted on a common bearing tube. The two outside cones of the pulleys are fixed, but you can shift the middle cone by turning the speed change handwheel. When you shift the middle cone, you cause the diameter of one pulley to increase and the diameter of the other pulley to decrease. This slowly changes the ratio between the two pulleys and permits you to gradually increase or decrease the speed of the machine. The variable speed unit is connected to a transmission which has high and low speed ranges. You can obtain a greater number of speeds by shifting the transmission gears and the inner cone of the variable speed unit.

6-8. Before operating the machine, you should thoroughly understand its operating features. Make sure that you are familiar with all the controls and their functions. A thorough knowledge of the machine is required to avoid damaging it or injuring yourself. For personal safety, certain rules must be followed. You should never wear watches or rings. Your sleeves should be rolled up and eye protective devices should be worn. Personal safety hazards and precautions will be explained as various machining operations are covered.

6-9. **Operation.** To operate a contour machine, you should first insure that the transmission shift lever is in NEUTRAL. Only then should you start the drive motor and shift the transmission lever to the desired position. CAUTION: *Never attempt to shift the transmission selection lever except at the very lowest speed in the range being used (50 fpm or less).* Turn the speed-change handwheel clockwise until the desired speed is indicated on the speed indicator. *Never* turn the speed-change handwheel unless the motor is on and the transmission is engaged. When you stop the machine, reduce the speed as low as possible and shift the transmission lever to neutral. *Then* turn the motor off.

7. Saw, File, and Polishing Bands

7-1. The contour machine is primarily a metal-cutting bandsaw. You can also file and polish with it. For sawing, the work is fed against the saw band; for filing and polishing, it is held against the file band or polishing band.

7-2. **Saw Bands.** There is a "best" saw band for every sawing job. You must know the various features of a saw band and the types that are available in order to select the proper saw band for a given job. After selecting the correct saw band, you must weld it properly to obtain good service from it.

7-3. *Saw terms.* You must be thoroughly familiar with saw band terms in order to select the correct saw band. Figure 63 illustrates some of the common terms. We will briefly review some important saw band terms:

a. Type. The type is indicated by the name of the saw band; it refers to the shape and spacing of the saw teeth.

b. Teeth. The teeth are the cutting portions of the saw band.

c. Gullet. The throat or opening between teeth is the gullet. It provides a chip clearance and helps remove the chips from the cut.

d. Width. The width is the measurement of the band from tooth tip to the back edge of the band.

e. Gage. The thickness of the band back is the gage. (For saws up to ½ inch wide, the gage is usually 0.025 inch.)

f. Pitch. The pitch is the number of teeth per inch.

g. Set. The set is the amount of bend given the teeth. The set makes it possible for a saw to cut a kerf or slot wider than the thickness of the band back (gage), thus providing side clearance.

h. Set pattern. This is the pattern of the teeth, depending upon the manner in which the teeth are set. There are three set patterns: (1) raker, (2) wave, and (3) straight, as shown in figure 64.

i. Temper. Temper refers to the hardness of the teeth and the band. Manufacturers provide different types of bands to meet various sawing requirements. Each type of band has a temper peculiar to that type. The job selectors on the earlier contour machines recommended different tempers for various metals and materials. The present practice is to recommend a particular type of saw band for a given metal or material.

j. Kerf. The kerf is the slot produced by the saw band. The kerf width depends upon the rate of feed and the saw pitch, in addition to the amount of set. Since the kerf consists of a series of grooves, the distance between the grooves

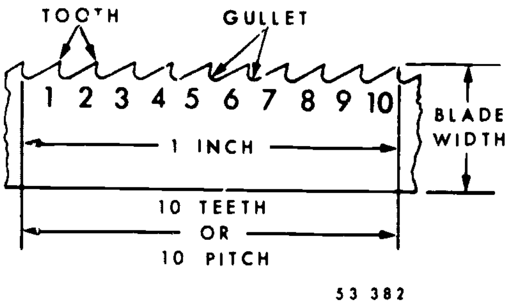


Figure 63. Saw band terms.

varies with the pitch and rate of feed. As the distance between the grooves increases, the peaks become higher. The result is a narrower slot or kerf. The corners of the saw teeth cut a groove equal to the measurement over the set, but the kerf width is the distance between the peaks, as shown in figure 65.

7-4. *Types.* Several types of saw bands with different tempers and differently shaped teeth are made by various manufacturers. Figure 66 will help you recognize some of the common shapes of saw band teeth. Give special attention to the shape and spacing of the saw teeth, since they will help you to identify the various types

7-5. *Selection.* The following factors should be taken into consideration when you determine the saw band to use:

a. At least two teeth should be in contact with the work at all times.

b. Thicker material requires fewer teeth per inch. This varies, however, with the type of material and saw speed.

c. Always use the widest and thickest saw possible. However, take into consideration the curvature of the cut since, as shown in figure 67, you cannot saw sharp curves with wide saw bands.

d. Use the raker set pattern for general sawing of most shapes.

e. Use the wave set pattern where thin work sections are encountered during the cut, such as tubing, angles, channels, etc. Use it also for tough materials, such as aluminum, bronze, and work-hardened stainless steel. The wave set is designed so that the band "dances" through the kerf to shake off the chip and minimize chip welding

f. The finish depends largely on the saw pitch. The faster the saw speed and the finer the saw pitch, the finer the finish. Lubricating helps to improve the finish. A fine saw pitch, high velocity, and light feed produce the finest finish.

g. Too fine a saw pitch for the work thickness causes a loading action in the gullets of the saw teeth. A lubricant will help correct this, but it is best to use the coarsest pitch that will give the finish desired.

h. For materials that are tough and stringy, such as brass, copper, and wrought iron, it is best to use coarse-tooth bands. Fine-tooth bands

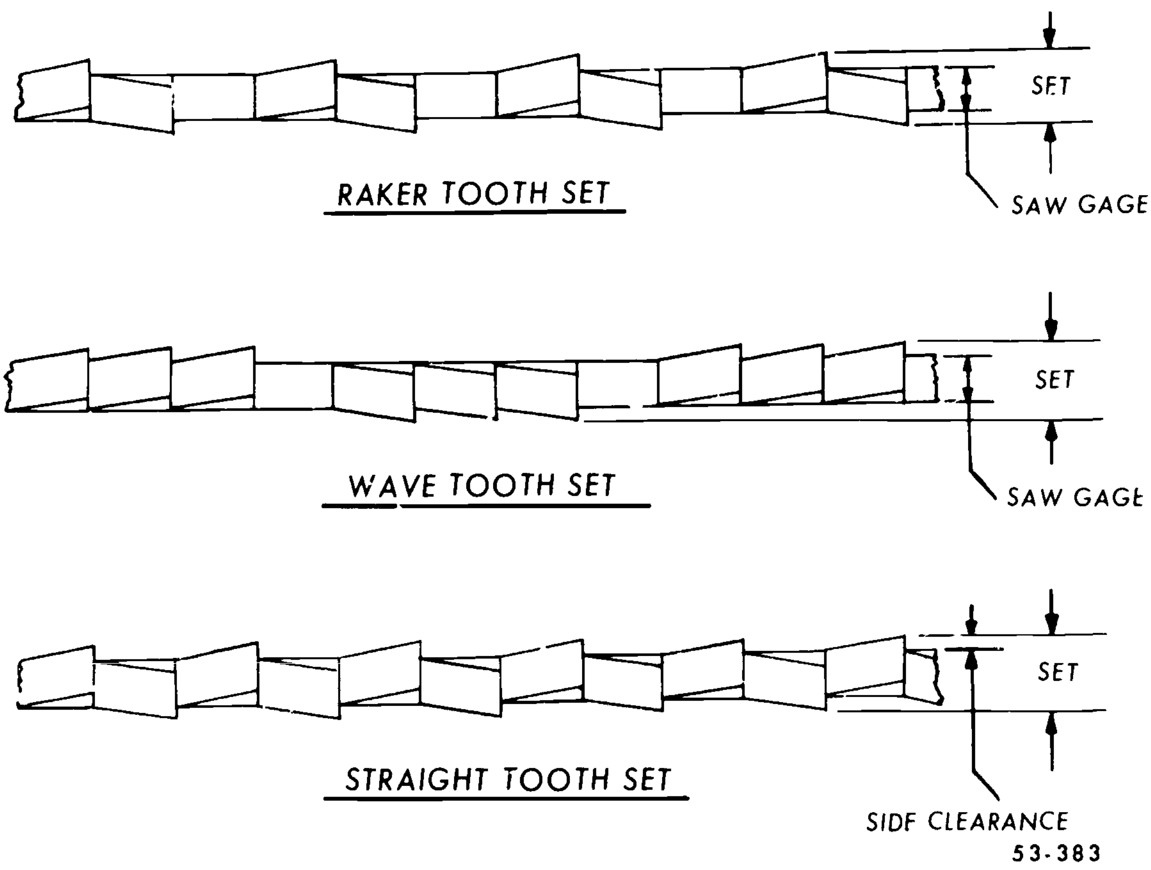


Figure 64. Set patterns

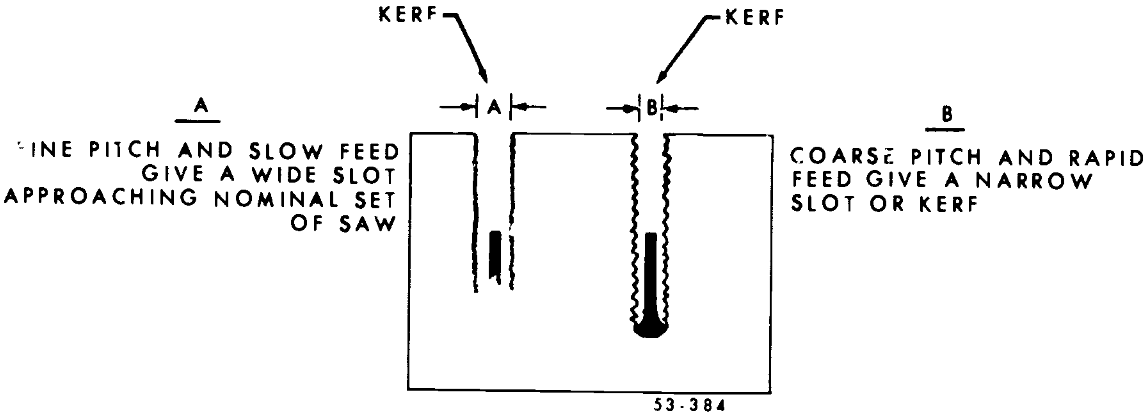


Figure 65 Saw kerf.

are better for harder, less stringy materials and steel.

i. The set prevents the band from binding. It provides a clearance which makes radius cutting possible. It is the difference between the set of the saw and the gage that enables the band to turn in a cut, as shown in figure 67. The amount of set determines the width of the kerf and the amount of material removed by the band. The narrower the kerf, the less feed pressure and the less power will be required for sawing.

j. When you cut irregular shapes you must consider the set. The wider the set, the wider the kerf and the easier it is to saw irregular shapes, since the band has more clearance in which to be turned.

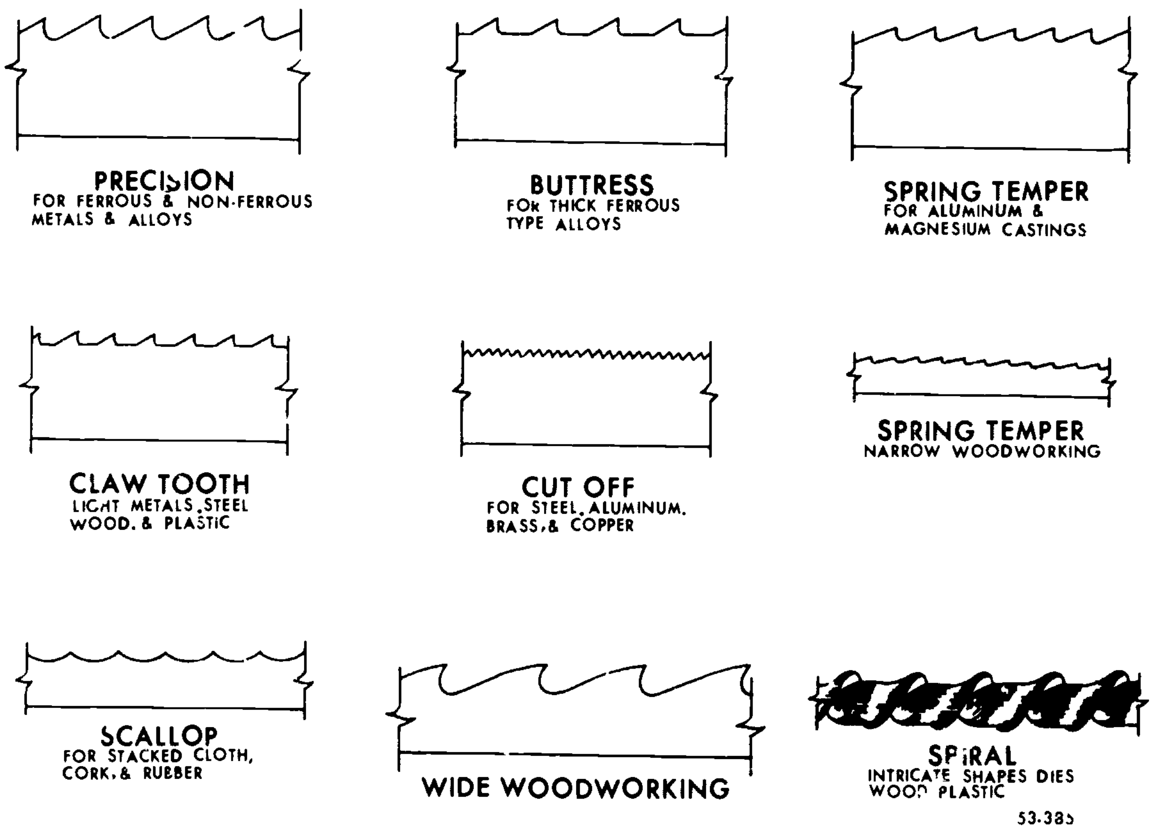


Figure 66. Shapes of saw band teeth.

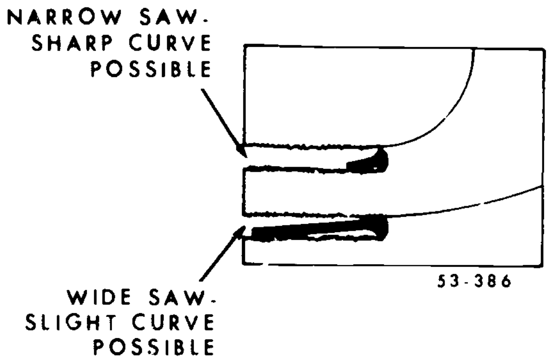


Figure 67 Effect of set when sawing radii

k For very gummy materials, a coarse-tooth band should be used.

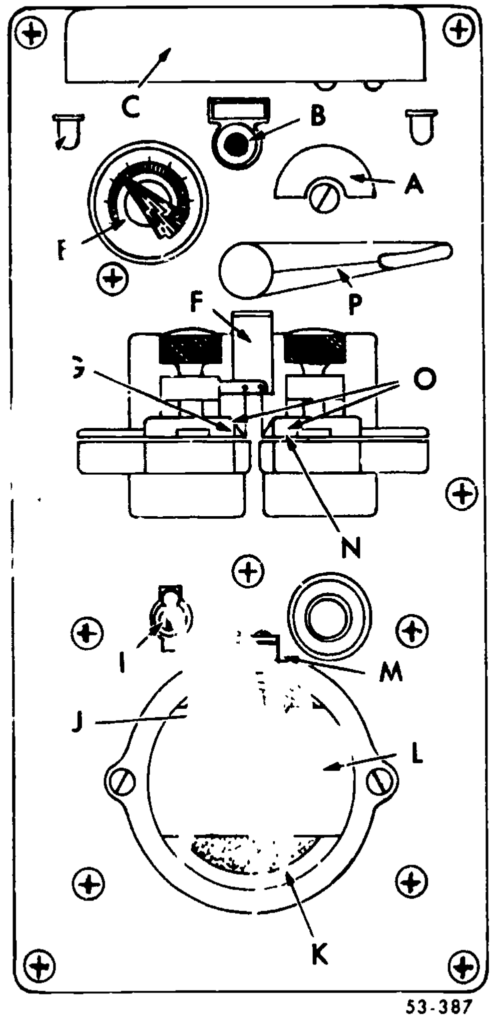
l. The recommended pitch for various types of material is given below·

Pitch	Material
8	Large solid sections over 2 inches thick
10	Soft metals.
12	Medium solid sections, ½ to 2 inches thick, and hard steels.
14	Heavy gage structurals and general-purpose cutting.
18	Light sections, $^1/_{16}$ to ½ inch thick, light structurals, and medium gage sheets and tubing.
22	Very light sections.
24	Very light structurals, and thin gage sheets and tubing.
32	Very thin sheets and tubing.

Chart 2 describes some of the various types of saw bands and their recommended uses.

7-6. *Welding*. Saw bands are usually received in the shop in 100-foot long strips. They must be cut to the required length, and the ends must be welded together to form an endless loop. The length required for a particular saw may be found in the instruction manual or in the technical order for the machine. It is sometimes given on a data plate that is mounted on the machine column.

7-7. The butt welder, which is built into the column of the machine, is used to weld new bands and to rejoin bands that have been broken. The butt welder panel assembly, shown in figure 68, shows the general arrangement of the panel as viewed by the operator. A 15-watt lamp is housed at the top of the panel. On either side of the panel, you will note two oil fillers. The spring cap oil fillers are connected by means of copper tubing to the sleeve bearings of the grinder motor. Below the oilers is the tension control dial or weld selector, which regulates the tension of the movable jaw for each width of saw you are welding. This dial is mechanical and enables you to control the force with which the movable jaw moves toward the stationary jaw Wider saws require greater force than smaller ones. Too much force on small saw bands will cause the ends to overlap. To the right of the tension control is the line voltage regulator switch.



A. Line voltage regulator
B. Etching pencil ground bushing
C Lamp
D. Oil cups (for grinder)
E. Tension control
F. Flash guard
G. Stationary jaw
H. Mounting screws
I Lamp and grinder switch
J. Step for squaring saw bands
K. Grinding wheel
L. Grinding wheel guard
M. Weld thickness gage (2 gaps)
N. Movable jaw
O. Welder jaws
P. Welding operating lever

Figure 68. Butt welder.

CHART 2

SAW BANDS DESCRIPTIONS AND USES

TYPE OF SAW (NAME)	DESIGNED FOR CUTTING	CHARACTERISTICS	TOOTH PATTERN	WIDTHS	PITCH	THICK-NESS	SET	TOOTH SHAPE	HARDNESS ROCKWELL "C"
PRECISION SAW BANDS -- The most used saw in machine shops	Ferrous & non-ferrous metals, some plastics and wood	Made for fine precision cutting	Raker & Wave	10 Standard 1/16" to 1"	6 to 32	.025" .032" .035"	.038" to .063"	Raker Angle	60 to 65
BUTTRESS SAW BANDS	Heavy thick ferrous metals, wood, plastics and non-ferrous metals	For fast cutting. Teeth retain cutting edge longer, large chip clearance, small sharp teeth. Greater tensile strength and longer flex life	Raker only	10 Standard	2 to 6	.025" to .035"	.042" to .062"	No Rake Angle	60 to 65
CUT-OFF SAW BANDS	Steel, brass, aluminum, copper, and manganese	For rapid & accurate production cutting. Use of a lubricant gives a better finish	Raker & Wave. Wave in welded lengths only	4 Standard	6 to 24	.025" to .035"	.042" to .063"	Rake Angle	
FRICTION SAW BANDS	Ferrous types of steels; also will cut hardened tool steel	Must be used on high-speed machines at about 15000 fpm		1/2" to 1"	10 to 14	.032" to .035"	.054" to .058"		
SPRING TEMPER SAW BANDS	Light metals & non-ferrous metals in foundry work	Long flex life. Can be resharpened and reset.	Straight	3/16" to 2"	2 to 6	.020" to .049"		10° Positive Rake Angle	38 to 44
SCALLOP EDGE SAW BANDS	Soft & fibrous materials. Felt, paper, cloth, cork and rubber	Separates material. Does not produce a chip. Has a slicing action. Furnished in welded bands.		1/4", 1/2" and 3/4"		.015" .020" .028" .037"			52
KNIFE EDGE SAW BANDS	Soft fibrous materials	Double bevel and single bevel. Furnished in welded bands	No teeth	1/4", 1/2" 3/4" & 1"					
CLAW TOOTH SAW BANDS	Light metals and alloy metals wood, plastic and steel	Produces chips more freely, teeth are super hard		7 Standard 1/4" to 1-1/2"	2 to 6	.025" to .035"	.042" to .065"	Positive Rake Angle	60 to 65

53 417

The line voltage regulator compensates for voltage variation in the electrical power supply. The voltage regulator permits the proper amount of heat to be generated at the weld by controlling the current flow between the saw clamps. You use a screwdriver to change the voltage regulator setting. Move the index point on the regulator switch toward the "MORE" position to increase the heat. Move it toward the "LESS" position to decrease the heat. The line voltage regulator seldom needs to be reset if the current supply remains fairly constant.

7-8. Directly below the line voltage regulator switch is the welder operating lever. The operating lever turns the current on and moves the movable clamping jaw toward the stationary clamping jaw. The movable jaw travels approximately 0.040 inch, forcing the molten ends of the saw band together and welding them into a solid unit. The clamping jaws hold the ends of the saw band during the welding operation and the welded portion of the saw band during the annealing operation.

7-9. The annealing switch button, which is usually red, is located below the clamping jaws. You hold this spring-loaded button in the depressed position to heat the weld to the annealing temperature.

7-10. You use the grinding wheel on the welder to square the ends of the saw band before welding them together and to remove the excess metal from the weld after the welding so that its thickness is no greater than that of the rest of the band. The weld must be ground flush with the saw band. A gage located above the grinding wheel is used to check the saw band thickness. The portion of the saw band behind the sawteeth will pass freely through the gage when the weld is correctly ground. CAUTION: *Always wear eye protection in using this grinder.*

7-11. To weld a saw band, first cut off the band to the required length and follow this procedure:

a. Always cut the band from the back toward the teeth. Grind the ends of the band square against the side of the grinding wheel. Then, insert the ends of the band into the jaws of the butt welder with the teeth pointed toward you and clamp them in this position by turning the thumb screws. Set the tension control switch for the width of band and the line voltage regulator for the required welding heat. After placing the flashguard down, depress the operating lever to complete the weld and hold it down until the weld has cooled. BEFORE releasing the operating lever, loosen the stationary jaw thumbscrew and then release the band from the movable jaws. Move the band forward (toward the operator) to the wide gap annealing position. Reclamp the band just behind the saw teeth, with the newly welded joint centered between the jaws. Now press the annealing switch button until the welded area becomes a dull cherry red.

b. Turn off the welding panel light so that the correct annealing heat can be observed. Cool the annealed portion gradually by pressing the annealing button several times during the cooling period. After it has cooled enough to be safely handled, remove the band from the jaws and grind the excess weld off both sides of the band. Grind until the welded joint is the same thickness as the band. Use the gage directly above the grinding wheel to check for correct thickness.

c. Do not grind the teeth.

7-12. *Storage.* Generally, storing saw bands and bulk saw material is not a problem, since each contour machine usually has a storage cabinet. These cabinets contain bins to store saw bands, space for accessories, and a place to hang file and polishing bands. Bulk saw bands are usually shipped to Air Force shops in 100-foot lengths in special containers, which are stored in the cabinet. A container has a slot from which you may pull out and cut off any desired length of band. After you have welded the band into a loop, you can coil it into smaller loops for storage in the cabinet. Tie the loops together with string or wire to prevent them from uncoiling. In coiling a band for storage, be careful not to kink or bend it. Kinked and bent bands do not produce good results. CAUTION: *Be careful not to cut or scratch your hands when you fold or unfold saw bands.*

7-13. When you finish sawing with a particular saw band, you should remove, coil, and store it in the storage cabinet. Saw bands may be coiled by using several methods, but we will discuss only one. Hold the band in one hand with the other end of the loop just touching the floor. Then place your foot on the portion of the band in contact with the floor just hard enough to prevent the band from moving. At the same time, twist the band by rotating your wrist 1½ to 2 times while lowering your arm. The band will automatically coil itself, generally into 3 loops.

7-14. *Cutting action.* The saw band passes over the upper saw wheel and below the drive wheel, as shown in figure 62. Guides are provided to guide and support the saw band above and below the section where the cutting action occurs. The saw guide block assembly consists of a cast iron guide block and hardened steel inserts, as shown in figure 69.

7-15. *Guide blocks.* The upper guide block has two screw holes and is mounted on the saw post. The lower guide block has one screw hole and one alining pin and is mounted on a keeper

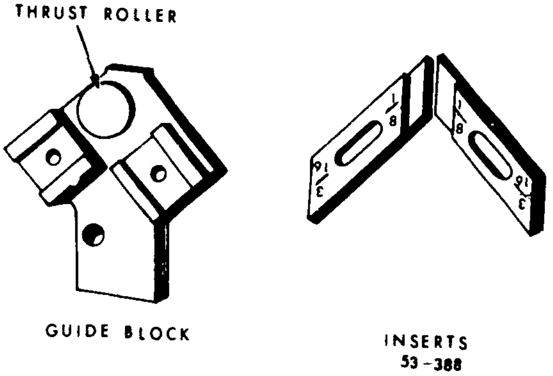


Figure 69 Guide block and inserts

block below the saw table. The saw guide inserts are mounted in grooves machined in the guide block. Two thrust rollers, one on the upper guide block and another on the lower guide block, prevent the saw band from springing under the pressure of the cut.

7-16. *Guide inserts.* The saw guide inserts are made of hardened tool steel and are available for every size of saw band. The size of the inserts must correspond to the size of the saw band. When the ends of the inserts become worn, they can be reground to an angle of 45°.

7-17. To mount saw guide inserts in the guide block, first mount the left-hand insert in the block by using an insert gage, as shown in figure 70, to position the insert. Then, use an insert gage as a thickness gage and position the right-hand insert, as shown in figure 71. Finally, mount the upper guide block on the saw post and the lower guide block on the keeper block, as shown in figure 72.

7-18. *Positioning.* When you place the saw band over the upper sheel and below the drive wheel, it should aline directly in the slots between the saw guide inserts. Tilting the upper wheel (fig. 73A) causes the band to "track" or move into the desired location. Tilt the upper wheel by means of the tilt screw, as shown in figure 73G, until the back of the saw band just touches the thrust roller. Then, lock the wheel in place with the tilt locknut (fig. 73F), which is the large diameter nut that encircles the smaller sized tilt screw.

7-19. *Tension.* You adjust the tension of the saw band by raising or lowering the upper wheel by means of the handwheel, shown in figure 73J. The saw band must be kept tight to prevent it from twisting and to keep it sawing straight. A new saw band will stretch slightly after use and must be readjusted for the proper tension. There is no hard-and-fast rule governing the tension adjustment. You must rely on your experience to guide you. Generally speaking, it is better to have the band too tight than too loose. NOTE: Overtightening the saw band can cause it to break or cause excessive wear on the wheel tires. Extra care should be used when you adjust the tension of bands less than 3/16 inch is width.

7-20. *Mounting.* The first step in mounting a saw band is installing the proper size inserts on the upper and lower guide blocks, as shown in figures 70 and 71. Open the upper and lower wheel access doors, remove the filler bar from the table slot, and mount the upper and lower guide blocks, as shown in figure 72. Place the right-hand portion of the saw band in the table slot and position the saw band over the upper wheel and below the drive wheel. Set the tension by using the handwheel, shown in figure 73, just

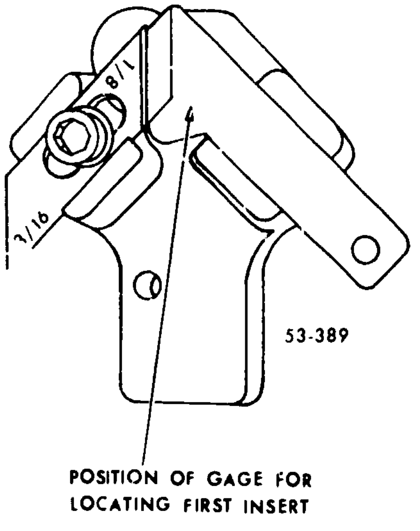


Figure 70 Installing left insert

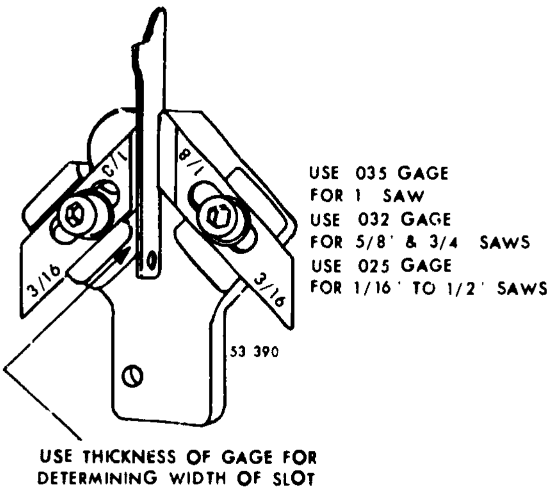


Figure 71. Installing right insert.

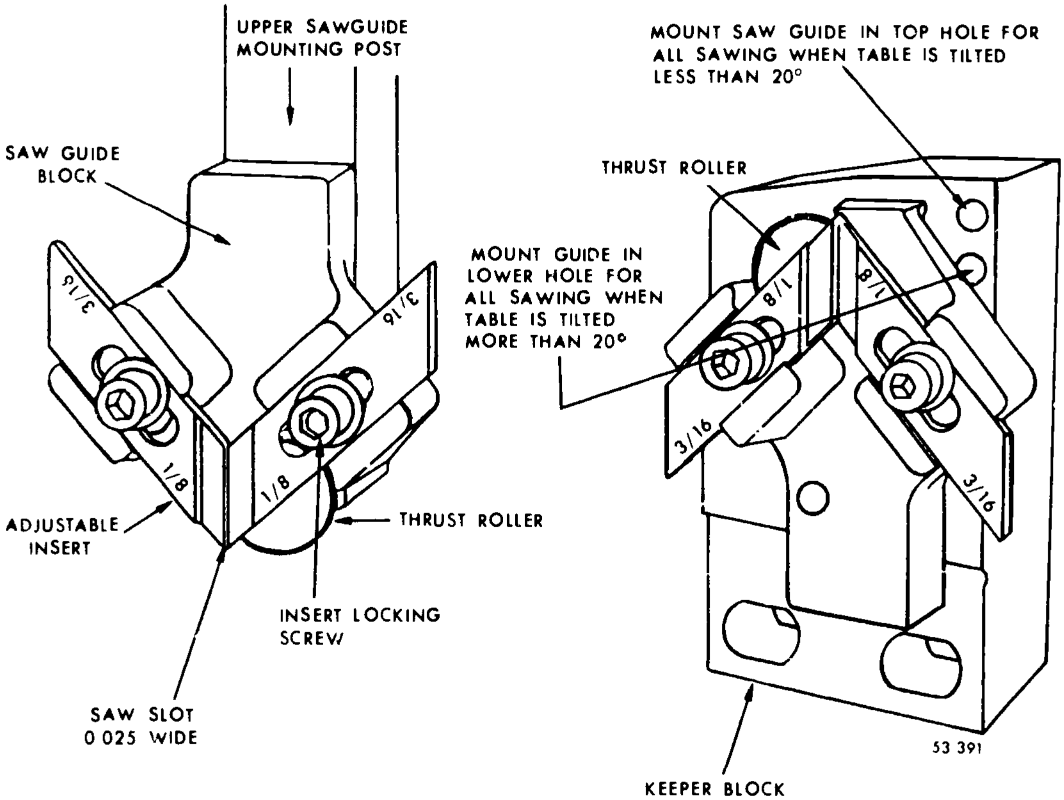
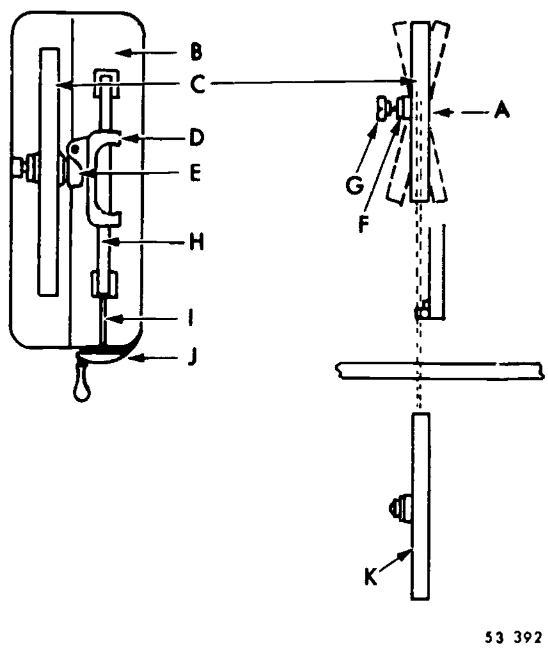


Figure 72. Mounting saw guides

tight enough to keep the band on the wheels. Then, check to insure that the saw band is positioned between both the upper and lower sets of inserts. After loosening the tilt locknut, rotate the upper wheel by hand and use the tilt screw to tilt the upper wheel until the back edge of the saw band contacts the thrust roller on the guide block. The moving band should cause the roller to turn, but light finger pressure should stop the roller from turning. *NOTE: The transmission should be in NEUTRAL so that the wheels can be turned by hand.* Once the band is tracking (moving between the inserts and rotating the thrust roller) correctly, lock the tilting wheel by means of the tilt locknut. Adjust the tension of the saw band, replace the filler bar in the table, and close the upper and lower doors. The machine is now ready for sawing.

7-21. **File Bands.** Three widths of file bands are available: ¼ inch, ⅜ inch, and ½ inch. Use the widest file band that will fit the contour of the work. A wide file band has a longer life than a narrow one, and the time required for filing is shorter. File bands are available in flat, oval, and half-round shapes. The shape of the work determines the shape of the file band to be used. File bands are available in a variety of cuts and pitches. Chart 3 shows some of the various types of file bands and the proper file bands to use for various metals.

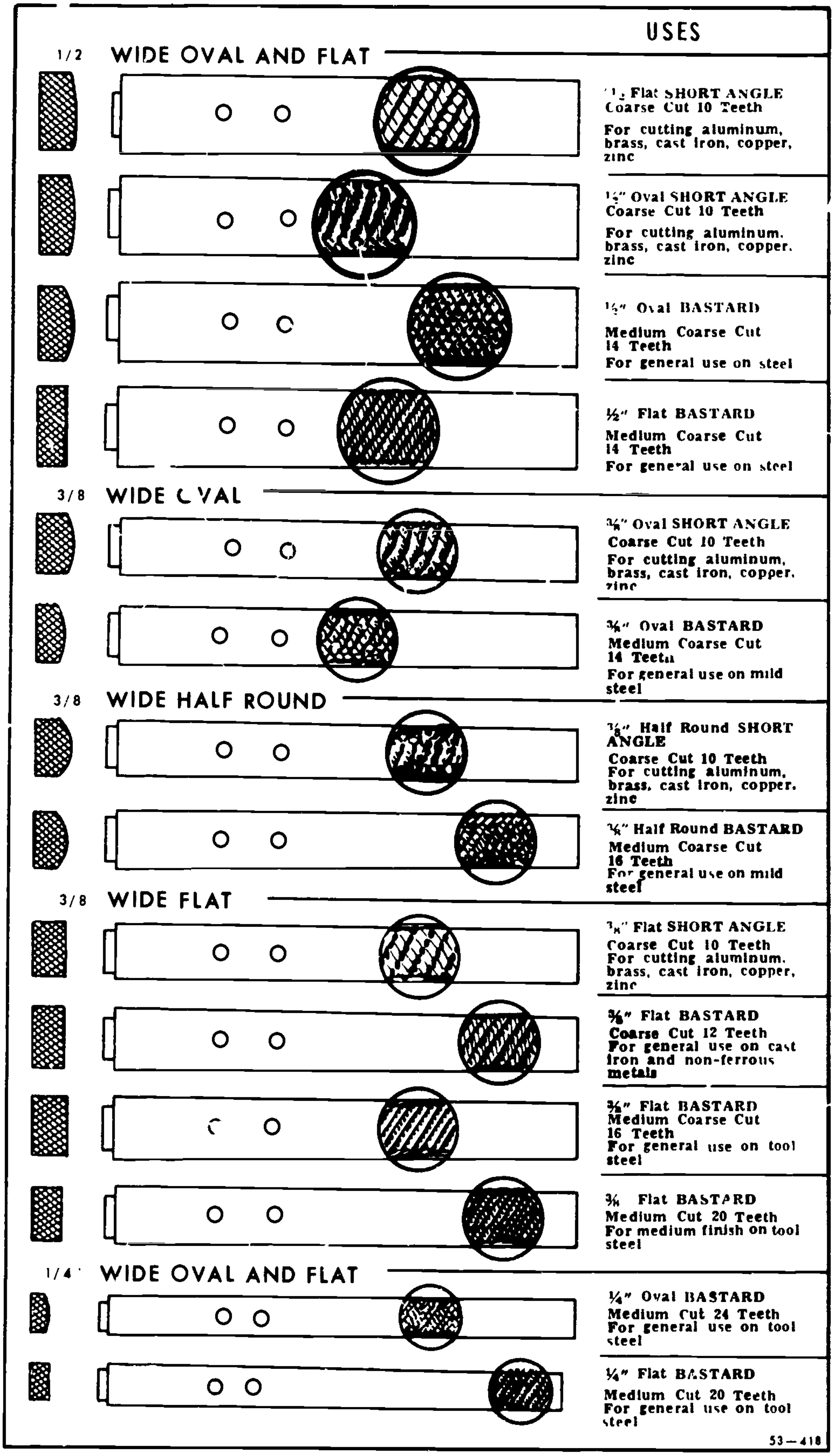


A. Upper wheel and adjustment assembly
B. Upper door
C. Upper (guide) wheel
D. Slide
E. Hinge
F. Tilt locknut
G. Tilt screw
H. Slide rods
I. Slide screw
J. Handwheel
K. Lower (drive) wheel

Figure 73. Tilting wheel.

CHART 3

TYPES OF FILE BANDS AND THEIR USES



297

7-22. A file band consists of a number of interlocking 3-inch file segments that are fastened to a spring steel band. The segments are fastened to the band at one end. This allows the band to bend as it passes around the upper and lower saw wheels and thus permits the segments to be locked solidly by the pressure of the work, as shown in figure 74. To connect the ends of the file band, place the keyhole-shaped hole in the metal strip over the head of the shoulder rivet in the gate segment, as shown in figure 74. Pull the narrow portion of the hole below the rivet head as you position the second hole over the dowel rivet. The gate segment, as shown in figure 75, is painted yellow to help you locate it when you are disconnecting it. CAUTION: BE SURE THAT THE FILE BAND HASN'T BEEN INSTALLED UPSIDE DOWN.

7-23. The segments are fastened to the spring steel back with two rivets at the leading end of the segment, as shown in figures 74 and 76. A chip removal bridge is located between each file segment and the spring steel band. When the band bends, the bridge lifts the loose or trailing end of the segment ahead of it and helps to remove chips.

7-24. You can easily replace damaged or broken file segments. The rivets which fasten the new segment to the spring steel band should be the same diameter as the holes in the band. Rivets that are too small will allow the segment to work loose during filing, and they may possibly damage the file band. The diameters of the standard sized rivets are 0.060 inch, 0.070 inch, and 0.075 inch. Figure 76 shows the location and types of rivets which are used to fasten a gate and a standard segment to the spring steel band. The method which is used to repair a broken file band is shown in figure 77.

7-25. You should handle file bands with care to prevent kinking the spring steel band. A kink prevents the segments at the location of the kink from interlocking, causing rough, uneven filing. When you carry a file band, hold it with both hands and form a loop with no less than a 16-inch radius at the top. Support the file band with your left arm, as shown in figure 78, when you join the file band in the machine. You can sometimes straighten out a kink by removing the kinked file segment or segments and by bending the band back into shape by hand or tapping it lightly with a hammer.

7-26. The file band is supported at the point of contact with the work by a file guide, which should be the same width as the file band. You mount the file guide on the guide post and support it at the lower end by the file guide support. The file guide support is mounted on the keeper block, as shown in figure 79. The file guide is designed to fit all three sizes of file guides (1/4 inch, 3/8 inch, and 1/2 inch). Be sure to use the correct slot in the file guide support, as the file guide will not aline properly in the wrong slot.

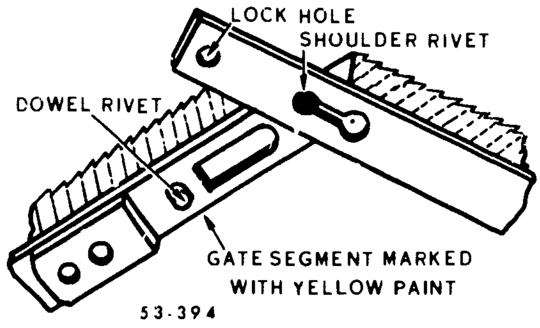


Figure 75. File band gate segment.

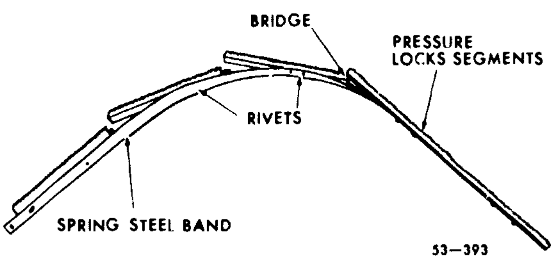
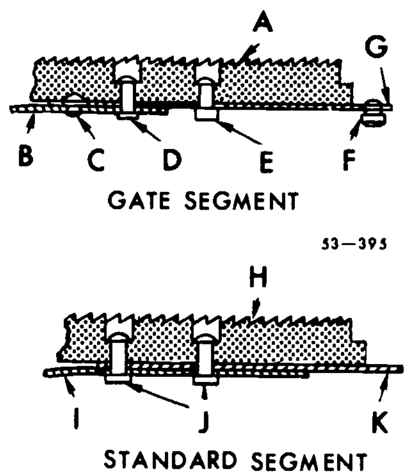


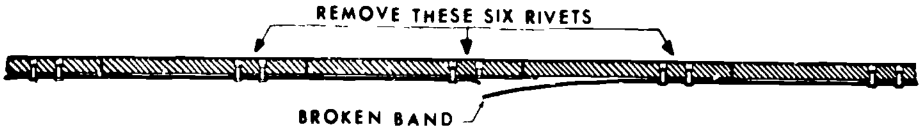
Figure 74. File band segments.



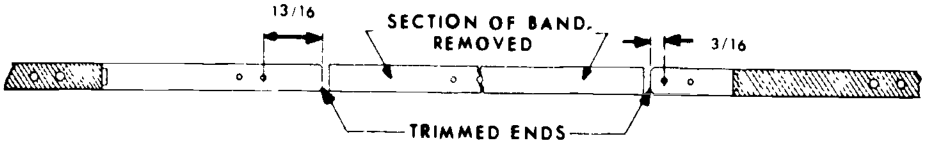
A. Yellow gate end segment
B. Steel band
C. Standard rivet (clipped)
D. Standard rivet
E. Dowel rivet
F. Shoulder rivet
G. Gate clip
H. Regular segment
I. Steel band
J. Standard rivets
K. Spacer

Figure 76. File band construction

STEP I REMOVE RIVETS AND SEGMENTS ADJOINING BREAK



STEP II TRIM BROKEN STEEL BAND AS SHOWN



STEP III FIT SPLICE PART INTO PLACE AND RIVET SEGMENTS TO SPLICED SECTION

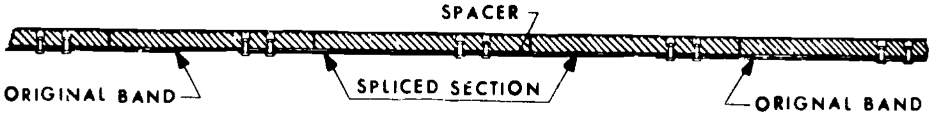


Figure 77 File band repair

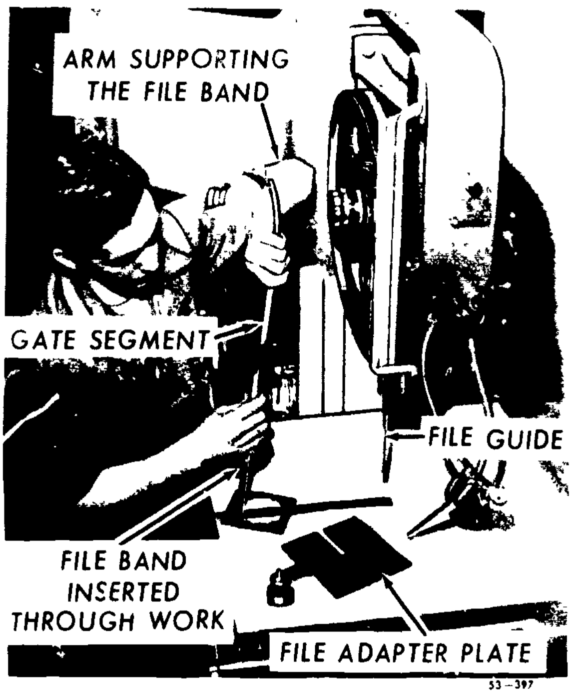


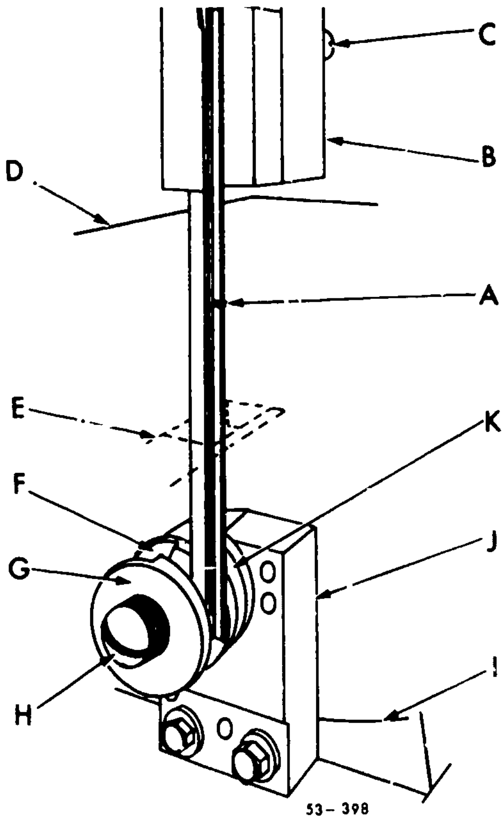
Figure 78. Connecting a file band

7-27. **Polishing Bands.** The standard polishing bands are 1 inch wide and are available in the following grit sizes:

Application	*Grit*
Grinding	50
Polishing, coarse	80
Polishing, fine	150

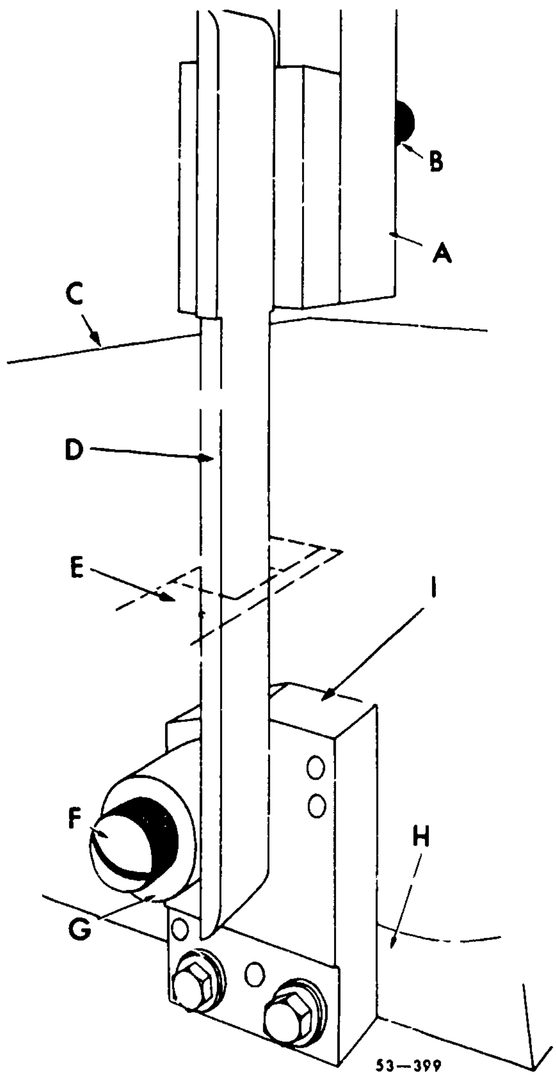
Polishing bands are in the form of a loop when they are received in the shop. You mount and track a polishing band in a manner similar to that used in mounting and tracking a saw or file band. Mount the polishing band guide on the guidepost and fasten the polishing guide support to the keeper block, as shown in figure 80. To prevent tearing, the polishing band must be mounted on the two wheels in a definite manner. There is an arrow mark on the back side of each band. When you place the band over the wheels properly, the arrow points in the direction in which the band travels. Proper mounting prevents the ends of the band, which are overlapped and glued, from separating. Too much work pressure will cause the edges of the band to fray. Remove any frayed edges immediately to prevent the band from tearing and possibly injuring you.

7-28. The recommended speeds for polishing are somewhat higher than the speed range of most contour machines. The polishing speed de-



A. File guide
B. Saw post
C. File guide screw
D. Table
E. Filler plate
F. Channel for ½" guide
G. File guide support
H. Support screw
I. Trunnion cradle
J. Trunnion keeper block
K. Channel for ¼" guide

Figure 79 File guide assembly



A. Saw post
B. Polishing guide screw
C. Table
D. Polishing band guide
E. Filler plate
F. Support screw
G. Polishing guide support
H. Trunnion cradle
I. Trunnion keeper block

Figure 80 Polishing band guide

pends upon the grit size of the polishing band. The coarser the grit, the slower the surface foot speed of the band should be. The speed for polishing various metals can be found in operating manuals or in machinists' publications, such as the *Machinery's Handbook*.

7-29. Tension for polishing bands should be just "snug" enough to eliminate slippage. Too much tension will stretch and possibly tear a band. The guide support, which acts as a backing for the polishing band when work is being polished, should be supported at the lower end. The lower end of the band polishing guide does not, however, get any support from the guide support until the post is lowered to a position of 4 inches or less, as marked on the guidepost.

Sawing, Filing, and Polishing Operations

YOU MAY THINK that sawing, filing, and polishing on a contour machine are simple tasks. Sometimes they are. There are times, however, when these operations require exacting machine and work setups. The accuracy of the work is strictly dependent on the knowledge and skill of the operator.

The first section of this chapter covers straight and contour sawing, filing, and polishing. In the final section, we will discuss the attachments that can enable you to do more accurate work. Some of these attachments increase the capacity of the contour machine and make it possible to perform certain operations more easily.

8. Straight and Contour Sawing, Filing, and Polishing

8-1. Most straight and contour sawing, filing, and polishing operations are performed to layout lines. The accuracy of the layout governs the accuracy of the completed work. You saw work to produce the general shape and size of the part. Filing and polishing are primarily finishing operations.

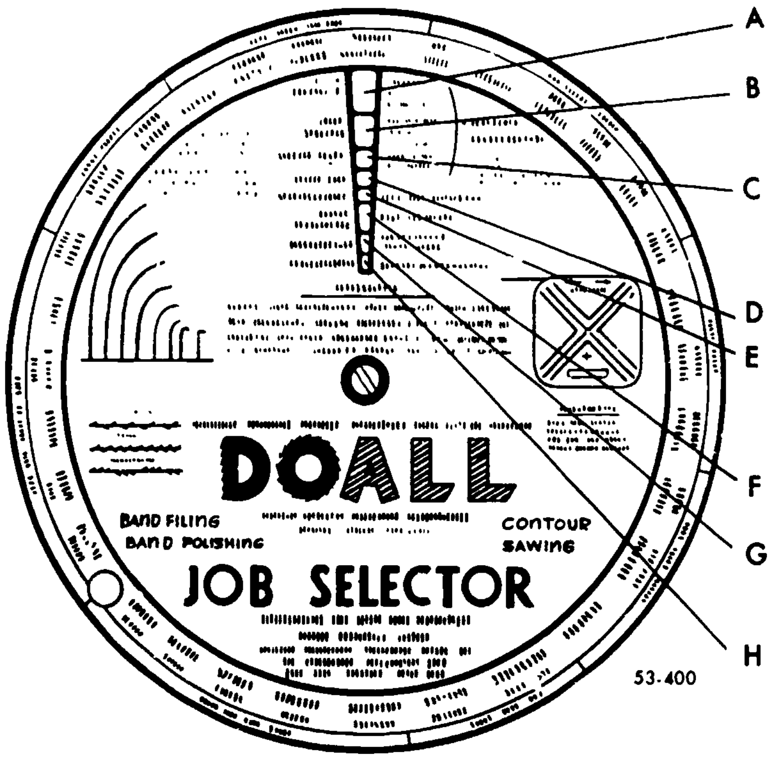
8-2. **Straight Sawing.** In straight sawing, the saw band presents a continuous flow of teeth to the work. You select the operating speed with the job selector dial, which is mounted on the upper door, as shown in figure 81. This dial gives the correct saw and file band to be used for 55 basic materials, along with their correct operating speeds.

8-3. The speed of a saw is expressed in feet per minute, or the number of feet of saw band that passes the work in 1 minute. The cutting speed that you select depends upon the type and thickness of the material that you are sawing. The job selector dial consists of a large diameter lower disc and a slightly smaller diameter upper disc. Various materials and thicknesses of materials are printed on the outer face of the large disc along the rim. Rotate the disc by hand to aline the desired type of material above the vertical line of holes in the upper disc, as shown in figure 81. The correct saw speed, type of saw, file speed, etc., are shown in the holes in the upper disc.

8-4. For straight sawing you should use the widest available saw band of the proper pitch. This will help keep the band from twisting and make the cut straighter. Thinner bands are required for contour sawing. This prevents the band from rubbing on the sides of the cut. When a sawed finish is desired, the kerf should just split the layout line on the waste side of the metal, as shown in figure 82A. If a filed finish is desired, approximately 1/64 inch should be allowed on the waste side of the layout line, as shown in figure 82B. If a corner requires a small radius, the proper size of drill to give this radius should be used, as shown in figure 83. In order to cut a sharp or square corner, the corner may be cut to a drilled hole first and then the radius of the corner notched out with the saw, as in figure 84A. Square turns may also be made without drilling. This is done by notching a space with the saw, as in figure 84B. The saw can then be turned on this notch and a cut can be made in another direction.

8-5. In straight sawing, you should maintain a constant pressure against the work. Permitting the saw to ride without cutting dulls the saw teeth and can work harden the metal. Use a wooden push stick to apply pressure to the work to avoid injury to your hands. Injury to the eyes from flying chips can be eliminated by always wearing eye protection. Saw band breakage during sawing is not common, but if it should occur, stand clear of the machine, press the stop swtich, and let the wheels coast to a stop.

8-6. To set up the contour machine for straight sawing, remove the filler plate from the table slots. Mount the saw guides and the correct inserts. Lower the upper wheel so that the saw band fits over both wheels. Place the saw band on the center of the wheels and adjust the tension to take up the slack. Track the saw for correct alignment. NOTE: *Place the transmission in neutral and turn the upper wheel by hand.* Readjust the saw band for the correct tension. Replace the filler plate. Adjust the upper saw guides to within 1/4 inch of the work surface. Consult the job selector for the correct saw velocity. Press the drive motor starting switch and engage the transmission Engage the shift lever



A. Saw velocity in feet per minute
B. Saw pitch or number of teeth per inch
C. Power feed
D. Saw set
E. Saw temper
F. File velocity in feet per minute
G. File cut
H. File type

Figure 81 Job selector dial

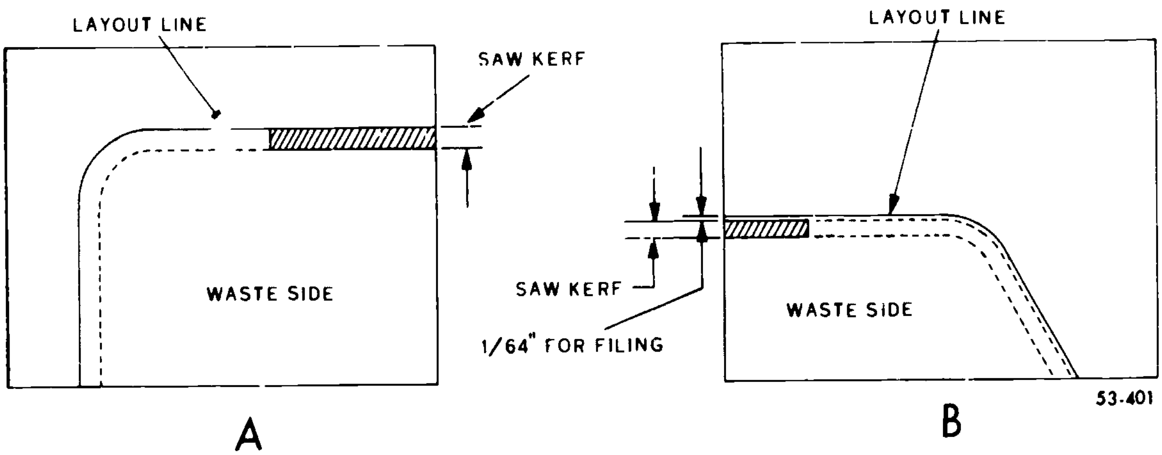


Figure 82 Location of saw kerf.

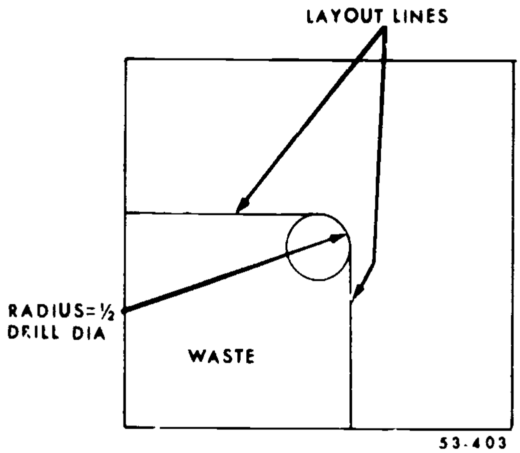


Figure 83. Drilled corner hole.

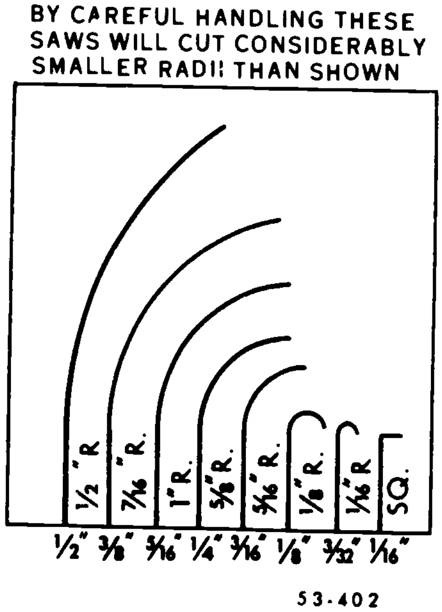


Figure 85. Saw selection for various radii.

and turn the speed change handwheel until the indicator shows the correct speed. Perform the sawing operation. When it is completed, slow the speed to about 50 fpm. Now stop the machine, and check the work for accuracy.

8-7. **Contour Sawing.** Contour sawing is sawing to a layout line of a definite radius or irregular contour. The size of the smallest radius to be cut and the thickness of the material must be considered in selecting the saw band. As the size of the radius decreases the width of the saw must be decreased to cut the curvature. You should use the widest saw band possible that will allow the contour to be cut. The saw band pitch and set must be suited to the thickness and kind of material to be sawed. Figure 85 shows the proper width of saw for various radii. This information is also given on the job selector dial.

8-8. When work is layed out, apply a thin coat of layout compound so that the scribed lines are visible and easy to follow. Work that is to be machined to a close tolerance should be scribed once to maintain a sharp layout line. Double lines, caused by scribing over a line, should be avoided. Prick punching the layout lines will often aid you in case the lines become rubbed off. Now, drill holes for sharp corners. Select the proper saw band and set up the machine for sawing. Determine the correct sawing speed. Saw the radii to specifications. NOTE: *If*

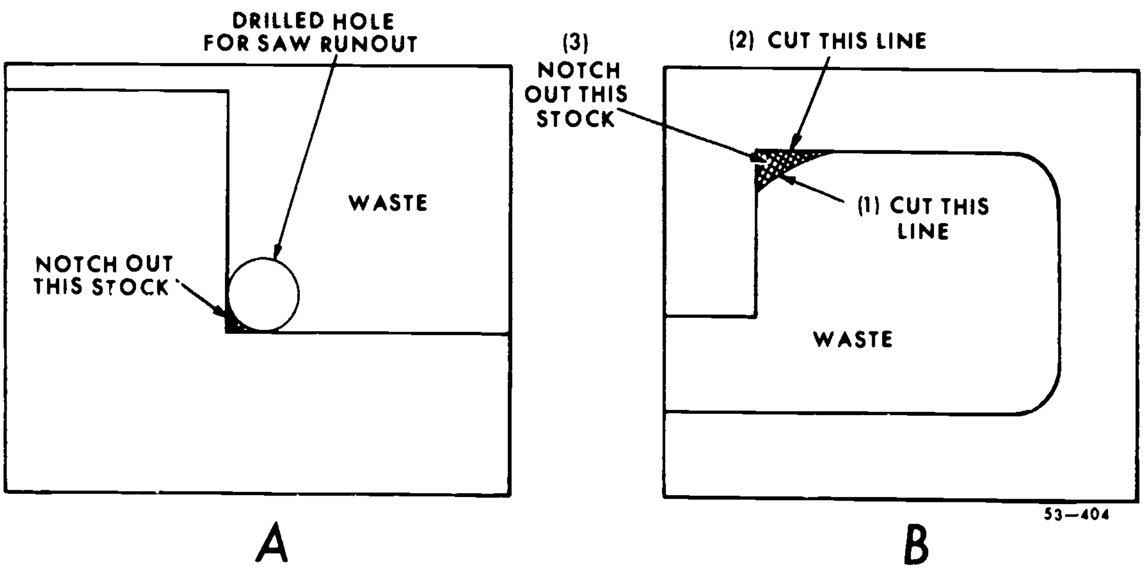


Figure 84. Notching square corners.

a smooth finish is necessary, allow about $\frac{1}{64}$ inch of stock on the waste side of the layout line for filing and polishing.

8-9. Set the machine for the recommended speed and saw the work. Remember, the edge of the saw kerf should split the layout line when a sawed finish is permissible, as in figure 82A. It should be approximately $\frac{1}{64}$ inch from the line if the work is to be filed, as shown in figure 82B. Maintain constant pressure in sawing, using a pusher block. *Be sure* that the kerf is on the waste or scrap side of the layout line! Position the blower nozzle to blow the chips off the layout line and away from you.

8-10. **Straight and Contour Filing and Polishing.** The contour machine may be used to file surfaces requiring a better finish than you can obtain by sawing. It may also be used for polishing when the file finish is not smooth enough and when more accurate dimensions are required. A file or polishing band is set up in much the same manner as for sawing. There is a continuous filing or polishing action.

8-11. *Filing.* Before filing work on a contour machine, saw the work, leaving approximately $\frac{1}{64}$ inch for filing. Place the transmission in *neutral.* Remove the saw band and determine which file band you are going to use. Install the file band guide and support. Then mount a file band (oval or half round, if concave surfaces are to be filed) and adjust it to a snug, but NOT tight, fit. Position the file band so that it tracks properly, and install the file adapter plate. Do not install the file band upside down, because doing so could ruin it and cause damage to the tires on the wheels and to the work. Determine the correct filing speed with the job selector dial. Turn the motor on and set the correct speed. File all concave surfaces first, then remove the file band and replace it with a flat file band. Use this band to file all straight and convex surfaces.

8-12. Use a light pressure when you file the work. Too much pressure will cause the file teeth to clog with chips. Move the work slowly from side to side. This will produce vertical filemarks on the filed surface, which is desirable. If you move the work too rapidly, the file marks will be diagonal. A file adapter plate, shown in figure 78, can be used in place of the standard filler plate. Position the file adapter plate close to the file band to eliminate a gap in which small pieces of work could wedge. Use a push block whenever possible while you are filing. CAUTION: KEEP YOUR FINGERS AWAY FROM THE FILE BAND. FILE BANDS CAN PULL FINGERNAILS OUT!

8-13. *Polishing.* Polishing is another operation that you can perform on the contour machine. A narrow band of abrasive cloth is used for polishing. The polishing band passes around the upper and lower wheels. Press the work against the polishing band and polish it to the desired size and finish. In addition to polishing work, you may also use the polishing band to remove scale, oxides, burrs, and toolmarks.

8-14. In polishing work, you should first mount the polishing band guide support. Lower the post to the 4-inch mark, and mount the polishing band guide on the post. Rub graphite powder on the guide to lubricate the band. Now, select a band with an abrasive of the proper grit size and mount it on the machine. Make sure that the arrow on the back of the band is pointing in the direction in which the band will travel. Adjust the tension to "snug." With the machine transmission in neutral, turn the upper wheel by hand and track the band. Insert the filler plate and start the machine. Operate it at the desired speed and proceed to polish the work as required.

9. Internal Sawing, Filing, and Polishing

9-1. Internal sawing produces a contour inside the boundaries of the external contour, as illustrated in figure 86. Fling and polishing are done on internal surfaces when greater accuracy and finishing are required.

9-2. **Internal Sawing.** The first step in internal sawing is to drill a starting hole in the waste portion of the workpiece tangent to layout line and drill any necessary corner holes, as shown in

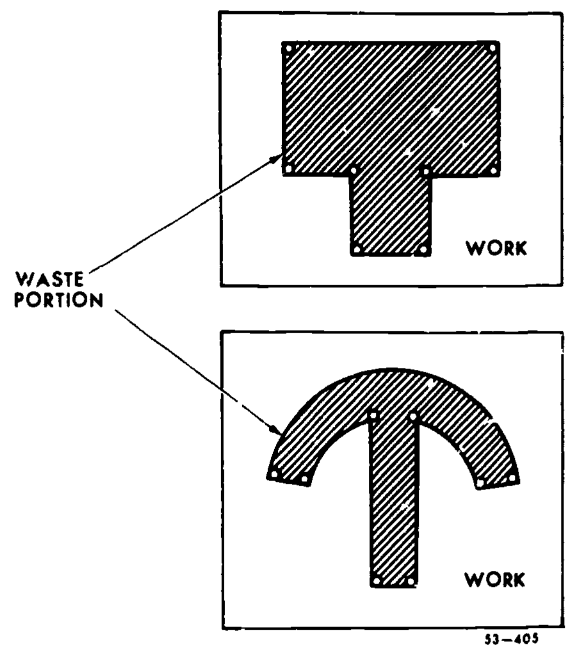


Figure 86. Internal contour.

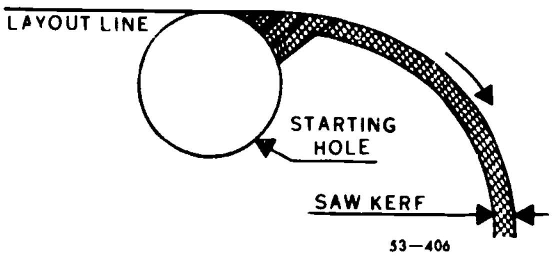


Figure 87. Sawing tangent to a starting hole

figures 86 and 87. The starting hole must be slightly larger in diameter than the width of the saw band. The width of the saw band that you should use depends upon the size of the smallest radius to be sawed. Figure 85 illustrates the largest radii that you can normally saw with saw bands of various widths. Check the table to see if it is 90° to the post. Mount the guide blocks with the proper inserts in them. Now, insert the saw band through the starting hole and weld it together. Make sure that the teeth of the saw band are pointing *down* when it passes through the hole. Place the work on the table, with the saw band in the table groove. Install the band on the wheels. Check to insure that the band is tracking properly. Then, insert the filler bar in the table groove. Turn the motor on and set the required speed. Position the air nozzle and perform the sawing. When you have finished sawing, set the machine for its lowest speed. Place the transmission shift lever in neutral and turn the motor off. Cut the saw band weld out and remove it. This keeps the number of welds on a band to a minimum.

9-3. **Internal Filing and Polishing.** You do internal filing and polishing to finish internally sawed surfaces. File bands are made with a joint or gate segment, as previously explained. This allows threading the file band through a sawed hole and joining it into a band. When polishing bands are used, they are threaded through a sawed hole and then glued together into a band. Internal polishing is limited to certain classes of large work. The width of the band and its guide will not allow polishing small and abrupt inside curves.

9-4. The selection of file and polishing bands for internal surfaces is governed by the same factors that determine the filing and polishing of outside surfaces. You use the same speeds and procedures for internal filing and polishing as for outside operations. Some classes of internal filing and polishing operations require more manual skills and exacting procedures than for outside operations. This is because of the difficult handwork and obscured vision. Chip buildup on the table and under the work presents a constant problem and can impair accuracy.

10. Attachments and Special Sawing Operations

10-1. The use of various attachments will enable you to perform certain operations with greater ease and accuracy. We will cover the common attachments and some of the special sawing operations.

10-2. **Magnifying Attachment.** You use the magnifying attachment for precision sawing and filing to close tolerances. The magnifying attachment, shown in figure 88, consists of a 3-inch rectangular lens mounted in a flanged housing. The housing contains a light socket for a 15-watt lamp. The lens and light are supported on an arm which is secured to the post by means of a C-type clamp. The arm has universal joints which permit you to set the glass at any position for all machining operations. A special plug connector on the extension cord connects with the outlet cap located on the front of the machine above the table light outlet. This outlet is fused for 1 ampere. You should not use it for any other light extension when more than 15 watts are used.

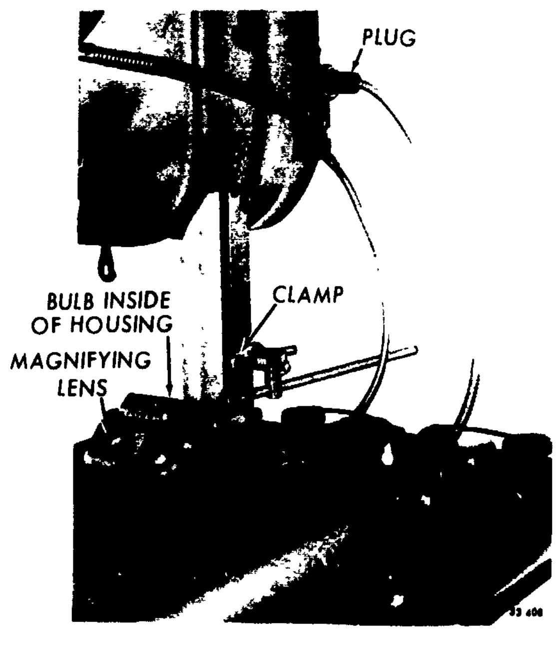
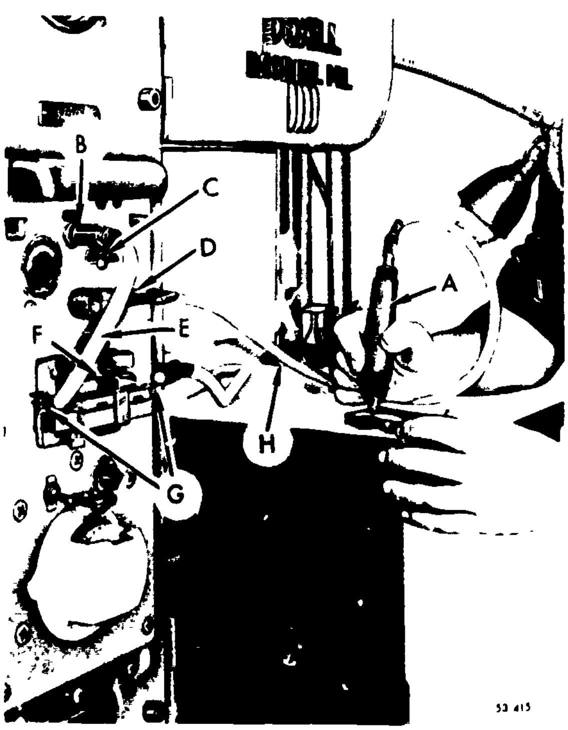


Figure 88. Magnifying attachment.

10-3. **Etching Pencil.** You may use the etching pencil, shown in figure 89, to mark metal, such as tools, jigs, and fixtures. The attachment consists of two cables with a cork-insulated copper pencil fastened to the end of the longer cable. Clamp the strip of the etching pencil cable in the movable jaw of the butt welder. Insert the ground cable in the etching pencil jack on the butt welder and clamp the terminal strip to the stationary jaw. A fiber spacer, which acts as an insulator between the jaws, also prevents any movement of the jaws when the welding lever closes the circuit. Any movement of the movable jaw will break the circuit. Place the work to be etched on the grounded table. When you apply the etching pencil to the work the circuit is closed, and the pencil burns a groove as you move it along the surface of the work.

10-4. **Power Feed Attachment.** The power feed attachment, shown in figure 90, permits you to use both hands to guide the work. The power is provided by a weight on a beam. The location of the weight on the beam determines the rate of feed and the pressure which is exerted on the saw band by the work. You vary the location of the weight by turning the power feed handwheel, shown in figure 62. Turning the handwheel clockwise reduces the pressure and rate of feed. Turning it counterclockwise increases the rate of feed. When the handwheel is in the extreme counterclockwise position, the weight exerts a pressure or pull of 60 to 75 pounds. The pressure is transmitted to the work by means of a cable and chain, as shown in figure 90. Place the power feed chain around the work, as shown in figure 90.



A. Pencil
B. Etching pencil jack
C. Voltage selector switch
D. Welding lever
E. Ground cable
F. Movable jaw
G. Terminals
H. Etching pencil cable

Figure 89. Etching pencil.

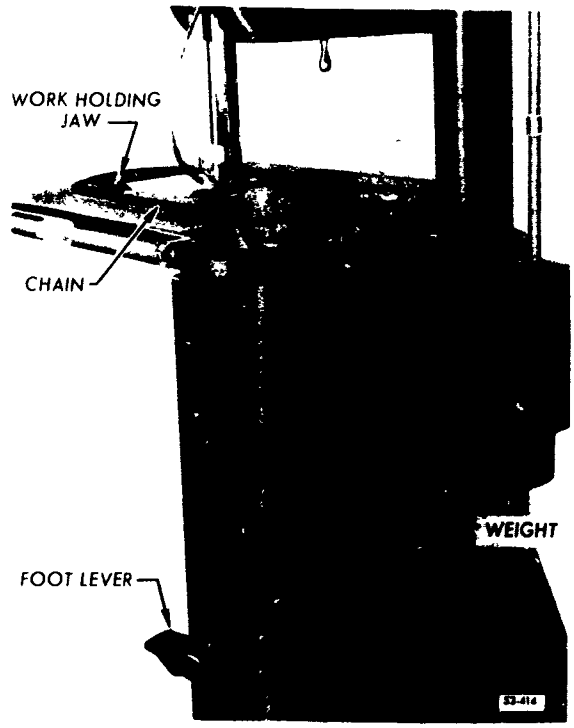


Figure 90. Power feed attachment.

10-5. **Angular Saw Guides.** Angular saw guides twist the saw band to a 30° angle to the right. This allows work to be sawed that would usually be too long to fit in the machine. In effect, angular saw guides increase the throat depth of the machine. An example is sawing the bar, as shown in figure 91. The tension must be less than normal to permit the saw band to twist without causing the inserts to wear excessively.

10-6. **Rip Fence.** You use the rip fence to cut stock so that the opposite sides are parallel. The fence must be set parallel to the table slot and located at the desired distance from the saw band. A rip fence may be improvised by clamping a metal bar to the table, as shown in figure 92. Accurate results can only be obtained if the saw is properly set up and a sharp saw band is used.

10-7. **Cutoff and Mitering Attachment.** You use the cutoff and mitering attachment, shown in figure 93, for cutting off and mitering operations. Clamp or hold the stock to be sawed in position against the attachment. You can set the attachment at an angle with a protractor, using the table slot as a reference line. A gage rod can be extended from the attachment and used as a stop

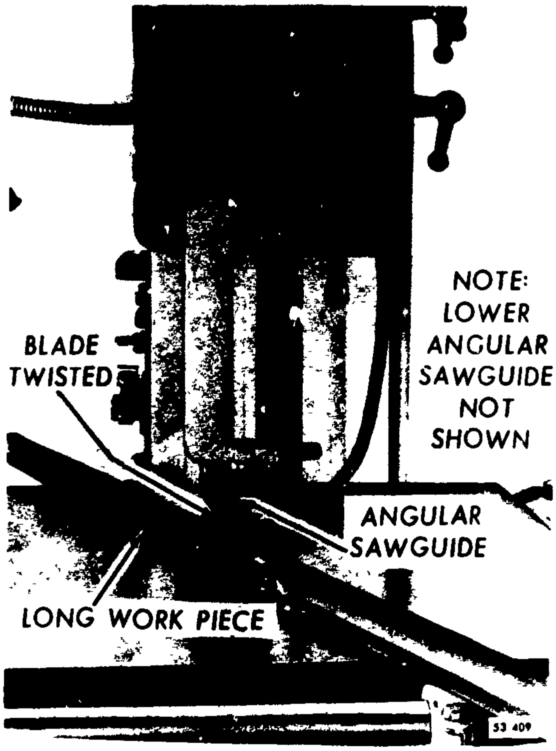


Figure 91. Angular saw guides.

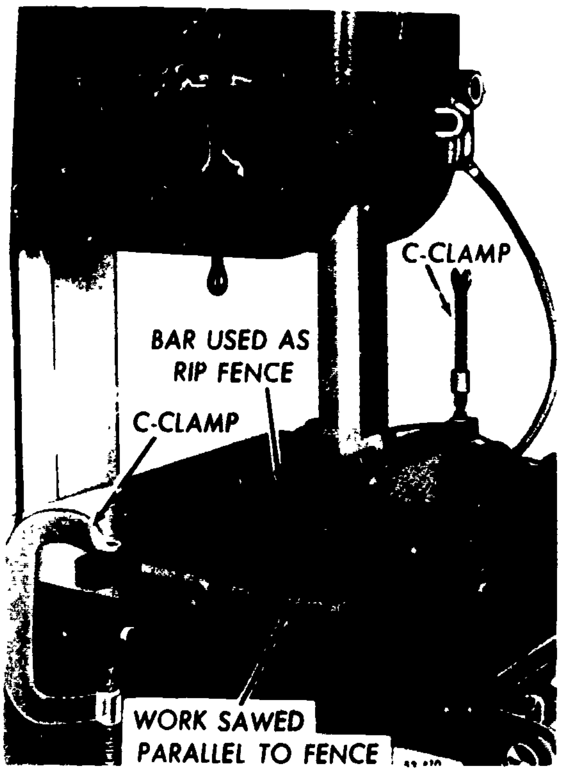


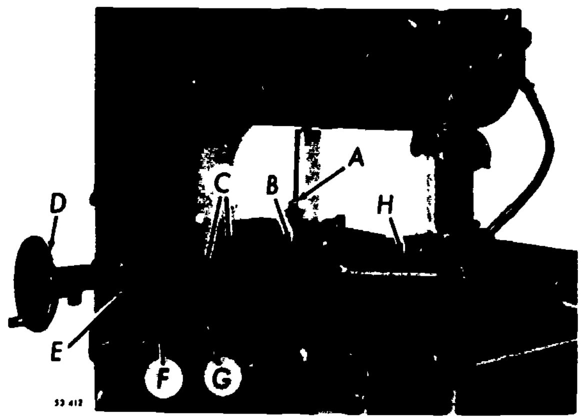
Figure 92. Rip fence.

Figure 93. Cutoff and mitering attachment.

when identical lengths are sawed. The attachment is swung on the slide rod and allowed to hang below the table top when it is not in use.

10-8. **All-Purpose Mitering Attachment.** You can perform three operations with the all-purpose mitering attachment, shown in figure 94. You can use it with hand or power feed for ripping, cutting off, or mitering. You can notch, square, rip, or miter rods, tubes, bars, channels, rails, and irregular shapes with accuracy. The attachment is mounted on the sawing side of the table and is fastened to a guiderail on the front edge of the table. The attachment has a graduated plate with an adjustable work stop on the mitering bar and a lock screw on the meter head. This enables you to set the attachment at any desired angle.

10-9. **Disc-Cutting Attachment.** You can use the disc-cutting attachment to saw internal or



A. Lockscrew
B. Degree graduations
C. Twin guide bars
D. Handwheel used to set angle
E. Guide bracket
F. Clamp
G. Guide rail
H. Work

Figure 94. All-purpose mitering attachment.

external circles and discs. The disc-cutting attachment consists of three parts: a clamp and cylindrical bar, which can be fastened to the saw guidepost; an adjustable arm that slides on the cylindrical bar; and a pivot or centering pin. These parts are shown in figure 95. The diameter of a disc is limited by the length of the cylindrical bar on the attachment. However, the diameter and location of an internal circle may be limited by the throat depth of the machine. The disc, or circle, must be laid out and the center located and drilled with a center drill to a depth of 1/8 inch to 3/16 inch to provide a pivot point for the centering pin You can feed the work into the saw band by hand or by power feed.

10-10. If you use power feed, you should lock the right cable so that all the force of the weight will be applied to the left cable. Wrap the chain on the left cable around the work two turns. When the weight is applied, the work will rotate clockwise into the saw. The centerline of the centering pin must be in line with the front edge of the saw teeth and at the desired distance from the saw band.

10-11. **Stack Sawing.** Stack sawing is the sawing of several pieces of material of the same shape in one operation. This method of duplicating identical parts saves time, since it is necessary to lay out only one piece (the top one). The number of pieces that you can stack depends upon the capacity of the saw. Do not stack pieces so high that they become top heavy. The stacked pieces must be fastened together before they are sawed. Large numbers of pieces can be compressed in an arbor press and welded together. NOTE: *Do not weld the pieces yourself. Get an authorized welder to do this for you.*

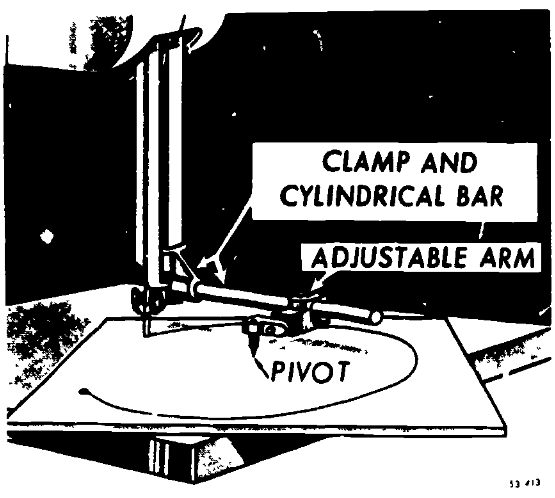


Figure 95. Disc-cutting attachment.

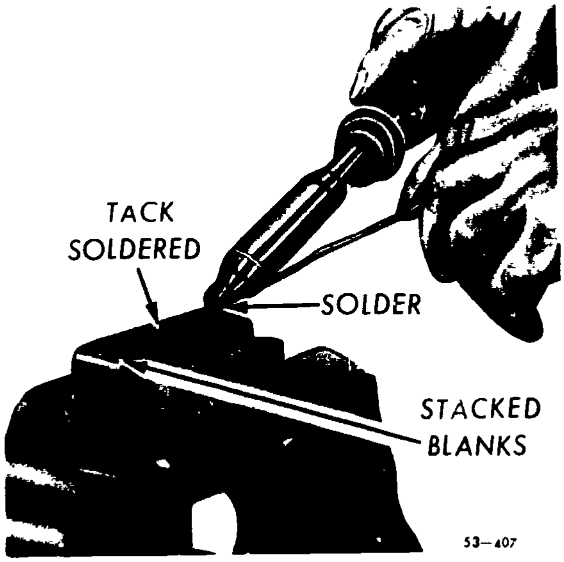


Figure 96. Compressing and soldering small objects

10-12. Small stacks can be compressed in a vise and soldered together, as shown in figure 96. They may also be drilled and fastened with bolts. The boltheads should be flush with the bottom surface of the stacked parts to allow the work to lie flat on the table. The stacked items should be considered as one solid object when the operating speeds are selected. NOTE: *All welding, soldering, or drilling should be in the scrap or waste portions of the workpieces.*

10-13. **Friction Sawing.** Friction sawing can only be used to saw hardened steels. This method of sawing makes use of the heat generated by friction. The heat generated by the contact between the saw band and the work is concentrated on the workpiece faster than it can be absorbed. The high temperature softens the metal, and the saw teeth scoop the heated metal away from the work. The temperature generated at the point of contact exceeds the red heat temperature, but is below 1600° F. The saw band does not overheat because of the limited time that a given portion of the band is in contact with the work. The teeth are air cooled before they reenter the cut. Dull saw bands friction-saw better than sharp ones. Dull teeth increase the friction, generate a higher temperature, and permit the material to be removed faster. The bands should be installed with the teeth upside down as this will increase friction, generate more heat and prevent the teeth from tearing off the band.

10-14. **Angular Sawing, Filing, and Polishing.** Sawed surfaces are usually at a 90° angle to the table surface. It is sometimes necessary for the sawed surface to be at an angle other than 90°. This is true, for example, when clearance must be provided on mating parts. The table can be tilted forward and backward up to 10°, to the left up to 10°, and to the right up to 45°. You tilt the table by loosening lockbolts located below the table and then setting it to the desired angle. The settings are indicated on graduated plates located below the table. Check the angular setting by measuring the angle formed by the post and the tabletop with a protractor head and blade. Tighten the lockbolts after you have positioned the table. Now, you can operate the machine the same way you did for straight and contour sawing. Compound angles can be sawed by tilting the table in two directions or planes at the same time. Both external and internal sawing can be performed.

10-15. Angular filing and polishing are done for the same reasons as other filing and polishing operations. Selection of bands, speeds used, and setup procedures are the same as those used for all filing and polishing. After the sawing operations, you file or polish the work as required without disturbing the table setting. This assures that the correct finished angles will be produced. External and internal mating parts are more easily fitted if the table setting is not disturbed when each part is finished.

Bibliography

USAFI Courses

Course C 835, *Shop Theory*. United States Armed Forces Institute.

Books

OBERG, Erik and Franklin D. JONES. *Machinery's Handbook,* 18th edition, New York, The Industrial Press 1968.

LE GRAND, Rupert, *The New American Machinist's Handbook*. McGraw-Hill, New York, 1955.

Dept of the Air Force Publications

AFM 127–101, *Accident Prevention Handbook*.

TO 34–1–10, *Fundamentals of Ordnance Corps Machine Tools,* 29 February 1959.

TO 34C2–8–6–62, *Operation and Maintenance, Band Saw Machines, Vertical Metal Cutting (Continental Model DoALL ML),* 2 July 1954.

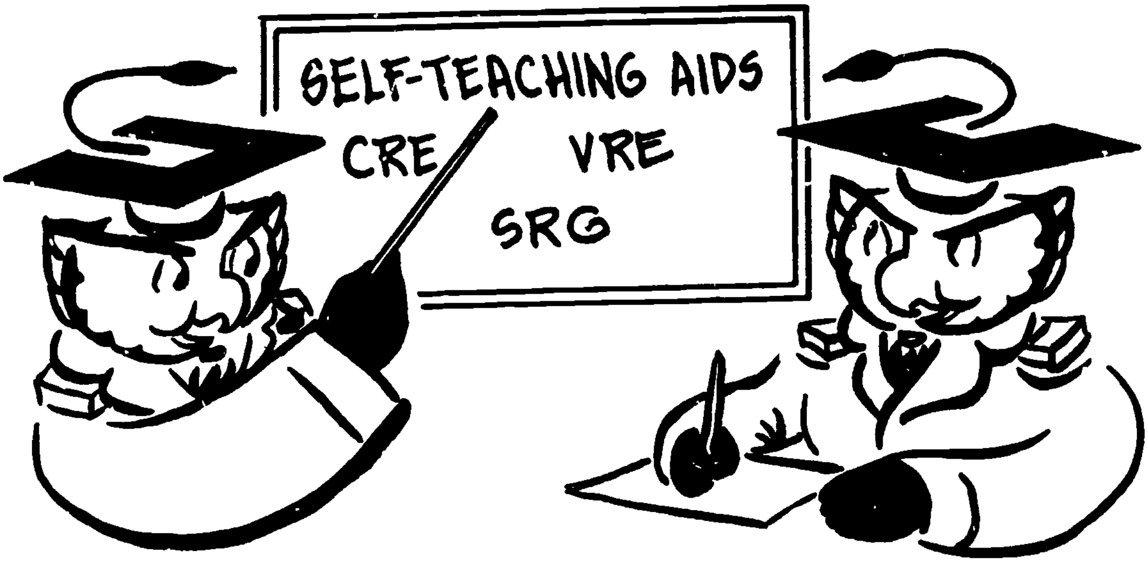
Resident Course

Training Literature from Block VI, ABR53130, *Machinist*.

NOTE None of the items listed in the bibliography above are available through ECI. If you cannot borrow them from local sources, such as your base library or local library, you may request one item at a time on a loan basis from the AU Library, Maxwell AFB, Alabama, ATTN ECI Bibliographic Assistant. However, the AU Library generally lends only *books* and a limited number of *AFMs*. TOs, classified publications, and other types of publications are *not* available. (For complete procedures and restrictions on borrowing materials from the AU Library, see the latest edition of the *ECI Catalog*.)

53130 03 21
WORKBOOK

Shaper and Contour Work



This workbook places the materials you need *where* you need them while you are studying. In it, you will find the Study Reference Guide, the Chapter Review Exercises and their answers, and the Volume Review Exercise. You can easily compare textual references with chapter exercise items without flipping pages back and forth in your text. You will not misplace any one of these essential study materials. You will have a single reference pamphlet in the proper sequence for learning.

These devices in your workbook are autoinstructional aids. They take the place of the teacher who would be directing your progress if you were in a classroom. The workbook puts these self-teachers into one booklet. If you will follow the study plan given in "Your Key to Career Development," which is in your course packet, you will be leading yourself by easily learned steps to mastery of your text.

If you have any questions which you cannot answer by referring to "Your Key to Career Development" or your course material, use ECI Form 17, "Student Request for Assistance," identify yourself and your inquiry fully and send it to ECI.

Keep the rest of this workbook in your files. Do not return any other part of it to ECI.

EXTENSION COURSE INSTITUTE

Air University

TABLE OF CONTENTS

Study Reference Guide

Chapter Review Exercises

Answers For Chapter Review Exercises

Volume Review Exercise

ECI Form No. 17

STUDY REFERENCE GUIDE

1. *Use this Guide as a Study Aid.* It emphasizes all important study areas of this volume.

2. *Use the Guide as you complete the Volume Review Exercise and for Review after Feedback on the Results.* After each item number on your VRE is a three digit number in parenthesis. That number corresponds to the Guide Number in this Study Reference Guide which shows you where the answer to that VRE item can be found in the text. When answering the items in your VRE, refer to the areas in the text indicated by these Guide Numbers. The VRE results will be sent to you on a postcard which will list the *actual VRE items you missed.* Go to your VRE booklet and locate the Guide Number for each item missed. List these Guide Numbers. Then go back to your textbook and carefully review the areas covered by these Guide Numbers. Review the entire VRE again before you take the closed-book Course Examination.

3. *Use the Guide for Follow-up after you complete the Course Examination.* The CE results will be sent to you on a postcard, which will indicate "Satisfactory" or "Unsatisfactory" completion. The card will list *Guide Numbers* relating to the questions missed. Locate these numbers in the Guide and draw a line under the Guide Number, topic, and reference. Review these areas to insure your mastery of the course.

Guide Number — *Guide Numbers 300 through 310*

Guide Number	
300	Introduction; Construction and Operation; Construction; pages 1-3
301	Construction and Operation; Operation; pages 4-8
302	Workholding Devices; pages 8-11
303	Shaper Cutting Tools and Toolholders; pages 11-16
304	Introduction; Machining Horizontal, Vertical, Angular, and Contoured Surfaces; pages 17-21
305	Machining Shoulders, Slots and Grooves, and Keyseats; pages 21-25
306	Introduction; Construction and Operation; pages 26-28
307	Saw, File, and Polishing Bands: Saw Bands; pages 28-35
308	Saw, File, and Polishing Bands: File Banks; Polishing Bands; pages 35-39
309	Introduction; Straight and Contout Sawing, Filing, and Polishing; Internal Sawing, Filing, and Polishing; pages 40-44
310	Attachments and Special Sawing Operations; pages 44-48

CHAPTER REVIEW EXERCISES

The following exercises are study aids. Write your answers in pencil in the space provided after each exercise. Immediately after completing each set of exercises, check your responses against the answers for that set. Do not submit your answers to ECI for grading.

CHAPTER 1

Objectives: To exhibit an understanding of the construction and operation of shapers; to show ability to identify the various types of workholding devices and to explain their use; and to be able to identify the various types of shaper cutting tools and toolholders and to explain their use.

1. When does the cutting action take place in the operation of a shaper? (1-2)

2. What designates the size of a shaper? (1-2)

3. Which type of shaper is the most common? (1-3)

4. How can you tell the difference between a universal shaper and a plain shaper? (1-5)

5. What causes the ram to move back and forth? (1-6,*b*)

6. What two mechanisms are contained in the crossrail? (1-6,*c*)

7. What is the purpose of the rapid traverse? (1-6,*f*)

8. How long should the stroke be in relation to the work? (1-8)

9. Why is a greater portion of the overstroke at the rear rather than at the front of the work? (1-9)

10. At what speed should a shaper be operated if the recommended CFS is 30 and the length of stroke is 10 inches? (1-10)

11. What is feed in shaper operations? (1-12)

12. What are the two functions of the directional feed control lever? (1-13)

13. How can the table be moved by hand? (1-14)

14. When is the rapid travese used? (1-15)

15. What part of the toolhead controls the depth of cut? (1-17)

16. What is the maximum depth of a roughing cut that is allowable? (1-19)

17. What determines the direction of clapper box tilt? (1-21)

18. What should be done to the table surface before alining it? (1-24)

19. What is one of the most important requirements that must be met before producing an accurately squared surface when the work is held in a vise? (1-27)

20. What is proper toolhead alinement? (1-28)

21. How may work be held in a shaper? (2-1)

22. What are two common types of shaper vises? (2-2)

23. What are parallels used for? (2-4)

24. What are two common types of angle plates? (2-5)

25. What is a jack used for on a shaper? (2-8)

26. What is another name for a heavy-duty roughing tool used in s shaper? (3-3,*a*)

27. What type of tool produces the best finish on s[illegible] and aluminum? (3-3, *b*, *c*)

28. What should be done to work before using a form tool? (3-3,*g*)

29. What angle must be compensated for in grinding end relief and back rake angles on a shaper tool? (3-5)

30. What toolholder is used to cut an internal spline? (3-6)

CHAPTER 2

Objectives: To demonstrate a knowledge of the planing of horizontal, vertical, angular, and contoured surfaces. To be able to explain the techniques for planing shoulders, slots and grooves, and keyseats.

1. What is the most common direction of feed in using a shaper on a horizontal surface? (4-2)

2. How can adjacent surfaces be machined perpendicular to one another in a shaper? (4-4)

3. Why is shim stock used between the work and the vise jaws? (4-6)

4. When can a roughing tool be used for the finish cut? (4-7)

5. When machining vertical surfaces, and great accuracy is *not* required, how can the toolhead and the vise be aliι.ed? (4-9)

6. In what range should the depth of cut in machining vertical surfaces be? (4-10)

7. How may angular surfaces be machined in a shaper? (4-12)

8. When is swiveling the vise the method to use in machining an angular surface? (4-13)

9. What is the difference in the method of feed in cutting angular surfaces by swiveling the toolhead and by seiveling the table? (4-14, 15)

10. What limitation applies to using an adjustable angle plate in a shaper? (4-16)

11. How are contour surfaces mahined in a shaper? (4-19)

12. How are shoulders machined in a shaper? (5-2)

13. How can time be saved in roughing out material between two shoulders? (5-3)

14. What maximum depth of cut should be used in vertical feeding? (5-5)

15. What are the best types of tools to use for finishing vertical surfaces? (5-6)

16. What is the difference between a keyway and a keyseat? (5-10)

17. To what depth and what diameter should the drill be that is used at the beginning and the end on an external keyseat machined in the middle of a shaft? (5-11)

18. How are depths of cut made when cutting an internal keyseat on a shaper? (5-14)

CHAPTER 3

Objectives: To demonstrate a knowledge of the construction and operation of a contour machine and a knowledge of the essential features of saw, file, and polishing bands.

1. What is a job selector dial? (6-3)

2. How can table alinement be checked? (6-4)

3. What are the two speed ranges of a contour machine? (6-5)

4. At what speed should the shift lever be moved from one range to the other? (6-6)

5. What functions do saw band gullets perform? (7-3*c*)

6. What is usually the gage of a saw band that is 3/8" wide? (7-3*e*)

7. What is the minimum number of teeth that should be in contact with the work at all times? (7-5)

8. What type of set pattern should be used on thin work sections and tough materials? (7-5)

9. Where can you locate information concerning the length of the new saw blade that should be cut off before welding a sawband? (7-6)

10. What will happen to small saw bands if too much tension is applied during the welding operation? (7-7)

11. Why do saw band teeth have set? (7-5*g*)

12. For what purpose is a grinder installed on a contour machine? (7-10)

13. When should the down pressure on the welder operating lever be released? (7-11)

14. What is the purpose of the hardened steel inserts mounted in the guide blocks? (7-14, 15, 16)

15. What is used to correctly position the saw guide inserts? (7-17)

16. How is the saw band made to track in the correct location? (7-18)

17. Why is maintaing the correct saw band tension necessary? (7-19)

18. What position should the transmission lever be when mounting a saw band? (7-20)

19. What are the advantages of using the widest file possible? (7-21)

20. Why are file segments fastened to the spring steel band by only one end? (7-22)

21. Why should kinking of the spring steel band be avoided? (7-25)

22. What factor determines the speed at which a polishing band is operated? (7-28)

23. What tension is placed on a polishing band? (7-29)

CHAPTER 4

Objectives: To demonstrate a working knowledge of straight and contour sawing, filing, and polishing; of internal sawing, filing, and polishing; and of the use of various attachments in performing special sawing operations.

1. What controls the accuracy of contour machine work? (8-1)

2. What factors govern the speed used to saw work? (8-3)

3. Why are thinner bands required for contour sawing than for straight sawing? (8-4)

4. What are two safety precautions to observe in sawing on a contour machine? (8-5)

5. What should you do if a saw band breaks while you are sawing? (8-5)

6. At what speed should a saw be running when it is shut off? (8-6)

7 What factors must be considered in the selection of a saw band? (8-7)

8. How should layout lines be scribed? (8-8)

9. How can a radius that is too small to be cut with a saw band be machined in the work? (8-8) (8-8)

10. How much material is left on the work for filing? (8-9)

11. When work has convex and concave surfaces to be filed on a contour saw, which should be filed first? (8-11)

12. What shapes of file bands are used to file concave surfaces? (8-11)

13. What amount of pressure is used in filing work? Explain your answer. (8-12)

14. Besides polishing, what are polishing bands used for? (8-13)

15. How are polishing bands lubricated? (8-14)

16. How large should the starting hole be when preparing to perform internal sawing operations? (9-2)

17. What are some of the problems that filing internal surfaces presents? (9-4)

18. What attachment is used to assist in precision sawing and filing? (10-2)

19. How is electrical power provided for the etching pencil? (10-3)

20. How is the rate of feed increased on a contour saw? (10-4)

21. How is power provided for the power feed? (10-4)

22. At what angle do angular saw guides twist the saw band at the point of cut? (10-5)

23. How can stock be sawed so that the opposite sides are parallel? (10-6, 8)

24. What should be done to work after laying out the circle and before inserting it into the disc-cutting attachment? (10-9)

25. What are the advantages of stack sawing? (10-11)

26 What is the principle of friction sawing? (10-13)

27. In what directions and how many degrees can the contour machine table be tilted? (10-14)

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 1

1. During the forward stroke of the ram.
2. The maximum length of the stroke.
3. The horizontal shaper.
4. A universal shaper table can be swiveled in a vertical plane and one surface can be tilted, while a plain shaper table cannot be swiveled or tilted.
5. The rocker arm.
6. The table elevating mechanism and the table traverse mechanism.
7. To reposition the table and the work quickly after the cutting stroke.
8. One inch longer than the length of the work.
9. The small amount of overstroke on the forward stroke is mainly to permit the chip to break off, while the larger overstroke on the return stroke permits the clapper box to seat before the cutting stroke and the feed to take place.
10. 21 strokes per minute.
11. The distance the table moves the work toward the tool before each cutting stroke.
12. To engage the power feed and to determine the direction of the table travel.
13. By placing the directional feed control lever in the neutral position and turning the cross feed screw with the hand crank.
14. When setting up the tool or work, and when returning work to its starting position for another cut.
15. The toolslide.
16. Roughing cuts can be as deep as the toolwork and setup permit.
17. The direction of the cut and the operation that is being performed.
18. Clean it and remove any burrs.
19. Accurate vise alinement.
20. When the toolhead is perpendicular to the table surface or to the inner face of the stationary vise jaw.

21. In a vise, between index centers, clamped to an angle plate, or fastened directly to the shaper table.

22. Single-screw and double-screw vises.

23. Supporting work at a convenient height while maintaining parallelism between the bottom of the work and the shaper table or vise.

24. Standard nonadjustable and adjustable.

25. For leveling work and supporting portions that could spring.

26. A bullnose roughing tool .

27. A shear tool.

28. Rough out the form with a roundnose tool and use the form tool to finish the machine operation.

29. The angle at which the holder holds the cutting tool.

30. An extension tool holder.

CHAPTER 2

1. Laterally below the ram, from left to right.

2. By machining them horizontally in the proper sequence.

3. To prevent damage to the vise jaws, increase vise holding power, compensate for unevenness in the work, and protect finished work surfaces.

4. When an exceptionally good finish is not required.

5. By their graduations.

6. From 1/32 to 1/16 inch, depending on the rigidity of the setup.

7. Swiveling the vise or the toolhead, swiveling or tilting the table, mounting the work on an adjustable angle plate, or holding it in a fixture.

8. When the surface to be machined is perpendicular to the surface of the table or the bottom of the vise and at an angle to the stroke of the ram.

9. The tool is fed by hand when the toolhead is swiveled, and power feed is used when the table has been swiveled.

10. It should only be used for light duty applications.

11. By using full or partial form cutting tools or by hand manipulation of tool and feed.

12. By using a roundnose tool to remove excess material, and a side-finishing or squaring tool to finish the surfaces.

13. By feeding the work in each direction on successive cuts.

14. .050 inch.

15. Side finishing and squaring.

16. A keyseat is a single groove or slot in a shaft that is alined with a like groove in a mating part to form a keyway.

17. The drill diameter should be equal to the keyseat width, while its depth, except for the drill point, should be equal to the keyseat depth.

18. By feeding the tool upward with the toolslide.

!APTER 3

1. It is a dial mounted on the upper door of a contour machine that gives information on sawing, filing, and polishing more than 50 different materials.

2. By placing a square against the saw guide support post and the table.

3. Low range is 0-400 fpm; high range is 0-1600 fpm.

4. Approximately 50 fpm or lower.

5. They provide chip clearance and help to remove the chips from the cut.

6 .025 inch.

7. Two.

8. The wave set.

9. By consulting the instruction manual or technical order for that particular machine.

10. The saw band ends will overlap instead of butt together during the welding operation.

11. To prevent the band from binding and to provide a clearance that permits cutting radii.

12 It is used to square the saw band ends before welding and to remove excess weld after the welding has taken place.

13. After the stationary jaw has been loosened.

14. They guide and support the saw band.

15. An insert gage.

16. By tilting the upper wheel in or out as necessary.

17. To prevent its twisting and not sawing straight.

18. Neutral.

19. The files remain sharper longer and less time is needed for filing.

20. This allows the band to bend as it moves around the upper and lower wheels and to lock in place as they move past the work.

21. A kink prevents interlocking of segments which causes rough, uneven filing.

22. The grit size of the band.

23. The tension should be just snug enough to eliminate slippage.

CHAPTER 4

1. The skill and knowledge of the operator.

2. The type and thickness of the work.

3. To prevent rubbing the bands on the sides of the cut.

4. Use a wooden push stick to apply feed pressure and wear eye protection

5. Stand clear of the machine and turn the power off.

6. About 50 fpm.

7. The size of the radius to be sawed and the thickness of the material.

8. Just once, with a sharp scriber.

9. By drilling a hole whose radius equals the desired radius.

10. 1/64 inch.

11. The concave surfaces.

12. Oval or half round.

13. Use a light pressure to prevent clogging the file teeth with chips.

14. Removal of scale, burrs, oxides, and toolmarks.

15. By rubbing graphite powder on the guide.

16. The diameter should be slightly greater than the saw band width.

17. Obscured vision, exacting procedures, difficult handwork, and chip buildup under the work.

18. The magnifying attachment.

19. By clamping the strip of the etching pencil cable into the butt welder.

20. By a weight mounted on a beam.

21. By turning the power feed handwheel counterclockwise.

22. 30° to the right.

23. By using a rip fence or an all-purpose mitering attachment.

24. Drill a center hole in the center to a depth of 1/8 to 3/16 inch

25. It saves time, since it is necessary to lay out only one piece. Also several pieces may be sawed at the same time.

26. Heat generated between the work and the saw band is concentrated on the work much faster than it can be absorbed, with the saw's teeth carrying the heated metal away from the work.

27. Forward and backward 10°. left 10°, and right 45°

S T O P -   1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER.
2. USE NUMBER 2 PENCIL ONLY.

EXTENSION COURSE INSTITUTE
VOLUME REVIEW EXERCISE

53130 03 21

Carefully read the following:

DO'S:

1. Check the "course," "volume," and "form" numbers from the answer sheet address tab against the "VRE answer sheet identification number" in the righthand column of the shipping list. If numbers do not match, take action to return the answer sheet and the shipping list to ECI immediately with a note of explanation.
2. Note that item numbers on answer sheet are sequential in each column.
3. Use a medium sharp #2 black lead pencil for marking answer sheet.
4. Write the correct answer in the margin at the left of the item. (When you review for the course examination, you can cover your answers with a strip of paper and then check your review answers against your original choices.) After you are sure of your answers, transfer them to the answer sheet. If you have to change an answer on the answer sheet, be sure that the erasure is complete. Use a clean eraser. But try to avoid any erasure on the answer sheet if at all possible.
5. Take action to return entire answer sheet to ECI.
6. Keep Volume Review Exercise booklet for review and reference.
7. If mandatorily enrolled student, process questions or comments through your unit trainer or OJT supervisor. If voluntarily enrolled student, send questions or comments to ECI on ECI Form 17.

DON'TS:

1. Don't use answer sheets other than one furnished specifically for each review exercise.
2. Don't mark on the answer sheet except to fill in marking blocks. Double marks or excessive markings which overflow marking blocks will register as errors.
3. Don't fold, spindle, staple, tape, or mutilate the answer sheet.
4. Don't use ink or any marking other than a #2 black lead pencil.

NOTE: **STUDY REFERENCE GUIDE NUMBERS ARE USED ON THE VOLUME REVIEW EXERCISE.** In parenthesis after each item number on the VRE is the Study Reference Guide Number where the answer to that item can be located. When answering the items on the VRE, refer to the Study Guide Numbers indicate by these numbers. The VRE results will be sent to you on a postcard which will list the actual VRE items you missed. Go to the VRE booklet and locate the Study Reference Guide Numbers for the items missed. Go to the text and carefully review the areas covered by these references. Review the entire VRE again be e you take the closed-book Course Examination.

Multiple Choice

Chapter 1

1. (300) The size of a shaper is determined by the

a. maximum capacity of the vise.
b. maximum stroke length.
c. length of the ram.
d. size of the table.

2. (300) The column and base of a shaper are designed

a. without a bearing surface for the ram.
b. with horizontal ways upon which the crossrail slides.
c. to maximize vibration.
d. to minimize vibration.

30 (300) The front part of the cylindrically machined casting which is gibbed to the crossrail is called the

a. ram.
b. column.
c. trunnion.
d. tilting surface.

4. (300) The location of the sliding block and pin with respect to the center of the bull wheel governs the

a. speed of the cutting stroke.
b. rate of table feed per stroke.
c. speed of the return stroke.
d. length of the cutting stroke

5. (301) Generally, the amount of overstroke for work in a shaper is

pendent on the work shape.
. dependent on the work material.
c. 1 inch longer than work length.
d. 1 inch per foot of work length.

6. (301) The formula $N = \frac{CFS \times 7}{L}$ is used to determine the cutting foot speed for shapers.

The machinist's handbook indicates that the recommended CFS is 20 for the metal you are to use. You need to rough machine a 6-inch piece of the metal. At what speed should you operate the shaper?

a. 20 strokes per minute
b. 2 strokes per minute.
c. $23\frac{1}{3}$ strokes per minute.
d. $23\frac{1}{3}$ inches per stroke

7 (301) The normal position of s shaper toolhead during use is

a. horizontal.
b. vertical.
c. 45° to the work.
d. 60° to the work.

(301) The depth of a shaper cut is controlled by the

a. toolslide.
b. toolhead.
c. lock screw.
d. clapper box.

9. (301) Roughing cuts for shaping metal

a. normally range from 0.010 - 0.020 inch.
b. should not exceed 0.005 inch.
c. should be 0.085 inch.
d. may be as deep as tool, work, and setup permit.

10. (301) To prevent damage to the work and cutting tool during the return stroke, the top of the clapper box should be

a. positioned opposite the cut direction
b. set perpendicular to the work.
c. positioned toward the direction of cut.
d. set to any convenient position.

11. (302) The work-holding device that makes possible machining operations on both horizontal and vertical positions is the

a. holddown strap.
b. adjustable angle plate.
c. swivel vise.
d. adjustable parallel.

12. (302) Light work that may spring during the machining operation is generally supported by

a. an angle plate.
b. an adjustable parallel.
c. a jack.
d. a bunter.

13. (303) The primary difference between lathe and shaper cutting tools is that shaper tools have

a. less effective side and end relief.
b. opposite angles to that of lathe tools.
c. more effective side and end relief.
d. no back or side rake angles.

14. (303) Two types of shaper tools that are usually ground with 0° side rake are the

a. shovelnose and parting tools.
b. roughing and roundnose tools.
c. form and roundnose tools.
d. shear and side-finishing tools.

15. (303) The toolholder used with a form tool to absorb vibration is the

a. spring type
b. swivel head type.
c. gooseneck type.
d. extension type.

Chapter 2

16. (304) The factors which restrict the types of shaper work that can be done are the maximum length of the stroke and the

a. inability to produce contour devices.
b. inability to produce slot and keyseats.
c. capacity of workholding devices.
d. multidirectional stroke of the shaper.

17. (304) The recommended direction of feed for horizontal planing that permits viewing the tool cutting action and the work surface is

a. down .
b. up.
c. from right to left.
d. from left to right.

18. (304) Vise jaws should not be tightened after seating work on parallels to prevent

a. marring the work surface.
b. unseating the work.
c. damaging the parallels.
d. damage to the vise jaws.

19. (304) The first step in machining a horizontal surface is to aline the stationary jaw and the bottom of the vise

a. perpendicular to the ram travel.
b. parallel to the ram travel.
c. perpendicular to the toolslide.
d. parallel to the toolslide.

20. (304) If the same tooling setup used to machine a vertical surface is to be used to machine an angular surface, which of the following methods should be used?

a. Use a fixture to hold the work.
b. Swivel the table.
c. Swivel the toolhead.
d. Swivel the vise.

21. (305) The best way to save time in roughing out material between two shoulders is to

a. take deeper depths of cut than normal.
b. increase the ram strokes per minute.
c. increase the feed per stroke.
d. feed work in both directions.

22. (305) To alter the depth of cut in machining a vertical surface with a shaper, the work is moved with the

a. toolslide.
b. crossfeed.
c. toolhead.
d. vertical feed.

23. (305) When using a squaring tool to finish a vertical surface, the top of the clapper box should be

a. swiveled away from the work.
b. swiveled toward the work.
c. horizontal.
d. vertical.

24. (305) The main difference between a shaper squaring tool and a shaper parting tool is the

a. back rake.
b. end relief.
c. cutting edge width.
d. side rake.

25. (305) Two adjacent holes are drilled and elongated at the start of an external keyseat to permit the

a. cutting tool to drop into its cutting position.
b. full length of the keyseat to be machined.
c. additional clearance between the tool and work
d. chips to break off without difficulty.

26. (305) When machining an external keyseat on a shaper, the clapper box should be

a. swiveled to the left.
b. swiveled to the right.
c. in the horizontal position.
d. in the vertical position.

27. (305) The setup for machining an internal keyseat differs from other types of setups primarily in that the clapper box is

a. locked in ie vertical position and the feed is downward.
b. locked in the vertical position and the feed is upward.
c. loose and can be fed in either direction.
d. swiveled toward the operator and the feed is upward.

Chapter 3

28. (306) The size of a contour machine is determined by the

a. height of the machine.
b. table size.
c. throat depth.
d. length of saw band.

29. (306) The maximum speed at which a contour machine can be shifted from one speed range to the other speed range is

a. 0 fpm.
b. 200 fpm.
c. 100 fpm.
d. 50 fpm.

30. (306) If the middle cone of the variable speed unit of a contour machine is shifted, the

a ratio between the two pulleys changes.
b. ratio betv the two pulleys remains constant.
c. diameter of both pullevs decreases.
d. diameter of both pulleys increases.

31. (307) The type of a saw band refers to the

a. width of the band.
b.. gage of the band.
c. spacing and shape of the teeth.
d. temper of the band.

32. (307) The set of a saw band is the

a. amount of bend given the teeth.
b. distance from back edge to tooth tip.
c. slot produced by the sawing action.
d. band-back thickness.

33. (307) In addition to the amount of set, the kerf width .s dependent upon the

a. band width and rate of feed.
b. gage and saw pitch.
c set pat rn and gullet size.
d. saw ptich and rate of feed.

34. (307) Which of the following provides the clearance that makes radius cutting possible?

a. Pitch.
b. Set.
c. Width.
d. Gage.

35. (307) To form an endless look, the ends of a saw band are

a. riveted.
b. lap welded.
c. butt welded.
d. interlocked.

36. (307) What prevents the saw band from springing as cutting pressure is applied?

a. Guide blocks.
b. Two thrust rollers.
c. Steel inserts.
d. An alining pin.

37. (307) The saw band is made to "track" in the slots formed by the guide inserts by

a. tilting the upper wheel.
b. adjusting the thrust bearings.
c. tilting the lower wheel.
d. adjusting the guide post.

38. (307) After a saw band has been in use for some time, the tension is readjusted by raising the upper

a. block.
b. wheel.
c. insert.
d. roller.

39. (307) Too much tension on a saw band will cause

a. the band to twist.
b. wear on the saw guides.
c. wheel tires to wear.
d. the saw band to slip.

40. (308) To replace a damaged file segment, the replacement segment is

a. riveted to the file band at the leading end.
b. interlocked at the leading end with the adjacent segment.
c. riveted to the file band at the trailing end.
d. interlocked at the trailing end with the adjacent segment.

Chapter 4

41. (309) To prevent the band from twisting when you are making a straight cut, you should use the

a. thinnest available saw band of the proper pitch.
b. widest available saw band of the proper pitch.
c. slowest cutting speed.
d. lightest feed pressure.

42. (309) To avoid dulling the saw teeth, the pressure of the feed against the saw blade should be

a. constant.
b. flexible.
c. heavy.
d. light.

43. (309) Radii of ½ inch, ⅓ inch, 1 inch, and 1¼ inches are to be sawed. The radius used to determine the width of the saw band selected would be

a. 1 inch
b. ¾ inch.
c. ⅓ inch.
d. 1¼ inches.

44. (309) The shape of the file ' sed on convex surfaces is

a. flat.
b oval.
c. half round.
d. round.

45. (309) The first step in internal sawing operations is

a. cutting the saw band to length.
b. welding the saw band together.
c. locating and drilling the starting hole.
d. threading the saw in the starting hold.

46. (310) The etching pencil is used to

a. burn burrs off work.
b. cut hardened steel.
c. finish small contours.
d. mark work and tools.

47. (310) Which of the following attachments is used to increase the throat depth of a contour machine to accomodate work that would otherwise be too long for the machine?

a. Power feed attachment.
b. Rip fenees.
c. Angular saw guides.
d. Mitering attachments.

48. (310) The attachment that enables you to do accurate ripping, notching, and squaring is the

a. rip fence.
b. all-purpose mitering attachment.
c. cutt-off attachment.
d. angular saw guides.

49. (310) If you are to cut 12 identical pieces of work from 1/4-inch-thick material by stack sawing two stacks of 6 pieces, you would select the proper cutting speed based on a thickness of

a. 1/4 inch.
b. 3 inches
c. 1¾ inches.
d. 1½ inches.

50. (310) For most effective friction sawing

a. the teeth of the saw band must be right side up.
b. sharp saw bands should be used.
c. dull saw bands should be used.
d. the saw must be stopped periodically to allow the band to cool.

STUDENT REQUEST FOR ASSISTANCE

PRIVACY ACT STATEMENT

AUTHORITY: 44 USC 3101. PRINCIPAL PURPOSE(S): To provide student assistance as requested by individual students. ROUTINE USES: This form is shipped with every ECI course package. It is utilized by the student, as needed, to place an inquiry with ECI. DISCLOSURE: Voluntary. The information requested on this form is needed for expeditious handling of the student's need. Failure to provide all information would result in slower action or inability to provide assistance.

SECTION I: CORRECTED OR LATEST ENROLLMENT DATA: MAIL TO ECI, GUNTER AFS, ALA 36118

1. THIS REQUEST CONCERNS COURSE
2. TODAY'S DATE
3. ENROLLMENT DATE
4. PREVIOUS SERIAL NUMBER
5. SOCIAL SECURITY NUMBER
6. GRADE/RANK
7. INITIALS
LAST NAME
8. OTHER ECI COURSES NOW ENROLLED IN
9. ADDRESS: (OJT ENROLLEES - ADDRESS OF UNIT TRAINING OFFICE/ALL OTHERS - CURRENT MAILING ADDRESS)
ZIP CODE
10. NAME OF BASE OR INSTALLATION IF NOT SHOWN ABOVE:
11. AUTOVON NUMBER
12. TEST CONTROL OFFICE ZIP CODE/SHRED

SECTION II: Old or INCORRECT ENROLLMENT DATA

1. NAME:
2. GRADE/RANK:
3. SSAN:
4. ADDRESS:
5. TEST OFFICE ZIP/SHRED:

SECTION III: REQUEST FOR MATERIALS, RECORDS, OR SERVICE

(Place an "X" through number in box to left of service requested)

ADDITIONAL FORMS 17 available from trainers, OJT and Education Offices, and ECI. The latest course workbooks have a Form 17 printed on the last page.

1	EXTEND COURSE COMPLETION DATE. (Justify in Remarks)
2	SEND VRE ANSWER SHEETS FOR VOL(s): 1 2 3 4 5 6 7 8 9 - ORIGINALS WERE: NOT RECEIVED, LOST, MISUSED
3	SEND COURSE MATERIALS (Specify in remarks) - ORIGINALS WERE: NOT RECEIVED, LOST, DAMAGED.
4	COURSE EXAM NOT YET RECEIVED. FINAL VRE SUBMITTED FOR GRADING ON (Date):
5	RESULTS FOR VRE VOL(s): 1 2 3 4 5 6 7 8 9 NOT YET RECEIVED. ANSWER SHEET(s) SUBMITTED ON (Date):
6	RESULTS FOR CE NOT YET RECEIVED. ANSWER SHEET SUBMITTED TO ECI ON (Date):
7	PREVIOUS INQUIRY (ECI FORM 17, LTR, MSG) SENT TO ECI ON:
8	GIVE INSTRUCTIONAL ASSISTANCE AS REQUESTED ON REVERSE:
9	OTHER (Explain fully in remarks)

REMARKS: (Continue on Reverse)

OJT STUDENTS must have their OJT Administrator certify this request.	I certify that the information on this form is accurate and that this request cannot be answered at this station. (Signature)
ALL OTHER STUDENTS may certify their own requests.	

ECI FORM 17 JUN 77 OLD EDITIONS MAY BE USED

SECTION IV: REQUEST FOR INSTRUCTOR ASSISTANCE

NOTE: Questions or comments relating to the accuracy or currency of textual material should be forwarded directly to preparing agency. Name of agency can be found at the bottom of the inside cover of each text. All other inquiries concerning the course should be forwarded to ECI.

VRE ITEM QUESTIONED:

Course No. __________

Volume No. __________

VRE Form No. __________

VRE Item No. __________

Answer You Chose (Letter) __________

Has VRE Answer Sheet been submitted for grading?

☐ YES ☐ NO

REFERENCE
(Textual support for the answer I chose can be found as shown below)

In Volume No: __________

On Page No: __________

In______ (Left)_____(Right) Column

Lines_____Through______

MY QUESTION IS:

Remarks:

53130 04 1171

CDC 53130

APPRENTICE MACHINIST

(AFSC 53130)

Volume 4

Milling and Grinding Machine Work

Extension Course Institute

Air University

Preface

VOLUME 4 deals with milling and grinding machine work. Chapter 1 covers the introduction to the milling machine; Chapter 2, milling machine operations; Chapter 3, grinding machine work; Chapter 4, fitting and assembly; and Chapter 5, machine tool maintenance. It will help you to understand the scope of this volume if you leaf quickly through the pages of each chapter and note the numbered headings.

Code numbers appearing on the figures are for preparing agency identification only.

If you have questions on course enrollment or administration, or on any of ECI's instructional aids (Your Key to Career Development, Study Reference Guides, Chapter Review Exercises, Volume Review Exercise, and Course Examination), consult your education officer, training officer, or NCO, as appropriate. If he can't answer your questions, send them to ECI, Gunter AFB, Alabama 36118, preferably on ECI Form 17, Student Request for Assistance.

This volume is valued at 24 hours (8 points).

Material in this volume is technically accurate, adequate, and current as of August 1971.

Contents

		Page
	Preface	*iii*
Chapter		
1	INTRODUCTION TO THE MILLING MACHINE	1
2	MILLING MACHINE OPERATIONS	25
3	GRINDING MACHINE WORK	51
4	FITTING AND ASSEMBLY	75
5	MACHINE TOOL MAINTENANCE	91
	Bibliography	97

Introduction to the Milling Machine

MANY YEARS AGO a machinist placed a mandrel, upon which he had mounted a cutter, between the centers of a lathe. He then fastened the object he was machining to the lathe cross slide. From this simple idea the milling machine was invented. It developed into one of the most versatile and important metal cutting machines.

2. Milling is the operation of removing metal by means of revolving multitooth cutting tools called milling cutters. The milling machine is used chiefly for milling flat, angular, or irregular surfaces. With the aid of attachments, the milling machine is used for boring, keyseat and gear cutting, slotting, the cutting of flutes and grooves, and many other operations. In this chapter we will discuss milling machine construction and operation, cutters and arbors, and indexing.

1. Construction and Operation

1-1. You must be familiar with the main parts of the milling machine and their functions and controls before you can understand how it operates. You must know about the types and sizes of milling machines and the effects of speed and feed. We will begin with a discussion of the types and sizes of milling machines.

1-2. **Types.** Air Force machine shops have either one or two types of milling machines. These are the ram type and the column and knee type. Your shop may have one or both of these types. Each type may have either a plain or universal table. The main difference between a universal table and a plain table is that the universal table can be swiveled in the horizontal plane. The plain table cannot be swiveled.

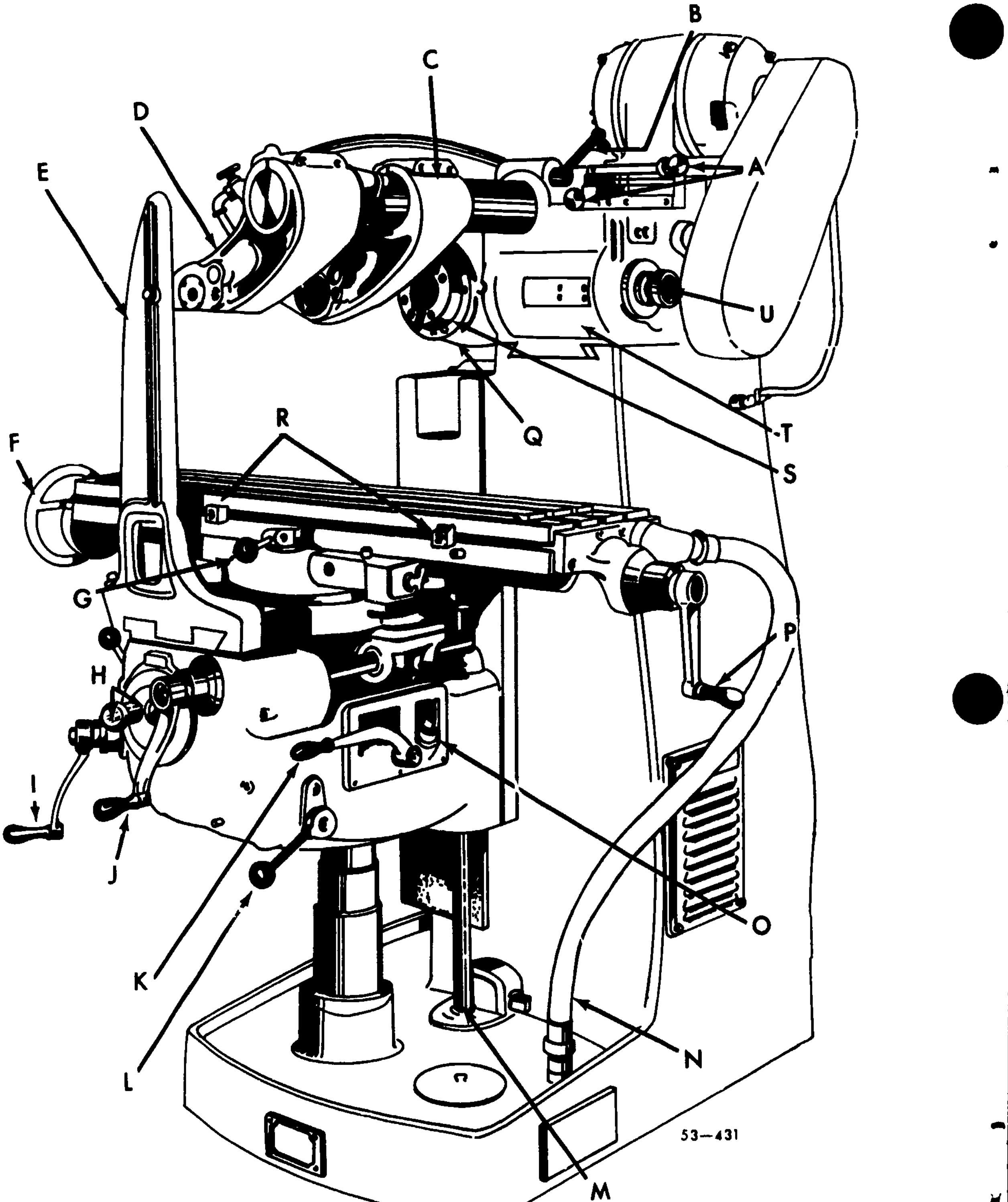
1-3. *Ram type milling machine.* As shown in figure 1, this machine is constructed with the cutter head, which contains the spindle, attached to the ram. The ram can be extended or retracted to any position over the table as required. The cutter head can be swiveled to any angle from horizontal to vertical. These two features permit you to machine vertical, horizontal, and angular surfaces with a minimum number of work setups. Moving the work less saves time and reduces the possibility of errors and inaccurate machining.

1-4. You will have no difficulty operating ram type milling machines if you are able to operate the more common column and knee type. Both types are basically the same. If you require more detailed information about this type of milling machine, you can obtain it from reference sources such as the 34 series technical orders and operator manuals.

1-5. *Column and knee.* The *horizontal* column and knee milling machine, as shown in figure 2, is the most common type of milling machine in AF shops. They are known as the column and knee type because the casting that the saddle and table are mounted on is called the knee. This knee moves vertically on the dovetail ways of the column. This type can have the spindle mounted either horizontally or vertically, as shown in figure 3. This does not affect the operation of the controls because they are so similar.

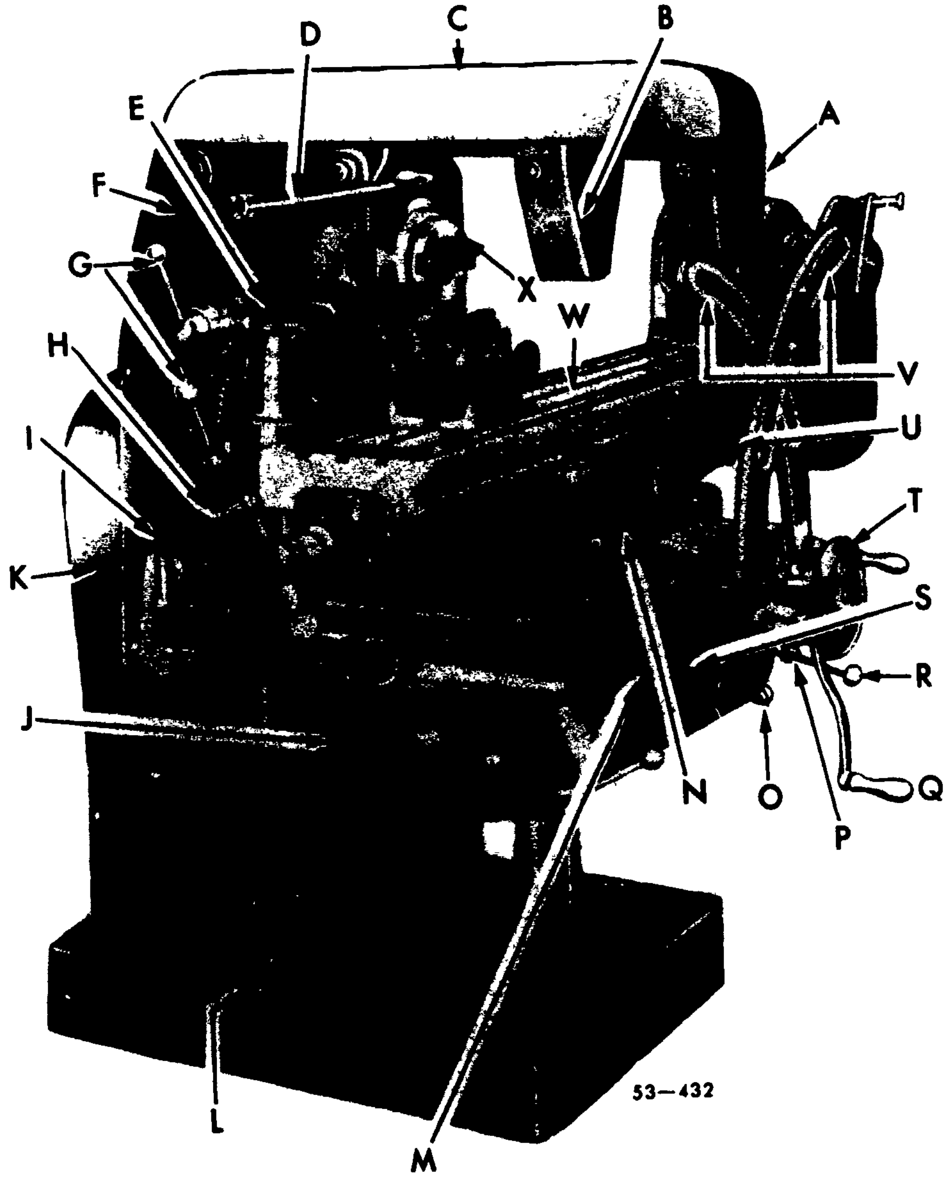
1-6. The *vertical* column and knee milling machine, as shown in figure 4, is not as common as the horizontal column and knee milling machine in the Air Force. Its main advantage is that you can observe cuts being made by end mills and face milling cutters. This is possible because all cutting is done horizontally, on the top of the work. It is not done behind the work, as is the case when an end mill or face milling cutter is used in a horizontal spindle machine.

1-7. **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. 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.



A. SPINDLE SPEED CHANGE LEVERS
B. OVERARM CLAMP
C. SLEEVE TYPE ARBOR SUPPORT
D. PILOT TYPE ARBOR SUPPORT
E. OUTER BRACE
F. TABLE HANDWHEEL
G. TABLE FEED LEVER
H. FEED SELECTOR
I. VERTICAL FEED HANDCRANK
J. CROSSFEED HANDCRANK
K. FRONT RAPID TRAVERSE LEVER
L. VERTICAL FEED LEVER
M. POWER FEED DRIVE SHAFT
N. COOLANT RETURN HOSE
O. KNEE OIL FILLER PLUG
P. TABLE HANDCRANK
Q. CUTTER HEAD
R. ADJUSTABLE TABLE STOPS
S. SPINDLE
T. RAM
U. SPINDLE HANDWHEEL

Figure 1. Ram type milling machine.



A. STYLE B ARBOR SUPPORT
B. STYLE A ARBOR SUPPORT
C. OVERARM
D. SPINDLE CLUTCH LEVER
E. GEAR SHIFTER BRACKET
F. COLUMN
G. SPEED LEVER CONTROL
H. LONGITUDINAL HANDCRANK
I. SPINDLE REVERSE LEVER
J. FEED REVERSE LEVER
K. MOTOR BEHIND THIS COVER
L. CENTRIFUGAL PUMP
M. FEED LEVER CONTROL
N. SADDLE
O. VERTICAL FEED LEVER
P. TRANSVERSE FEED LEVER
Q. KNEE ELEVATING CRANK
R. RAPID TRANSVERSE LEVER
S. KNEE
T. TRANSVERSE HANDWHEEL
U. LONGITUDINAL FEED CONTROL LEVER
V. OVERARM BRACES
W. TABLE
X. SPINDLE

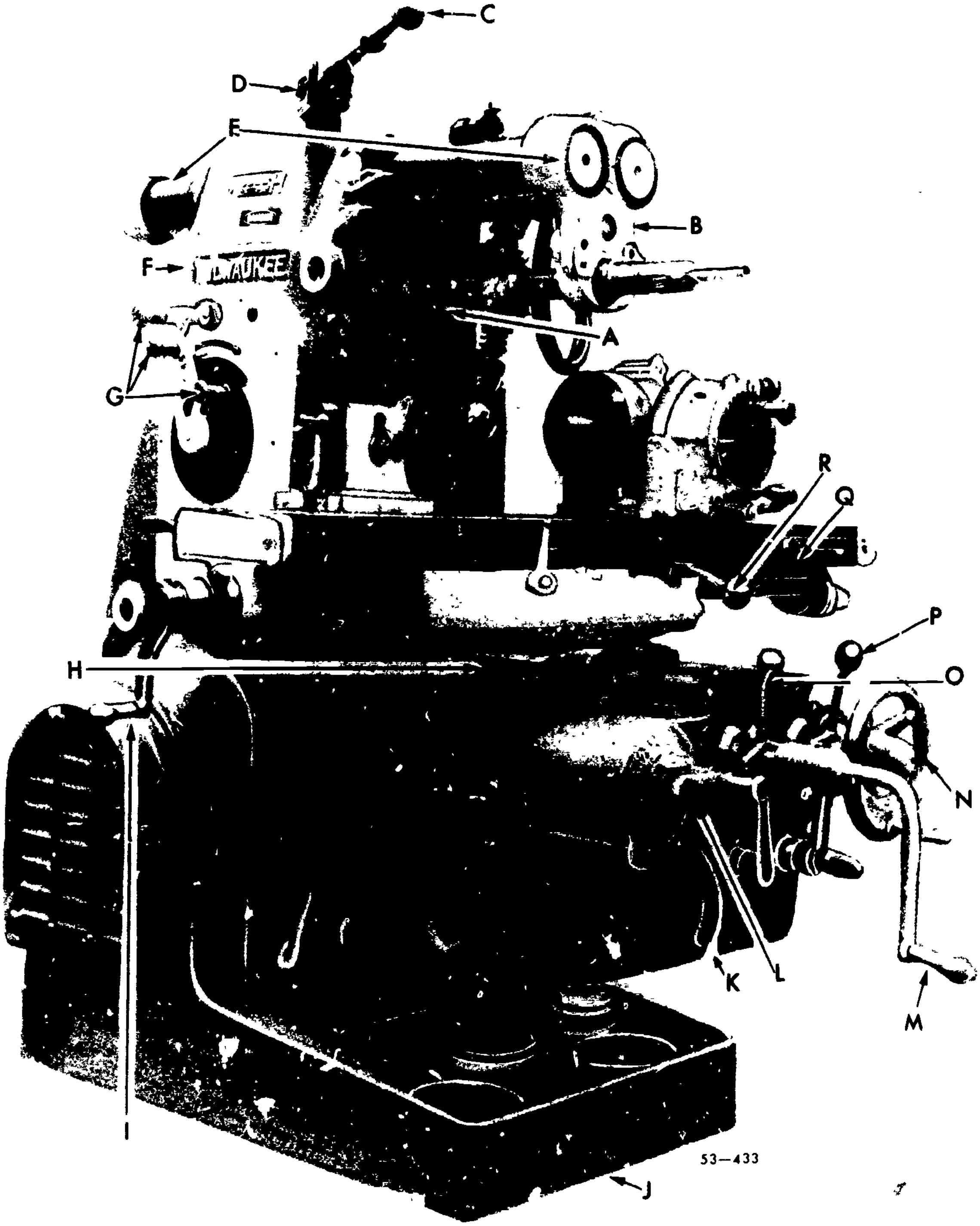
Figure 2. Plain column and knee milling machine with horizontal spindle.

Standard Size	*Longitudinal Table Travel*
No. 1	22 inches
No. 2	28 inches
No. 3	34 inches
No. 4	42 inches
No. 5	50 inches
No. 6	60 inches

If the milling machine in your shop is labeled No. 2HL, it has a table travel of 28 inches, and if it is labeled No. 5LD, it has a travel of 50 inches.

1-8. **Main Parts.** Since the column and knee milling machine is able (if you use the proper attachments) to perform the same operations as the vertical spindle and ram types, we will discuss it in detail. We begin with a description of the main parts.

1-9. You must know the name and purpose of each of the main parts of a milling machine in order to understand the operations discussed later in the volume. Keep in mind that, although we are



A. SPINDLE
B. ARBOR SUPPORT
C. SPINDLE CLUTCH LEVER
D. SWITCH
E. OVERARM
F. COLUMN
G. SPINDLE SPEED SELECTOR LEVERS
H. SADDLE AND SWIVEL
I. LC' ~'TUDINAL HANDCRANK
J. BA~.
K. KNEE
L. FEED DIAL
M. KNEE ELEVATING CRANK
N. TRANSVERSE HANDWHEEL
O. VERTICAL FEED CONTROL
P. TRANSVERSE FEED LEVER
Q. TABLE FEED TRIP DOG
R. LONGITUDINAL FEED CONTROL

Figure 3. Universal column and knee milling machine with horizontal spindle.

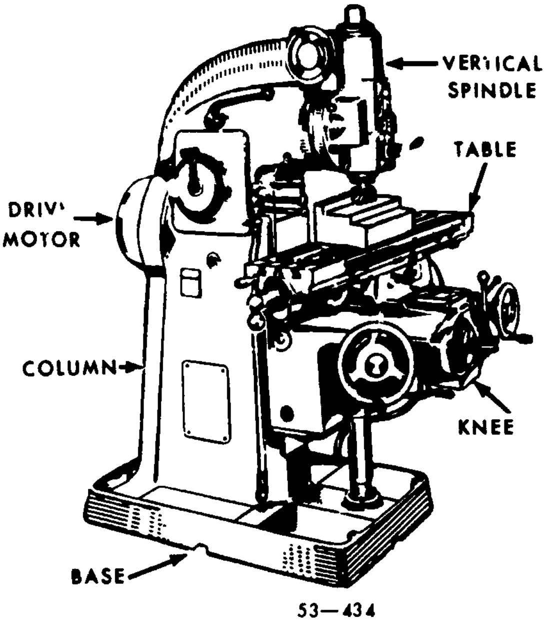


Figure 4. Vertical spindle milling machine.

discussing a column and knee milling machine, you can apply most of the information to the other types. Figure 2, which illustrates a plain column and knee milling machine, and figure 3, which illustrates a universal column and knee milling machine, will help you to become familiar with the location of the parts.

1-10. *Column.* The column, which is the main casting of a milling machine, is called a column because of its height and shape. The column contains the gearing and drive shafts. An oil reservoir and a pump in the column supply the spindle with the necessary lubrication. 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.

1-11. *Knee.* The knee is the casting that supports the table and saddle. It acquired its name because it resembles the knee used in building construction to reinforce joints. 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.

1-12. *Power feed mechanism.* The power feed mechanism, which is contained in the knee, controls the longitudinal, transverse (in and out) and vertical feeds. You can obain the desired rate of feed on machines, such as the one shown in figure 2, by positioning the feed selection levers as indicated on the feed selection plate. On machines such as the one in figure 3 you obtain the feed that 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 wish to temporarily increase the speed of the longitudinal, transverse, or vertical feeds. For example, you would engage this lever when you are positioning or alining the work.

1-13. *Table.* The table is the rectangular casting located on top of the saddle. It contains several T-slots that enable you to fasten work or workholding devices to it. You can move the table by hand or by power. To move the table by hand, you engage and turn the longitudinal handcrank. To move it by power, you 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. You place the ball end of the directional feed control lever to the left to feed the table toward the left. You place it to the right to feed the table toward the right. You place it in the center position to disengage the power feed or to feed the table by hand.

1-14. *Spindle.* The spindle on a milling machine is somewhat similar in purpose to a lathe spindle. A milling machine spindle normally holds and drives a cutter or a cutting toolholder. A lathe spindle normally holds and drives a workholding device. The spindle extends completely through the column of the milling machine. The front end of the spindle, which is near the table, has an internal taper machined in it. The internal taper per-

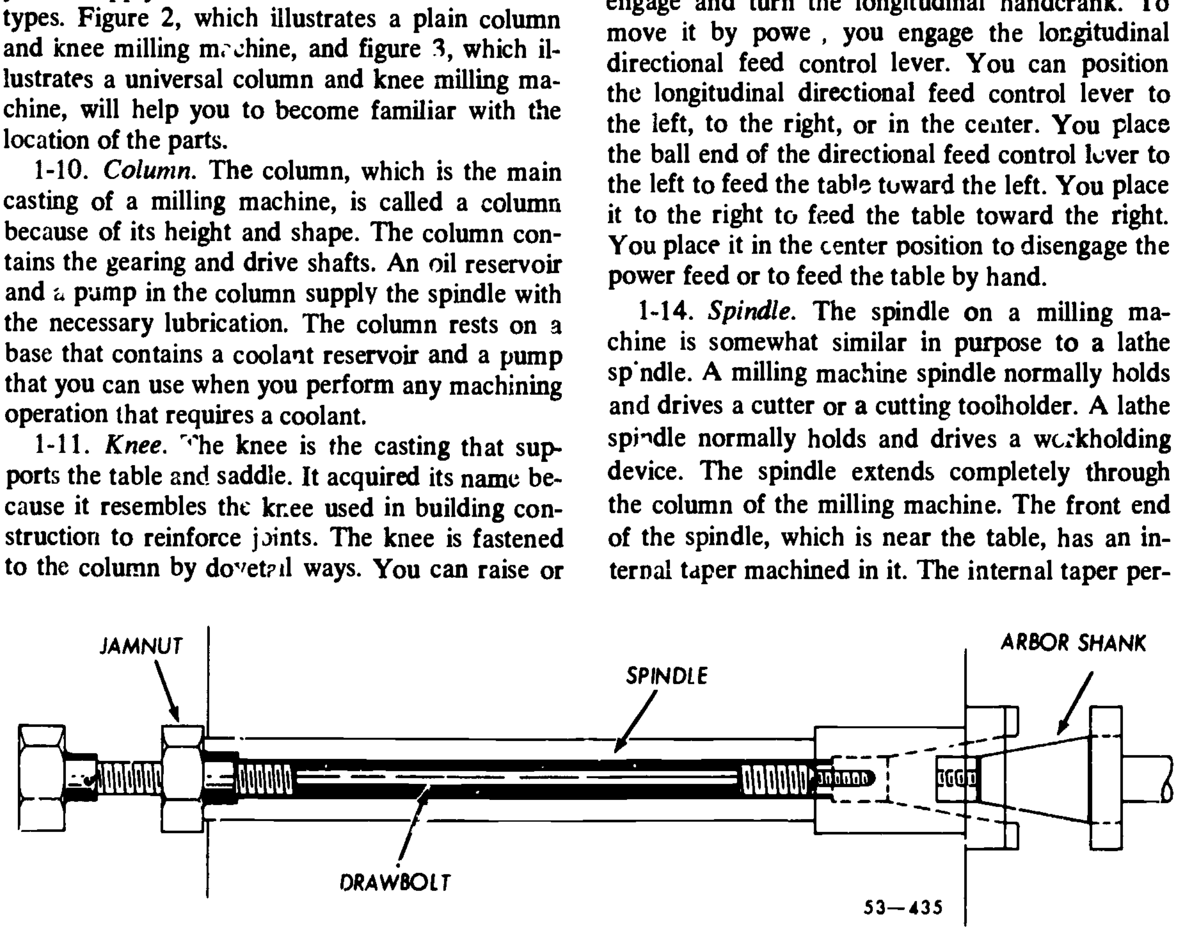


Figure 5 Spindle drawbolt.

mits 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 5. Large face mills are sometimes mounted directly to the spindle nose.

1-15. *Overarm.* The overarm is the horizontal beam to which you fasten the arbor support. The overarm, shown in figure 3, may be a single casting that slides in dovetail ways on the top of the column. It may consist of one or two cylindrical bars that slide through holes in the column, as shown in figure 4. You position the overarm on some machines by first unclamping locknuts and then extending 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 over the arbor bearing in order to make the setup as rigid as possible. You can place arbor supports, which consist of two arms, on an overarm such as the one shown in figure 3 if you extend one of the bars approximately 1 inch farther than the other bar. Tighten the locknuts after you have positionc the overarm. On some milling machines the coolant supply nozzle is fastened to the overarm. A split clamp permits you to mount the nozzle to the overarm after you have placed the arbor support in position.

1-16. *Arbor support.* The arbor support is a casting that contains a bearing which alines 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. Type A has a small diameter bearing hole. Type B has a large diameter bearing hole. An oil reservoir in the arbor support supplies the bearing surfaces with the necessary lubrication. You can clamp an arbor support at any desired location on the overarm. Type A arbor supports provide additional clearance below the arbor supports when you are using small diameter cutters. Type A arbor supports can only provide support at the extreme end of the arbor. For this reason they are not recommended for general use. You can position type B arbor supports at any point on the arbor. Therefore, they can provide support near the cutter, if necessary. For this reason, you should position the type B arbor support as close to the cutter as possible in order to provide a rigid tooling setup.

NOTE: Before loosening or tightening the arbor nut, the arbor support *must* be installed. This will prevent bending or springing the arbor.

1-17. **Speed and Feed.** Which spindle speed should you use? What should the rate of feed be? These are questions that you will ask yourself as you make work and tool setups on a milling machine. Unfortunately, there are no simple answers to these questions. Every job presents a new set of variables that must be taken into consideration. You will have to decide which speed and feed to use at the beginning of the machining operation. After observing the cutting action, make any changes that you feel are necessary.

1-18. *Speed.* As was explained in drill press work (Volume 1), cutting speed is always given in feet per minute. You must convert cutting foot speed to spindle or cutter speed (rpm). When you use attachments that change the ratio between the rpm of the cutter and the spindle, first determine the cutter speed and then determine the spindle speed. You must consider this ratio and increase or decrease spindle rpm to give the required cutter rpm. You will remember from our discussion of drill press speed in Volume 1 that a point on the periphery of the cutting tool should travel at a surface foot speed that is as near as possible to the cutting foot speed recommended for the material you are machining. Also, the diameter of the drill (in this case the cutter) must be included when you calculate the spindle speed. You can use the same formula to select the milling cutter speed that you use to select the drill speed if you substitute the diameter of the cutter for the diameter of the drill. The formula would then be as follows:

$$\text{rpm} = \frac{4 \times \text{CFS}}{\text{cutter diameter}}$$

Use the lower CFS given in chart 1 when you are rough machining and the higher CFS when you are finish machining.

1-19. For example: What spindle speed would you use to rough mill low carbon steel with a high-speed steel milling cutter 3 inches in diameter? Chart 1 shows that the recommended CFS for rough milling low carbon steel is 80. Therefore:

$$\text{rpm} = \frac{4 \times 80}{3}$$

$$\text{rpm} = \frac{320}{3}$$

$$\text{rpm} = 107$$

To rough mill low carbon steel with a 3-inch cutter, set the spindle speed controls to obtain a speed as near as possible to 107 rpm.

1-20. *Feed.* Feed is the rate in inches per minute at which the work is moved into the revolving cutter. The thickness of the chip removed by each cutter tooth as it contacts the work is the basis for determining the feed. Figure 6 shows the chip thickness for both conventional and down milling.

1-21. Climb milling is feeding the work in the same direction as that of the cutter rotation. The

Chart 1
Cutting Foot Speeds

MATERIAL	CUTTING FOOT SPEED
LOW CARBON STEEL	80 TO 110
MEDIUM CARBON STEEL	60 TO 80
HIGH-CARBON TOOL STEEL	50 TO 60
STEEL FORGINGS	50 TO 60
STAINLESS STEEL	30 TO 40
SOFT CAST IRON	100 TO 150
HARD DRILLED CAST IRON	70 TO 100
MALLEABLE IRON	80 TO 90
ORDINARY BRASS AND BRONZE	200 TO 300
HIGH-TENSILE BRONZE	70 TO 150
MONEL	40 TO 150
ALUMINUM AND ITS ALLOYS	200 TO 300
MAGNESIUM AND ITS ALLOYS	250 TO 400
BAKELITE	100 TO 150
WOOD	300 TO 400

NOTE: CARBON STEEL DRILLS SHOULD BE RUN AT SPEEDS OF FROM 40 TO 50 PERCENT SLOWER THAN THOSE GIVEN ABOVE.

53—436

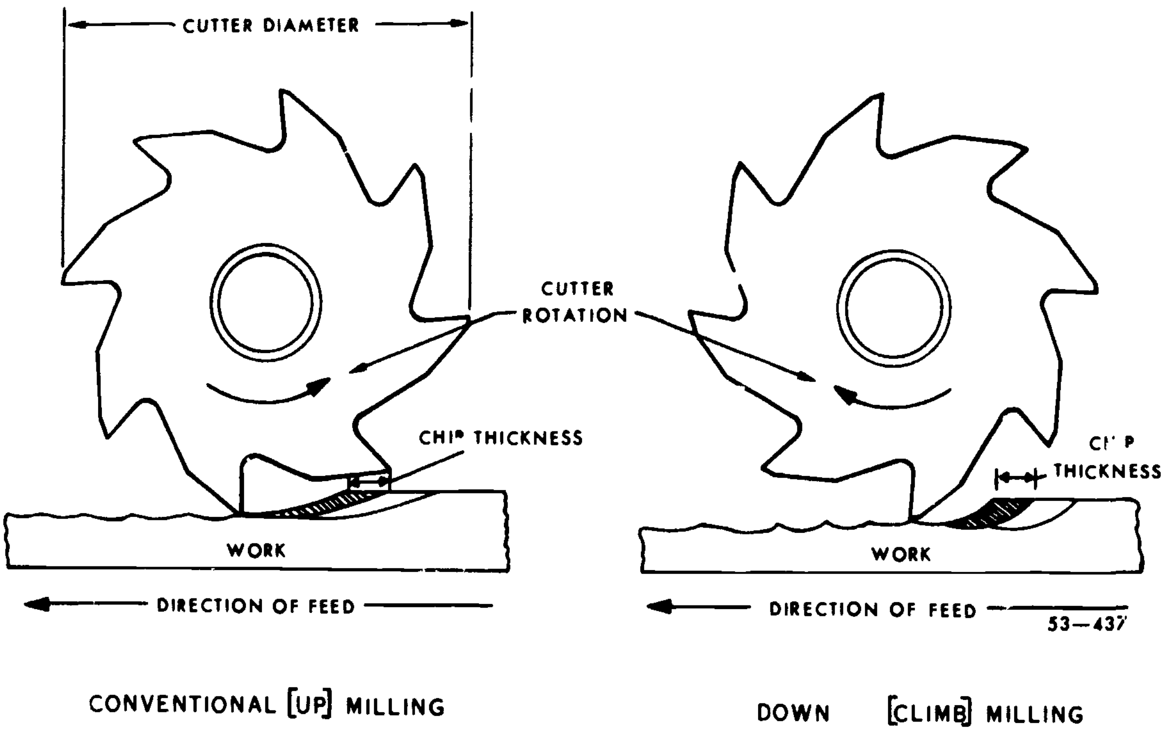


Figure 6. Chip thickness.

thrust of the cut is DOWN or against the work. You use climb milling when the work tends to spring or lift, as in milling thin or easily distorted work. During climb milling the cutter tends to climb or pull the work into the cutter. This may cause damage to the work or cutter. To prevent this, make sure that the gibs are snug and that excessive end play (backlash) in feed screws is removed. You should also tighten the table lock until a slight drag is felt when you use the hand feed. Conventional, or up, milling is feeding the work in a direction opposite that of the cutter rotation. The thrust of the cut is UP, or away from the work. Use conventional milling whenever the nature of the work and the holding device will permit.

1-22. The thickness of the chip, or as it is often called, the feed per tooth, is normally from 0.001 inch to 0.015 inch, but it can be more. Use fine feeds for light finishing cuts and coarse feeds for heavy roughing cuts.

1-23. You can determine the chip thickness by multiplying the number of teeth on the cutter by the cutter rpm and then dividing the feed per minute by the product. Determine the chip thickness by using the following formula if you know the feed per minute, the rpm, and the number of teeth on the cutter:

$$\text{Thickness} = \frac{\text{feed}}{\text{teeth} \times \text{rpm}}$$

Example: You are using a cutter that has 20 teeth. The rpm is 40 and the feed is 1 inch per minute. What is the chip thickness?

$$\text{Thickness} = \frac{1}{20 \times 40}$$

$$\text{Thickness} = \frac{1}{800}$$

$$\text{Thickness} = 0.00125 \text{ inch}$$

1-24. The rate of feed is usually expressed as inches of feed per minute. Most milling machine feed dials are calibrated in inches per minute. You must convert the chip thickness into inches per minute to set the machine for the feed you select. You can do this by using the formula:

$$\text{Feed} = \text{chip thickness} \times \text{number of teeth} \times \text{rpm}$$

Example: You have determined that a speed of 80 rpm and a chip thickness of 0.004 inch should be used to mill the job you are working on. The cutter you have selected has 12 teeth. What feed would you set the machine for?

$$\text{Feed} = 0.004 \times 12 \times 80$$
$$\text{Feed} = 0.048 \times 80$$
$$\text{Feed} = 3.840 \text{ inches}$$

You would set the machine to produce a feed as near as possible to 3.840 inches.

1-25. Varying conditions in milling machine work make it impractical to have fixed rules for cutting speeds and feeds. Generally, you should select a cutting speed that gives efficient cutting action without undue wear on the cutter. Several factors have to be considered in selecting cutter speeds and feeds.

a. Hardness of the material. The harder and tougher the metal, the slower the cutting speed.

b. Depth of cut and finish desired. Light finish cuts are made at higher speeds than heavy roughing cuts.

c. Type of cutter material. High-speed steel cutters can be run at higher speeds than high carbon cutters, while carbide cutters can be run faster than high-speed steel ones.

d. Type of milling operation. Form milling operations using form cutters require slower speeds than some of the plain or face milling operations.

e. Sharpness of the cutter. A sharp cutter always cuts better and easier than a dull one.

f. Coolant. All milling cutters can be operated at faster speeds and feeds when a coolant is used.

2. Cutters and Arbors

2-1. To perform a milling operation you move the work into a rotating cutter. The spindle may drive the cutter directly. More often the spindle drives an arbor on which the cutter is mounted. We will discuss cutters in the first part of this section and arbors in the second part.

2-2. **Cutters.** There are many different milling machine cutters. Some cutters can be used for several operations and others for only one operation. Some cutters have straight teeth and others have helical teeth. Some cutters have mounting shanks and others have mounting holes. You must decide which cutter to use. To make this decision, you must be familiar with the various milling cutters and their uses. The information in this section will help you to select the proper cutter for the various operations you will be performing. In this section we will cover cutter classification, cutter types, and cutter selection.

2-3. *Classification.* Milling cutters may be classified according to the type of relief on the cutting edge (profile or form) and by the method used to mount the cutter (arbor or shank).

2-4. Profile-relieved cutters are often referred to simply as profile cutters and form-relieved cutters as formed cutters. Profile cutters are sharpened by grinding teeth on the circumference of the cutter. You provide clearance by grinding behind the cutting edge, as shown in figure 7. Profile cutters are used more often than formed cutters. Formed cutters are used to mill shapes that you

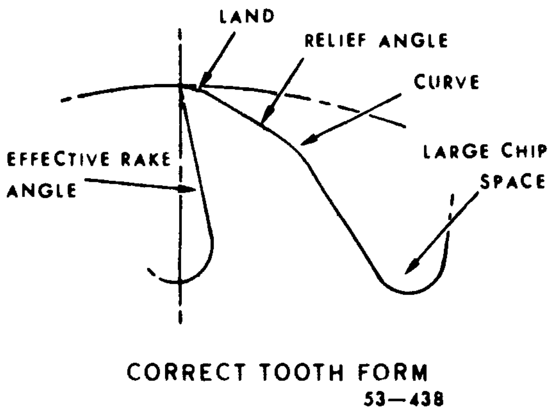


Figure 7. Profile of a cutter tooth.

cannot obtain with profile cutters. Formed cutters have an eccentric relief. All of the material indicated by the dark portion in figure 8 is ground away, and the relief behind the cutting edge has the same contour as the cutting edge. You usually sharpen formed cutters by grinding the face of the tooth parallel to the axis of the cutter, as shown in figure 8. This permits them to retain their shape indefinitely. You sharpen some formed cutters (usually those that have a simple, easily reproduced shape, such as a radius) by grinding the curved profile of each tooth. Formed cutters that are sharpened by grinding the face of the tooth are known as face-ground formed cutters. Formed cutters that are sharpened by grinding the profile of the tooth are called profile-ground formed cutters.

2-5. Milling cutters may also be classified as arbor-mounted or shank-mounted. Arbor-mounted cutters are mounted on the straight shanks of arbors, and then the arbor is inserted into the milling machine spindle. We will discuss the methods of mounting arbors and cutters in greater detail later in this section.

2-6. 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.

2-7. *Types and uses.* There are many different types of milling cutters. We will now discuss these types and their uses.

2-8. You will use *plain milling cutters* to mill flat surfaces that are parallel to the cutter axis. As you can see in figure 9, a plain milling cutter is a cylinder with teeth cut on the circumference only. Plain milling cutters are made in a variety of diameters and widths. Note in figure 9 that the cutter teeth may be either straight or helical. When the width is more than ¾ inch, the teeth are usually helical. A straight cutter tooth is parallel to its axis. It cuts along its entire width at the same time, causing a shock as the tooth starts to cut. Helical teeth eliminate this shock and produce a free cutting action. A helical tooth begins the cut at one end and continues across the work with a smooth shaving action. Plain milling cutters usually have radial teeth. Some coarse helical tooth cutters have the tooth face undercut to produce a smoother cutting action. Coarse teeth decrease the tendency of the arbor to spring and give the cutter greater strength. The helical teeth of some heavy-duty plain milling cutters are nicked, as shown in figure 10. The nicks are staggered so that the cutting edge is behind each nick. The nicked tooth does not cut one continuous chip, but breaks up the chip into a number of separate pieces. Each cutting edge produces a chip. A plain milling cutter has a standard size arbor hole to enable it to be mounted on a standard size arbor. The size of the cutter is designated by the diameter of the cutter, width of the cutter, and diameter of the hole.

2-9. The *side milling cutter* is a plain milling cutter with teeth cut on both sides, as well as on

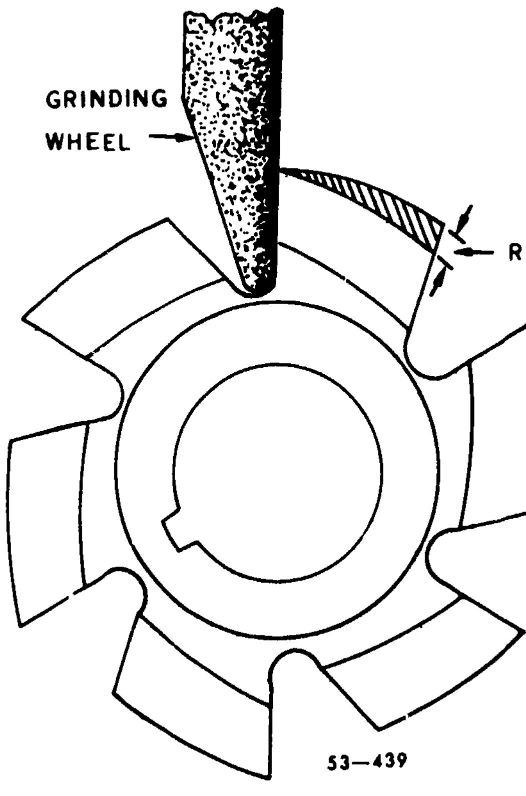


Figure 8. Sharpening formed cutters.

53—440

Figure 9. Plain milling cutters.

the circumference of the cutter. You can see in figure 11 that the portion of the cutter between the hub and the side of the teeth is thinner to provide additional chip clearance. These cutters are often used in pairs to mill parallel sides. This process is called straddle milling. Cutters over 8 inches in diameter are usually made with inserted teeth. The size designation is the same as for plain milling cutters.

2-10. *Half-side milling cutters,* shown in figure 12, 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.

2-11. *Side milling cutters* whose teeth interlock (fig. 13) can be used to mill standard size slots. The width is regulated by inserting thin washers between the cutters.

2-12. You can use a *metal slitting saw* to cut off work or to mill narrow slots. A metal slitting saw is similar to a plain or side milling cutter. The face width is usually less than $\frac{3}{16}$ 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 provide proper clearance for milling deep slots. Figure 14 shows a metal slitting saw with teeth cut in the circumference of the cutter only. Some saws, such as the one in figure 15, 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 16. These cutters are usually $\frac{3}{16}$ inch to $\frac{3}{8}$ inch in thickness.

2-13. You will use the *screw slotting cutter* (fig. 17) 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 gage wire numbers.

2-14. You will use *angle cutters* to mill surfaces that are not at a right angle to the cutter axis. You can use angle cutters for a variety of work, such as milling reamer flutes and dovetail ways. On work such as dovetailing, where you cannot mount a cutter in the usual manner on an arbor, you can mount an angle cutter that has a threaded hole or is constructed like a shell end mill on the end of a stub or shell end mill arbor. When you select an angle cutter for a job you should specify type, hand, outside diameter, thickness, hole size, and angle.

2-15. There are two types of angle cutters: single and double. The *single angle* cutter, shown in figure 18, has teeth cut at an oblique angle with the one side at an angle of 90° to the cutter axis and the other usually at 45°, or 50° or 80°. The

53—441

Figure 10. Nicked tooth cutter.

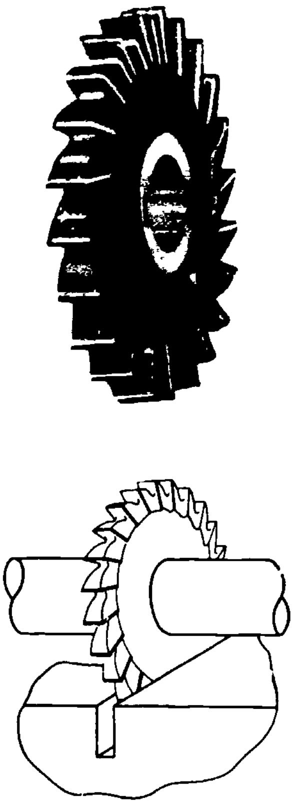


Figure 11. Side milling cutter.

double angle cutter, figure 19, 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 at different angles, you specify the angle of each side with respect to the plane of intersection.

2-16. 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. Flv :ng cutters are marked with the range of diameters they are designed to mill.

2-17. You will use *end mill cutters* to mill slots, tangs, and the ends and edges of work. They have teeth cut ‿ .he end as well as on the circumference of the cutter. The cutters may be solid with two or more teeth. They may be the shell type. Figure 20 shows a two-lipped end mill. This cutter is especially adapted for milling slots without first drilling a hole. It should be operated at high speed. Figure 21 shows a center cutout end mill. You can use this cutter to mill work to a depth of cut equal to the length of the end teeth. Shell end mills, shown in figure 22, bridge the gap between solid end mills and face milling cutters. Shell end mills are attached at the end of a taper shank arbor. In most cases they are more economical than large solid cutters because they are cheaper to replace when they break or wear out.

2-18. Inserted tooth *face milling cutters,* shown in figure 23, are similar to shell end mills in that they have teeth on the circumference and on the end. They are attached directly to the spindle nose and use inserted teeth made of carbide or an alloy steel. The teeth are replaceable.

2-19. The *T-slot cutter,* figure 24, 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 slc* with a T-slot cutter.

2-20. A *Woodruff keyseat cutter,* figure 25, is used to cut curved keyseats. A cutter less than 1½ inches in diameter has a shank. When the diameter is greater than 1½ inches, the cutter is usually mounted on an arbor. The larger cutters have staggered teeth to improve the cutting action.

2-21. There are several types of *gear cutters* such as bevel, spur, involute, etc. Figure 26 shows an involute gear cutter. You must select the correct type of cutter to cut a particular type of gear.

Figure 12. Half-side milling cutter.

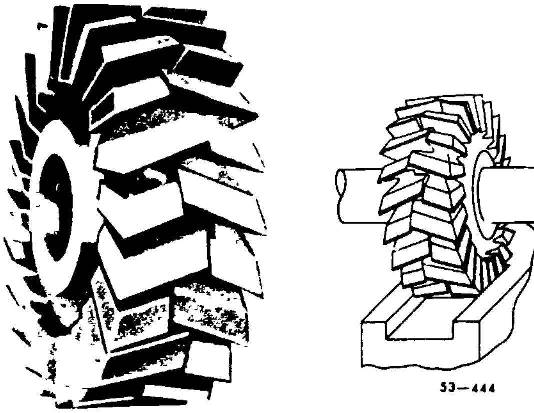

53—444

Figure 13. Interlocking teeth side milling cutter.

2-22. You use a *concave cutter,* figure 27, to mill a convex surface and a *convex cutter,* figure 28, to mill a concave surface.

2-23. *Corner rounding cutters,* figure 29, are formed cutters which are used to round corners up to one-quarter of a circle.

2-24. The *sprocket wheel cutter,* figure 30, is a formed cutter that is used to mill teeth on sprocket wheels.

2-25. The *gear hob,* figure 31, is a formed milling cutter with teeth cut like threads on a screw. You can use it for finishing spur gears, helical gears, worm wheels, etc., and for cutting ratchets and splined shafts.

2-26. The *fly cutter,* as shown in figure 32, is a cutter that is often manufactured locally. It is a single-point cutting tool similar in shape to a lathe or shaper tool. It is held and rotated by a fly cutter arbor. You can grind this cutter to almost any form that you desire. There will be times when you will need a special formed cutter for a very limited number of cutting or boring operations.

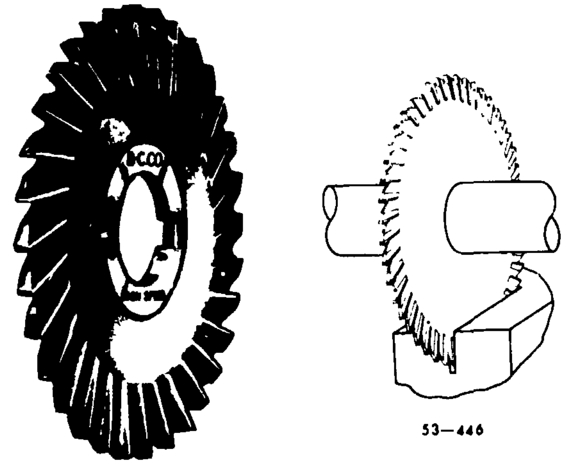

53—446

Figure 15. Slitting saw with side teeth.

2-27. 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 order. We will now discuss the selection of cutters.

2-28. *Selection.* Each cutter can do one kind of job better than any other in a given situation. A cutter may not be limited to a specific milling operation, although this is true in some cases. In selecting the most suitable cutter for a particular milling operation, you must consider the kind of cut to be made, material to be cut, number of parts to be machined, and the type of milling machine available.

2-29. Another factor that affects a milling operation is the number of teeth in the cutter. If the number is too great, the space between the teeth is so small that it prevents the free flow of chips. The

53—445

Figure 14. Metal slitting saw.

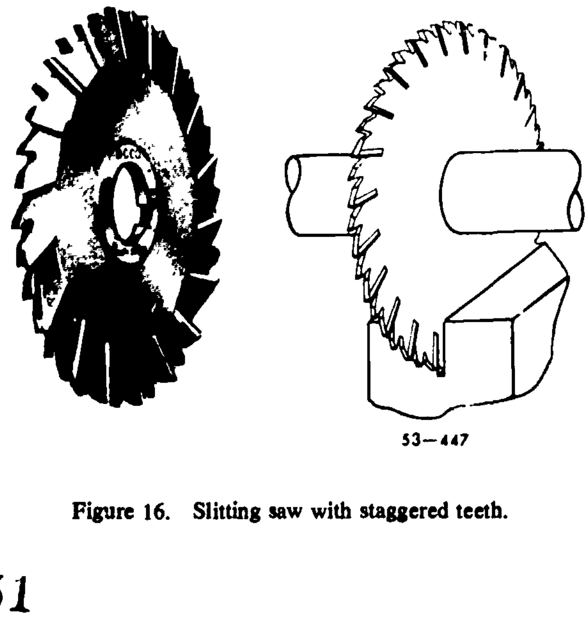

53—447

Figure 16. Slitting saw with staggered teeth.

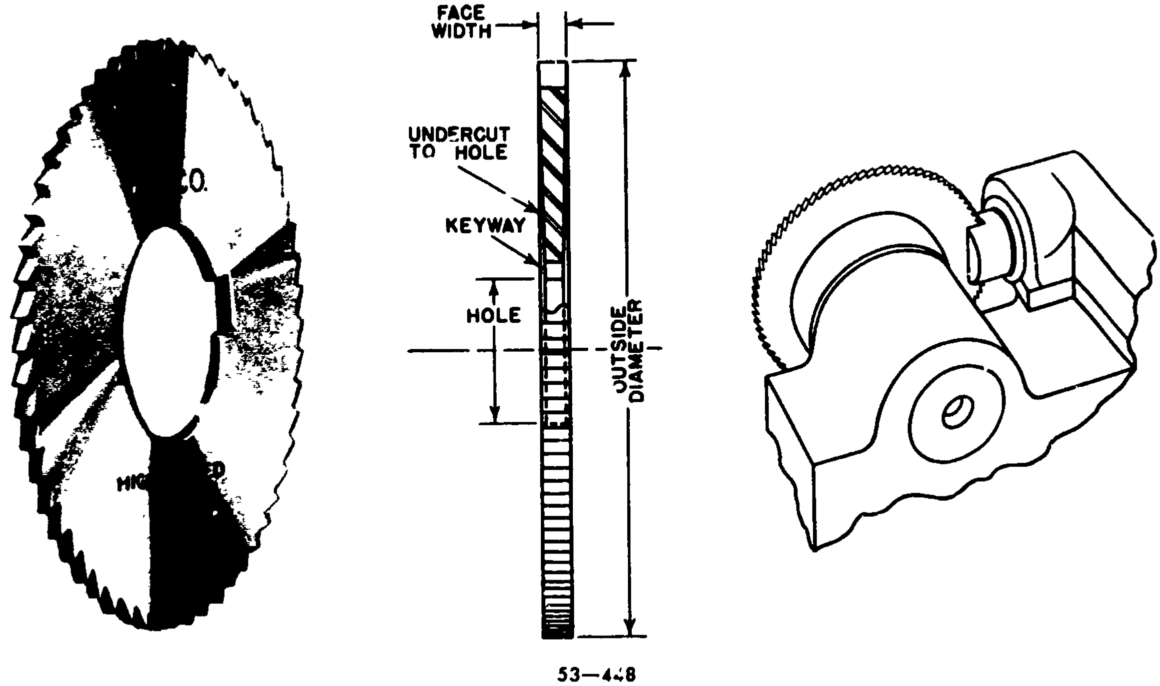


Figure 17. Screw slotting cutter.

chip space should also be smooth and free of sharp corners to prevent clogging of the chips in the space. A coarse-tooth cutter is more satisfactory for milling material that produces a continuous and curled chip. The coarse teeth not only permit an easier flow of chips and coolant but help to eliminate chatter. A fine-tooth cutter is more satisfactory for milling a thin material. It reduces cutter and workpiece vibration and the tendency for the cutter teeth to "straddle" the work and dig in. As a general rule *no more than two teeth at a time should be engaged in the cut.*

2-30. Still another factor that you should consider is the diameter of the cutter. Select the smallest diameter cutter which will allow the arbor to pass over the work without interference when the cut is taken. As illustrated in figure 33, a small cutter requires less time than a larger one to take a cut.

2-31. **Arbors.** You can mount milling machine cutters on several types of holding devices. You must know what the devices are and the purpose of each of them in order to make the most suitabl tooling setup for the operation you are performing. We will cover the various types of arbors and the mounting and dismounting of arbors in this section.

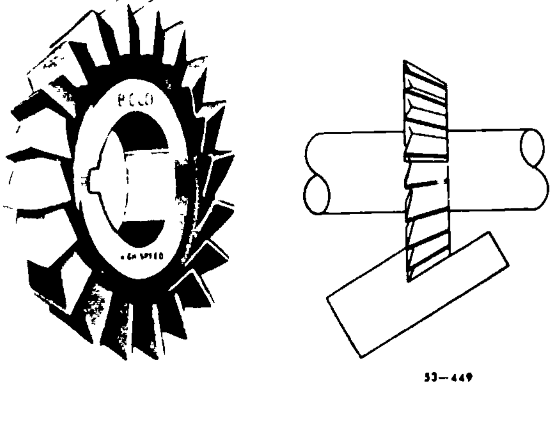


Figure 18. Single angle cutter.

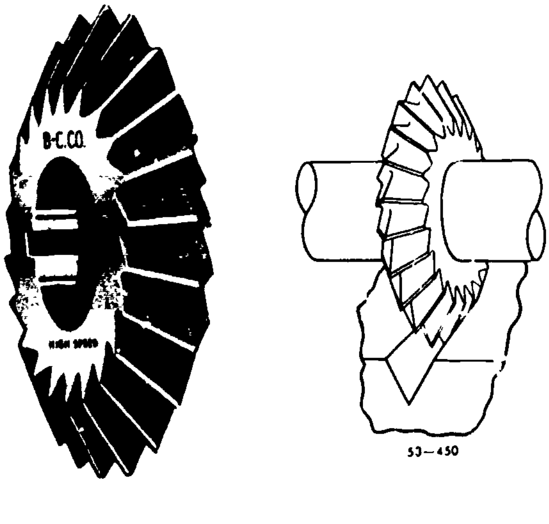


Figure 19. Double angle cutter.

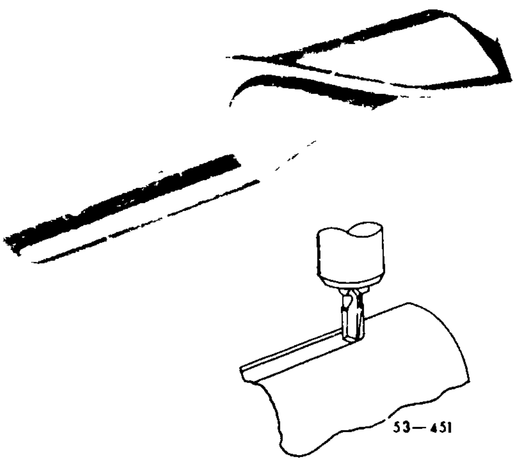


Figure 20. Two-lipped end mill.

NOTE: Technically, an arbor is a shaft on which a cutter is mounted. For convenience, since there are so few types of cutter holders that are not arbors, we will refer to all types of cutter holding devices as arbors.

2-32. *Types*. There are several types of milling machine arbors. You use the common or standard types, shown in figure 34, to hold and drive cutters that have mounting holes.

2-33. One of the arbor usually has a standard milling machine spindle taper of 3½ inches per foot. The largest diameter of the taper is identified by a number. For example, the large diameter of a number 40 milling machine spindle taper is 1¾ inches. The numbers representing common milling machine spindle tapers and their sizes are as follows:

Number	*Large Diameter*
10	⅝ inch
20	⅞ inch
30	1¼ inches
40	1¾ inches
50	2¾ inches
60	4¼ inches

2-34 *Standard arbors* are available in styles A and B, s shown in figure 34. Style A arbors have a pilot ʼpe bearing, usually $1\frac{1}{32}$ 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	1⅞ inches
4	2⅛ inches
5	2¾ inches

The inside diameter can be any one of several standard diameters that are used for he 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 ¼–A–18–4 is as follows:

5 = taper number - 50 (the 0 is omitted in the code)
1¼ = shaft diameter - 1¼ inches
A = style A bearing - pilot type
18 = usable shaft length - 18 inches
4 = bearing size - 2⅛ inches diameter

Arbors that have very short shafts, such as the one shown in figure 3ᶠ, are called stub arbors. Stub arbors are used when it would be impractical to use a longer arbor.

2-35. You will use arbor spacing collars of various lengths to position and secure the cutter on the arbor. The spacers are tightened 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. To prevent the cutter from slipping, you insert a square key into the keyway formed by the keyseat that extends the full length of the arbor shaft and the keyseat in the cutter.

2-36. *Shell end mill arbors,* shown in figure 36, 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 while it is held

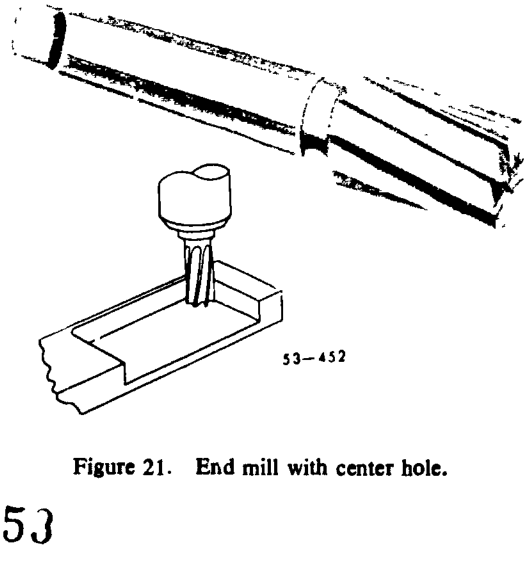


Figure 21. End mill with center hole.

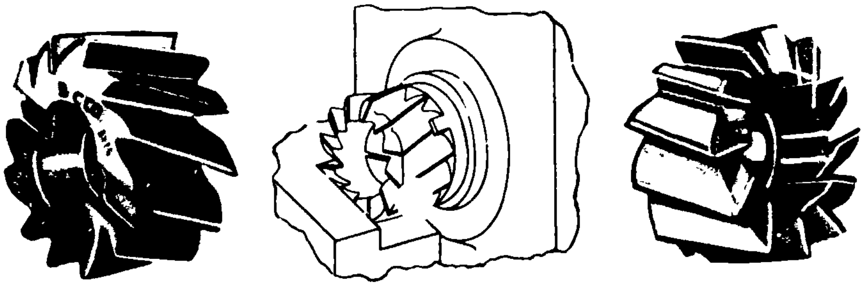


Figure 22. Shell end mills.

against the face of the arbor by a bolt. You use a special wrench, shown in figure 36, 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 ½–C–⅞ is as follows:

4 = taper code number - 40
1½ = diameter of mounting hole in end mill - 1½ inches
C = style C arbor - shell end mill
⅞ = length of shaft - ⅞ inch

2-37. *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, as shown in figure 32. Fly cutter arbor shanks may have a standard milling machine spindle taper, a Brown and Shape taper, or a Morse taper.

2-38. *Screw slotting cutter arbors* are used with screw slotting cutters. The flanges provide support for the cutter and prevent the cutter from flexing. The shanks on screw slotting cutter arbors may be straight or tapered, as shown in figure 37.

2-39. *Screw arbors,* shown in figure 38, are used with cutters that have threaded mounting holes. The threads may be left or right hand.

2-40. *Tape 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 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:

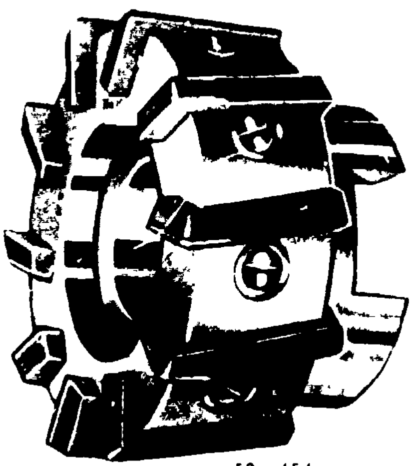


Figure 23. Inserted tooth face milling cutter.

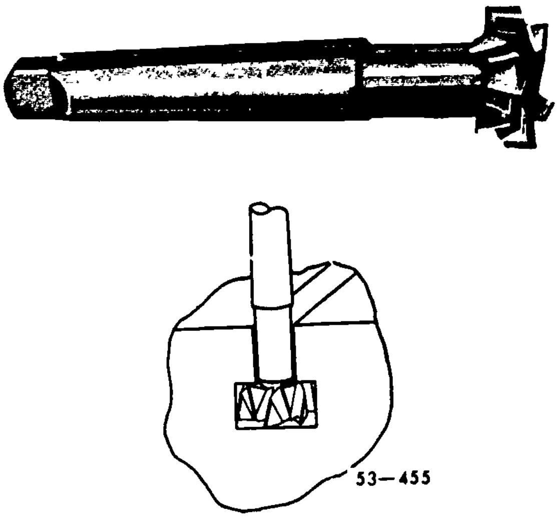


Figure 24. T-Slot cutter.

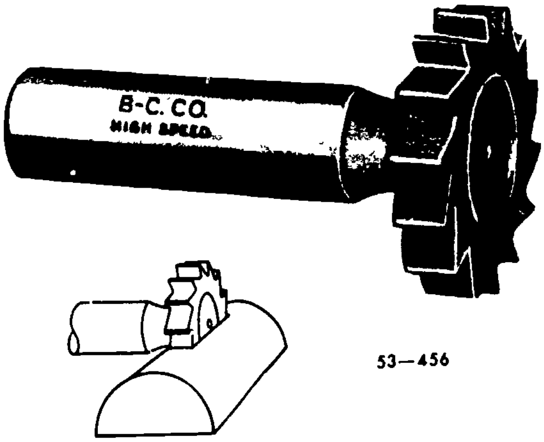


Figure 25. Woodruff keyseat cutter.

50 = taper identification number
10 = internal taper - number 10 B and S

Figure 39 shows a typical taper adapter. Some cutter adapters are designed to be used with tools that have taper shanks and a cam locking feature.

Figure 26. Involute gear cutter.

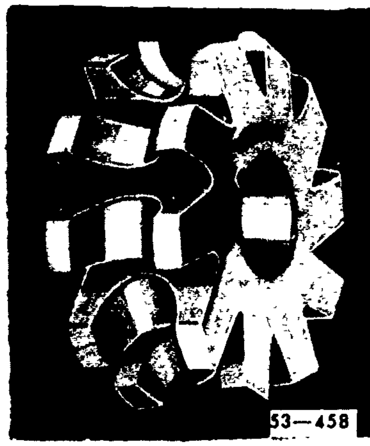


Figure 27. Concave cutter.

The cam lock adapter code indicates the number of the external taper, number of the internal taper (which is usually a standard milling machine spindle taper also), and the distance that the adapter extends from the spindle of the machine. For example, 50–20–3 ⅝ inches means:

50 = taper identification number (external)
20 = taper identification number (internal)
3⅝ = distance adapter extends from spindle is 3⅝ inches

2-41. *Cutter adapters,* such as the one shown in figure 40, 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, etc. The cutting tool is secured in the adapter by a setscrew. The code number indicates the number of the taper and the diameter of the hole. For example, 50–⅝ means that the adapter has a number 50 taper and a ⅝-inch-diameter hole.

2-42. *Spring collet chucks,* shown in figure 41, 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 col-

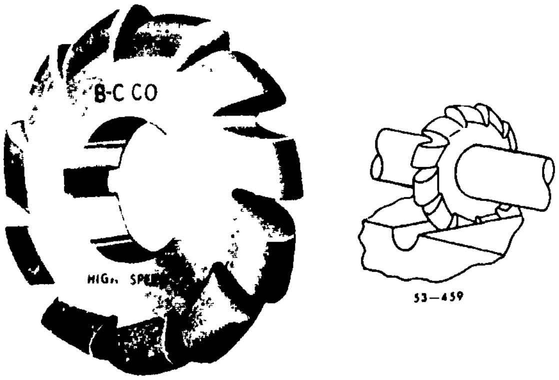


Figure 28. Convex cutter.

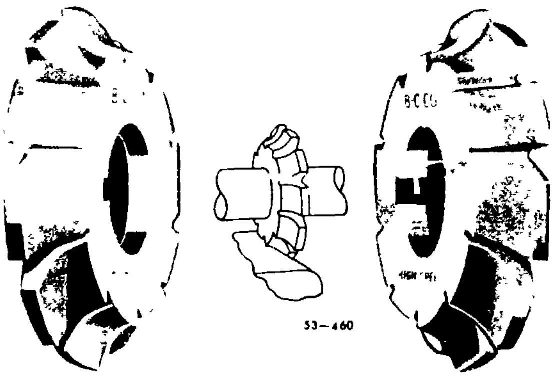


Figure 29. Corner rounding cutter.

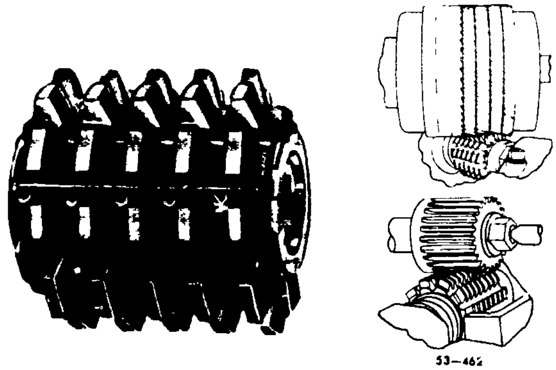


Figure 31. Gear hob.

lets are similar to lathe collets. The cup nut 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.

2-43. *Mounting and dismounting*. Mounting and dismounting arbors are relatively easy tasks. Take care not to drop the arbor on the milling machine table or floor.

2-44. To mount an arbor, first place the spindle in the lowest speed, disengage the spindle clutch lever, and turn the motor switch off. To insure accurate alinement of the arbor inside the spindle, clean the spindle hole and the arbor shank thoroughly. Stand near the column at a point where you can reach both ends of the milling machine spindle and aline the arbor keyseats with the keys in the spindle. Insert the tapered shank of the arbor into the spindle. Hold the arbor in place

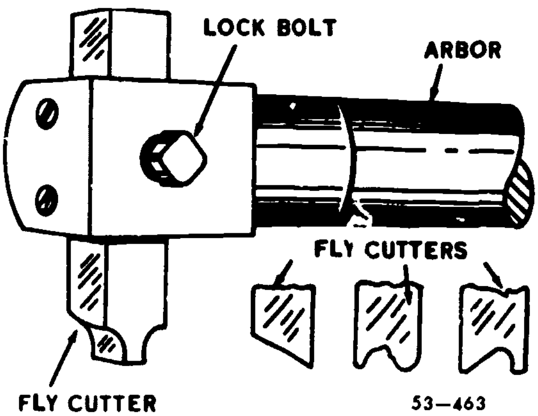


Figure 32. Fly cutter arbor and fly cutters.

with one hand and screw the drawbolt into the arbor with your other hand.

NOTE: Turn the drawbolt a sufficient number of turns to insure that the drawbolt extends into the

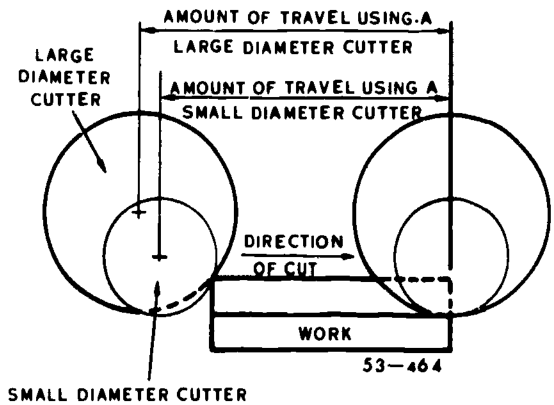


Figure 33. Cutter diameter selection.

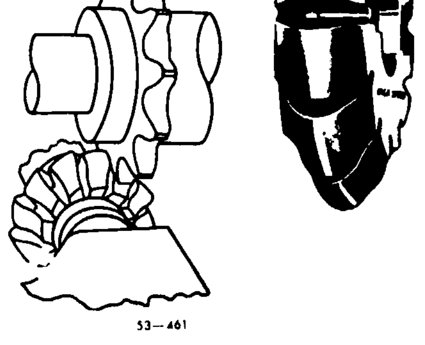


Figure 30. Sproket wheel cutter.

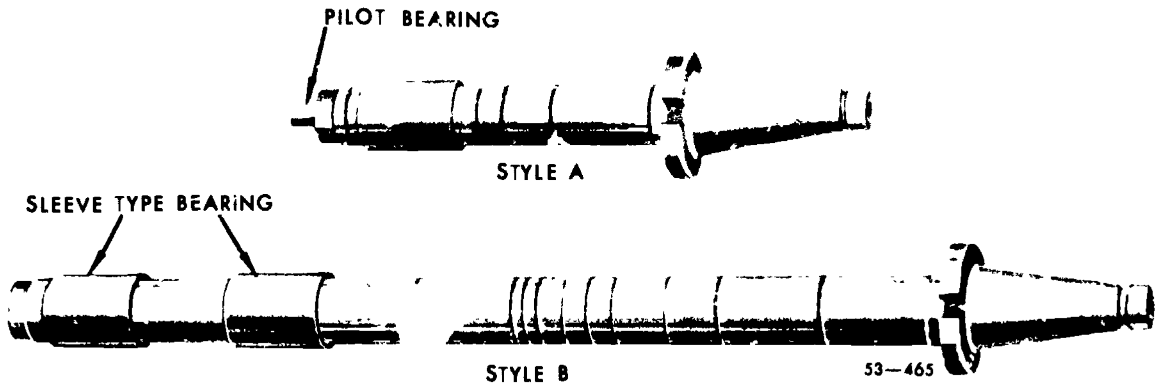


Figure 34. Standard milling machine arbors.

arbor shank a distance approximately equal to the major diameter of the threads being used. This will help to prevent stripping the threads on the drawbolt or in the arbor shank when the jamnut is tightened.

Hold the arbor in position by pulling back on the drawbolt and tighten the jamnut by hand. Tighten

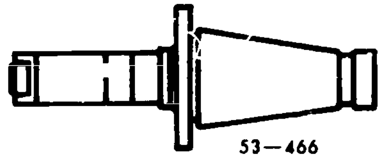


Figure 35. Stub arbor.

the jamnut with one wrench while you use a second wrench to keep the drawbolt from turning.

2-45. To dismount an arbor, again place the spindle in the lowest speed, and turn the motor switch off. Loosen the jamnut approximately two turns. Use one wrench to turn the jamnut and another wrench to keep the drawbolt from turning. 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. Hold the arbor in place with one hand and unscrew the drawbolt with your other hand. Remove the arbor from the spindle.

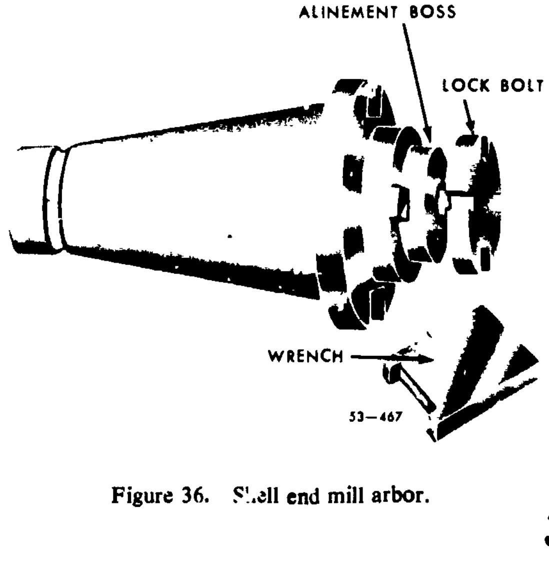


Figure 36. Shell end mill arbor.

3. Indexing

3-1. Indexing is the process that you use to divide the circumference of a workpiece into a desired number of divisions. For example, you would use indexing to space gear teeth; flutes on taps, reamers, and cutters; and the sides of bolt heads.

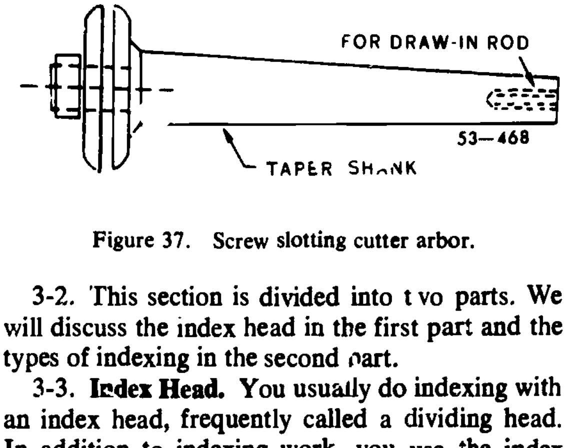


Figure 37. Screw slotting cutter arbor.

3-2. This section is divided into two parts. We will discuss the index head in the first part and the types of indexing in the second part.

3-3. **Index Head.** You usually do indexing with an index head, frequently called a dividing head. In addition to indexing work, you use the index head to mount work between centers and to rotate the work when you machine helical flutes and gear teeth. Index heads are manufactured by several machine tool companies. They vary slightly in their construction, but they are all similar. You

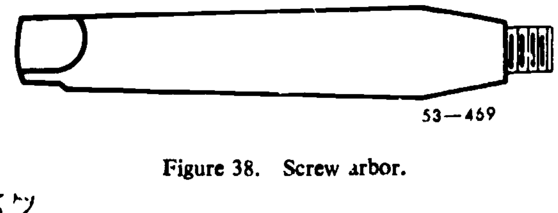


Figure 38. Screw arbor.

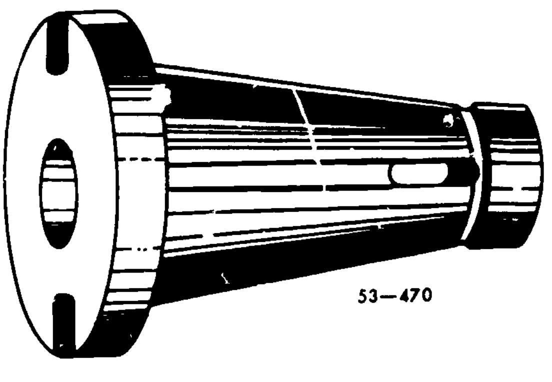


Figure 39. Taper adapter.

should have little difficulty in applying the information we will discuss to the index head that you are using. You can find additional information pertaining to indexing in machinist publications, such as the *Machinery's Handbook*. Detailed information on a specific index head can be found in the manufacturer's manual.

3-4. You must have a knowledge of the basic parts of an index head and their functions. You must know the methods which you will use to check the horizontal and vertical alinement of an index head spindle.

3-5. *Basic parts*. The basic parts of an index head are the base plate, swivel block, spindle index crank, and index plate. Refer to figures 42 and 43 as we discuss these parts.

3-6. The *base plate* is the U-shaped casting that supports the rest of the index head. The base plate is fastened to the milling machine table by T-bolts. The index head is alined with the longitudinal axis of the milling machine table by keys which are fastened to the bottom of the base plate and extend into the T-slots of the table. If a right-angle plate is not available, you can remove the keys from the mounting plate when it is necessary to position the index head at a 90° angle to the longitudinal axis of the table. A large bearing in each upright of the base plate supports the swivel block.

3-7. The *swivel block* contains the spindle and the worm and worm wheel that rotate the spindle. You can position the swivel block at any desired angle from slightly below the horizontal position to a point beyond the vertical position by means of degree graduations on the swivel block. The degree graduations vary between index heads from different manufacturers. Many index heads have a vernier scale on the base plate, graduated in 5 minutes of a degree, which enable you to position the swivel block (and the spindle) at any angle with accuracy.

3-8. The *spindle* extends through the swivel block. An internal taper in the spindle work end enables you to mount taper shank tools and centers in the spindle. You can also mount chucks on the external work end of the spindle. The chucks are usually screwed onto the spindle or bolted directly to a flange on the spindle.

3-9. You use the *index crank* to rotate the index head spindle. This crank turns a shaft which is geared to the worm shaft. You must remove the index pin from the hole in the index plate prior to turning the spindle index crank handle. You can aline the pin with the desired circle of holes by loosening the nut on the center shaft, shown in figure 43, and sliding the shaft along the slot in the crank. Some pins, such as shown in figure 44, are positioned by loosening the nut on the crank and pivoting the portion of the crank that contains the pin. Remember to tighten the nut after you position the index crank.

3-10. The *index plate,* such as the one shown in figure 45, is a disc that contains several concentric rings of holes. Each of the concentric rings has a different number of holes. As you will see when we discuss the types of indexing, the difference in the number of holes is necessary in order to index the number of divisions that may be required. Several different index plates are usually provided with each index head. The index plate is located behind the index crank. You must remove the index crank if you wish to change the index plate for one having a different number of holes. Some index plates, such as the one shown in figure 44, have a series of closely spaced notches in the edge of the plate. The notches permit you to aline holes in the index plate with the index pin when a previously machined surface is properly alined with the cutter but a hole does not aline with the pin. You use the notches in conjunction with the side plate stop. First, aline the machined surface with the cutter. Then, if a hole and index pin do not aline, loosen the side plate stop and rotate the index plate until a hole and the pin do aline. Engage the side plate stop with a notch and tighten the stop.

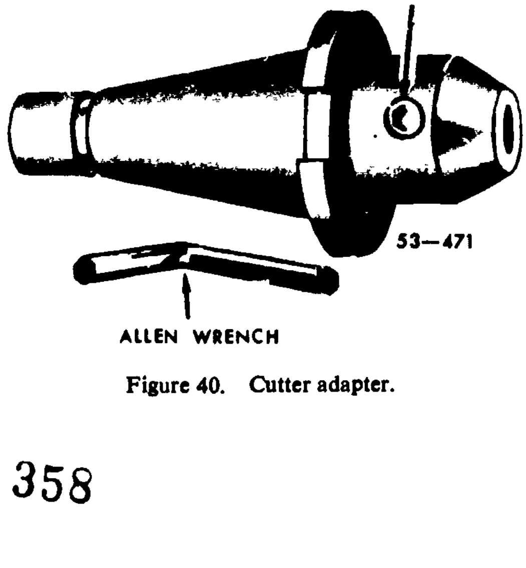


Figure 40. Cutter adapter.

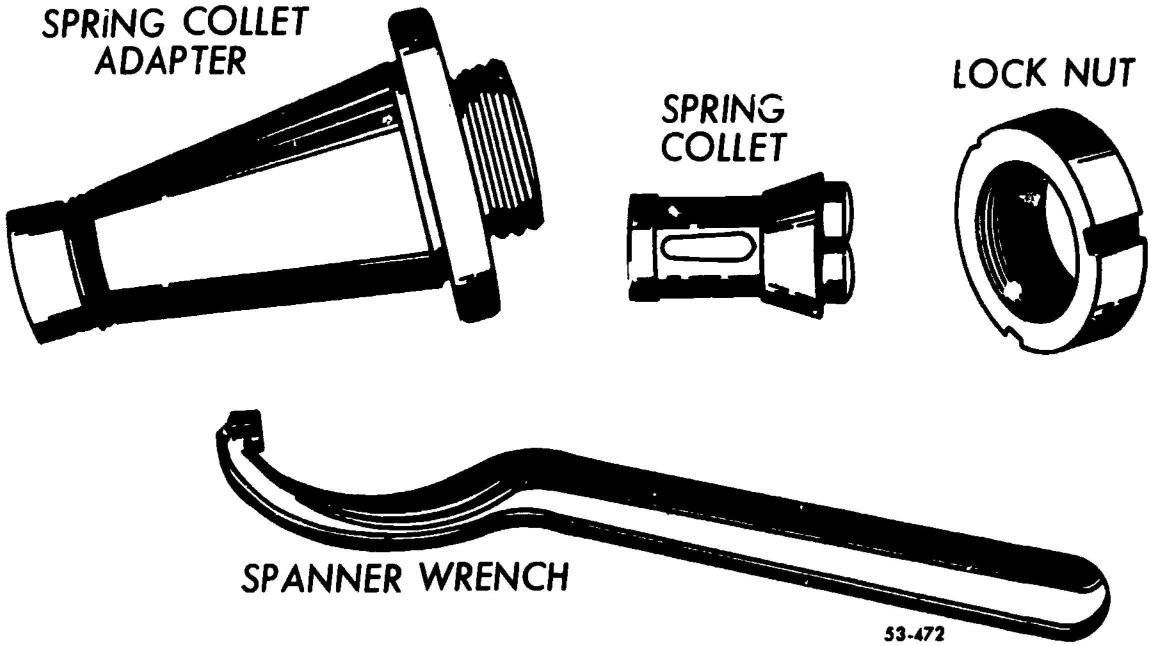


Figure 41. Spring collect chuck adapter.

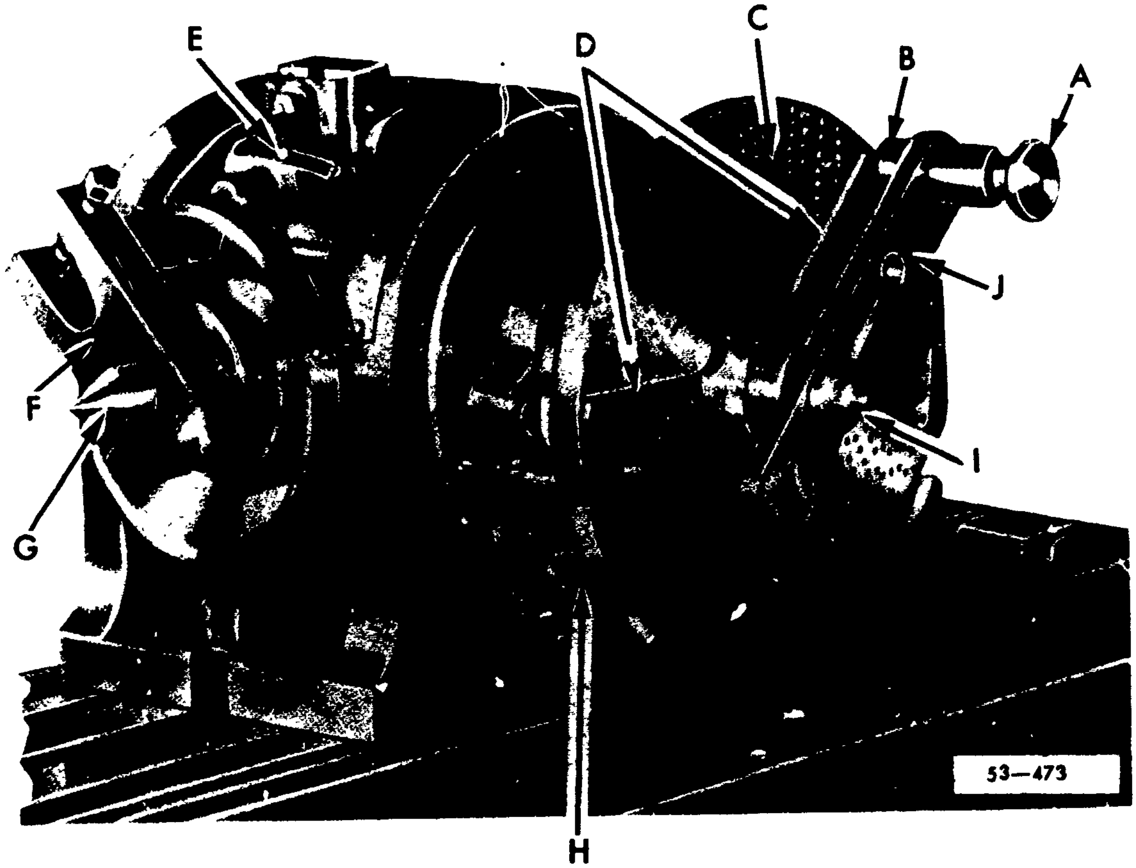
3-11. *Alinement.* Normally, the index head spindle must be in either the vertical or the horizontal position. You aline the index head spindle by loosening the swivel block clamps and tilting the swivel block to the desired position. If average accuracy is permissible, you can use the degree graduations on the swivel block to position the swivel block. When a higher degree of accuracy is required, use a dial indicator to obtain a reading along the length of a test bar held in the chuck. The indicator readings must be identical at both ends of the test bar. Unclamp and adjust the swivel block until you obtain identical readings. Check the alinement after you reclamp the swivel block. The clamping action may cause the block to shift slightly. The indicator can be clamped to the arbor, cutter, or any other suitable part of the milling machine. When work is to be machined between centers, the footstock center should be at the same height as the index head center height. You can adjust the footstock center height by loosening the lock screw and adjusting the center elevating screw. You can visually check the center alinement by placing the point of the footstock center near the point of the index head center, by cricket marks on the footstock, or by means of a dial indicator and a test bar. Place the test bar between the centers and raise or lower the footstock center until you obtain identical readings at both ends of the test bar.

3-12. **Types of Indexing.** Direct, plain, and degree indexing are methods of indexing that you may use. The method you select will depend upon the number of divisions required and the method that you use to measure the spacing between the divisions. The following information will help you understand when to use and how to perform the various types of indexing.

3-13. *Direct.* Direct indexing is the simplest means of dividing a workpiece into a required number of equal divisions. You use a plate, such as the one shown in figure 46, to divide the work. Simply rotate the work until the correct hole is alined with the index pin. The direct index plate is usually located directly behind the chuck or is a part of the spindle. A special index pin is used for direct indexing. Therefore, you do not use the index pin on the crank. Most index heads have a provision for disengaging the worm and worm wheel so that the work and spindle can be rotated by hand. You can use a plate with 24 evenly spaced holes, such as the one shown in figure 46, to index any number of divisions that can be divided evenly into 24, for example, 2, 3, 4, 6, 8, etc. Dividing 24 by the number of divisions desired gives the number of holes that you should rotate the spindle. For example, if you desire to machine a hexagonal head on a bolt, divide the number of holes on the index plate (24) by the required number of divisions (6). Therefore, you must rotate the work four holes for each of the six sides.

NOTE: Do *not* count the hole that the pin is in when you begin counting.

3-14. *Plain.* You use plain indexing when you cannot obtain the required number of divisions by



A. PLUNGER
B. INDEX PIN
C. INDEX PLATE
D. SECTOR ARMS
E. DIRECT INDEXING PIN LEVER
F. DRIVE PLATE
G. CENTER
H. SIDE PLATE STOP
I. LOCKNUT
J. CRANK

Figure 42. Universal index head.

direct indexing. In plain indexing you rotate the work by turning the index crank. The ratio between the crank and the spindle is usually 40 to 1. This means that the work rotates $\frac{1}{40}$ of a turn for each full turn of the index crank. Stated another way, it requires 40 turns of the index crank to rotate the work one complete revolution. Therefore, to determine the number of revolutions of the index crank, you must divide 40 by the number of divisions required. For example, to divide the work into 2 divisions you would rotate the index crank 20 full turns; and to obtain 4 divisions, 10 full turns, etc. Rings of holes, called hole circles, are used when the index crank must rotate a portion of a turn. If a fraction remains after you have divided 40 by the number of divisions desired, the numerator indicates the number of holes to use and the denominator indicates the number of holes in the hole circle Suppose that you are making a special bolt for an aircraft part and are ready to mill the hexagonal head. How many turns of the index crank are required to place each of the six sides in the proper position?

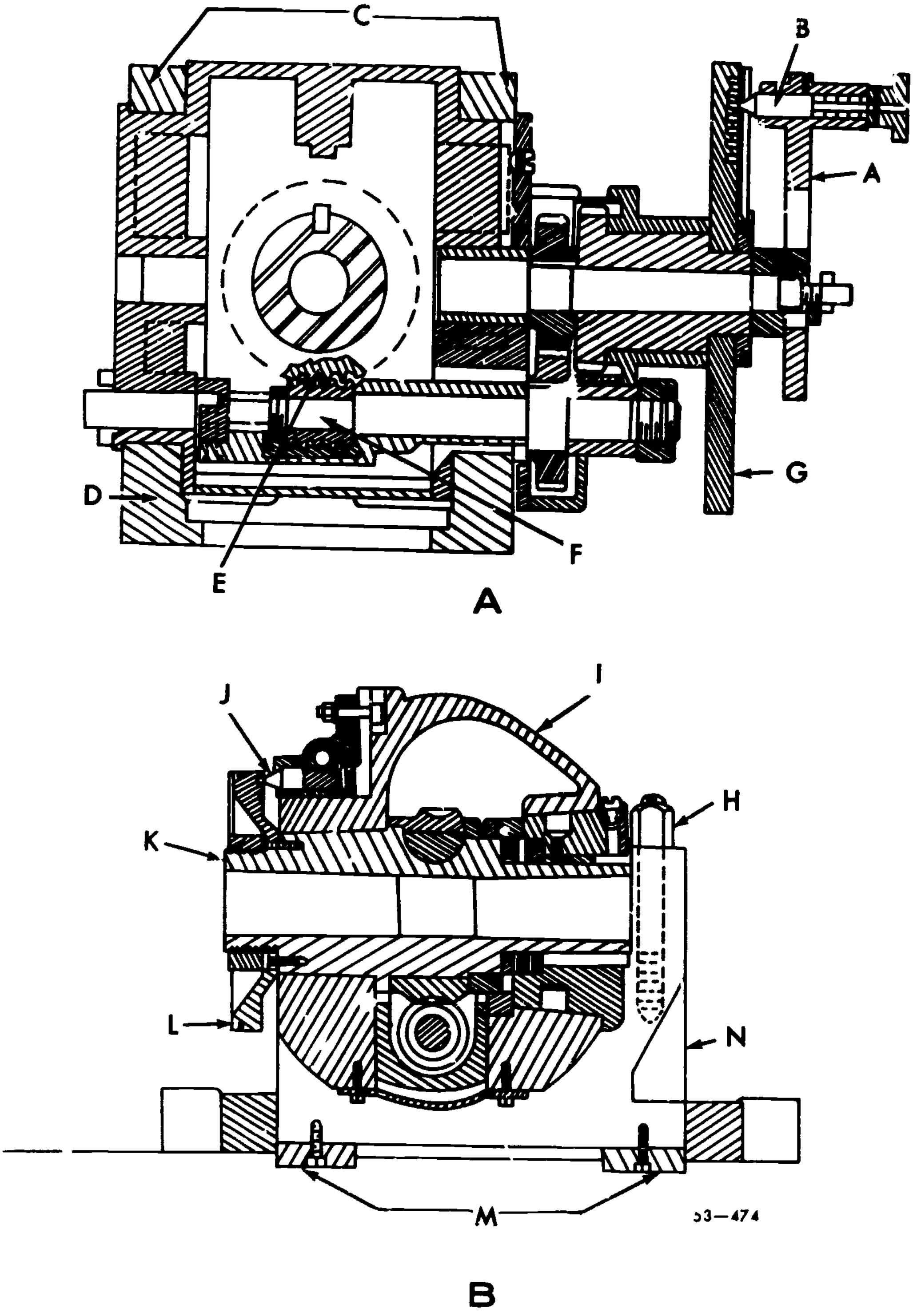
$$40 \div 6 = 6\tfrac{4}{6} \text{ turns}$$

You would move the index crank $\frac{4}{6}$ of a turn by moving it 4 holes in a 6-hole circle, if you have an index plate that has a 6-hole circle. Normally, you will not have such a plate. Therefore, you must change the fraction to an equivalent fraction that has a denominator equal to an available hole circle. If a 24-hole circle is available, you could obtain the $\frac{4}{6}$ turn by moving the index crank 16 holes in the 24-hole circle, since:

$$\frac{4}{6} = \frac{16}{24}$$

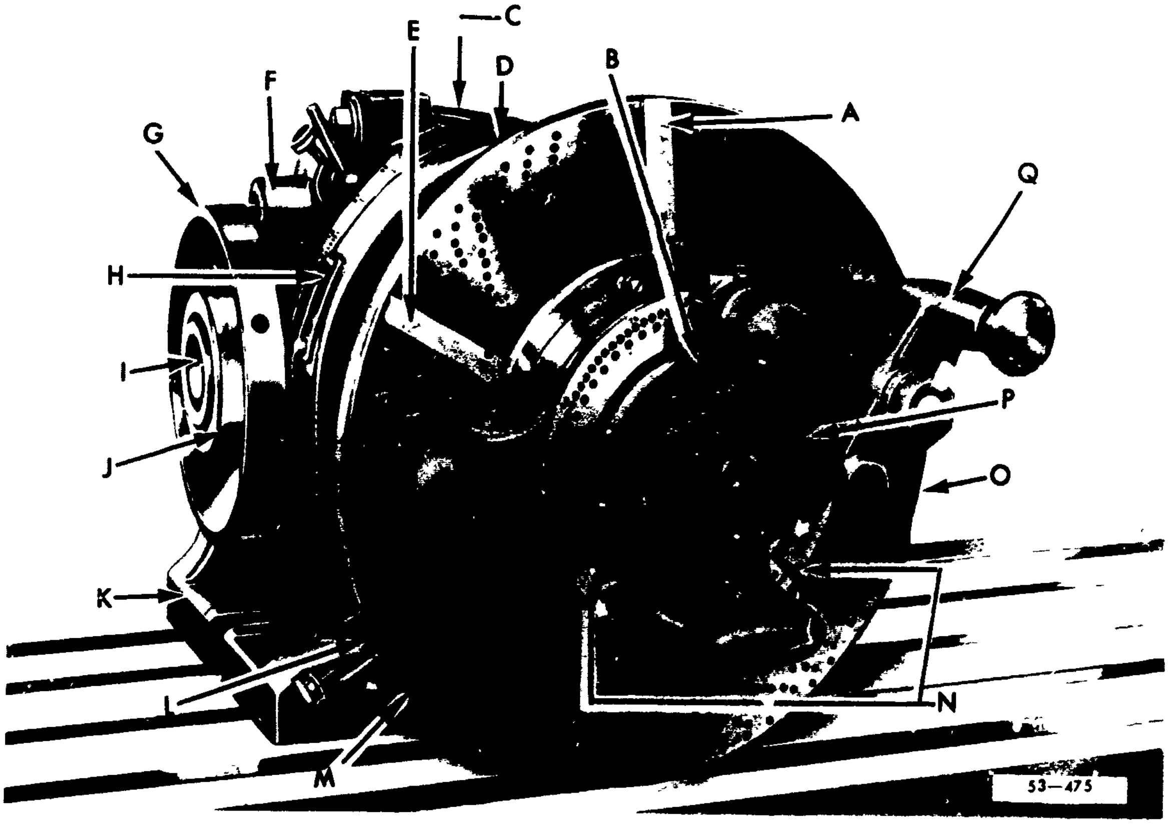
Therefore, to mill the hexagonal bolt head, you would turn the index crank 6 full turns and 16 holes in a 24-hole circle for each of the six sides

3-15. Do not change the index plate until you have checked to insure that none of the hole circles on the index plate mounted on the index head are suitable for use. For example, your calculations may indicate that you must move the index



A	B
A. INDEX CRANK	H. SWIVEL BACK CLAMP
B. INDEX PIN	I. SWIVEL BLOCK
C. BASE PLATE SIDE	J. INDEX PIN
D. BASE PLATE	K. SPINDLE
E. WORM WHEEL	L. DIRECT INDEX PLATE
F. WORM	M. ALIGNMENT KEYS
G. INDEX PLATE	N. BASE

Figure 43. Sectional view of a universal index head.



A. LARGE SECTOR ARM
B. SMALL I 'DEX PIN CRANK
C. SWIVEL BLOCK
D. CLAMP
E. LARGE SECTOR ARM
F. INDEX PIN HOLDER
G. FRONT PLATE
H. VERNIER
I. SPINDLE
J. THREAD COVER OVER SPINDLE
K. HOUSING
L. SIDE PLATE STOP
M. NOTCHES
N. SHORT SECTOR ARMS
O. INDEX PLATE—LARGE
P. INDEX PLATE—SMALL
Q. LARGE INDEX PIN CRANK

Figure 44. Wide range index head.

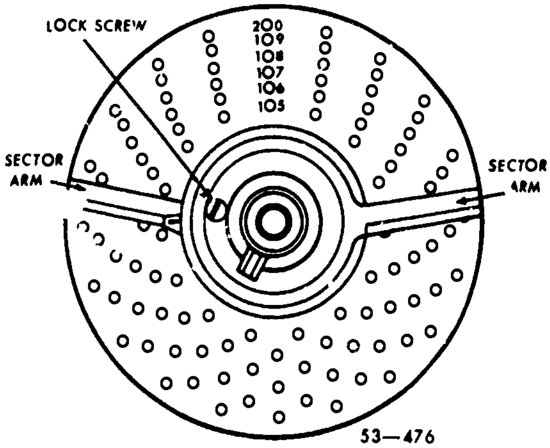


Figure 45. Index plate.

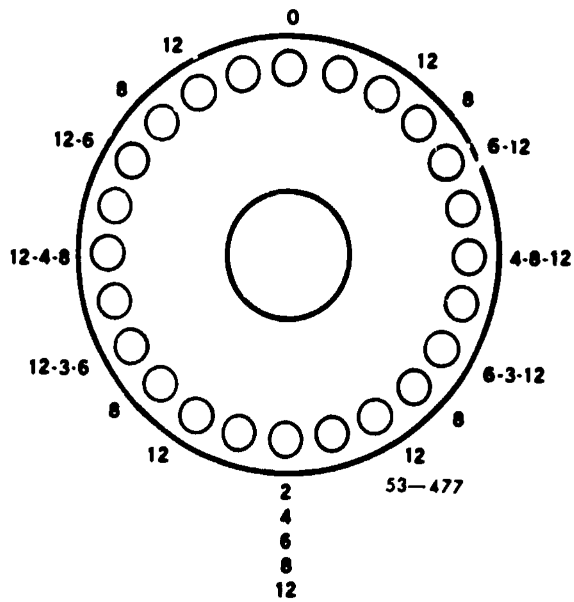


Figure 46. Direct indexing plate.

362

crank 3 holes in an 18-hole circle. Perhaps the mounted plate has a 36-hole or a 54-hole circle which you could use and thus you would not have to expend time and effort needlessly changing the plate. The sector arms, shown in figure 45, are time-saving devices that eliminate the need for counting the holes for each division. You adjust the sector arms so that the index pin and the required number of holes are located between their *beveled* sides. The arms are locked at the desired setting by tightening the setscrew shown in figure 45. Then, you position the section formed by these arms by moving it in the direction of the index crank rotation until one sector arm contacts the index pin. After you have turned the index crank the required number of full turns, continue turning the index crank until the index pin is positioned next to the other sector arm.

3-16. *Degree.* Degree indexing is indexing when the spacing between holes or surfaces is in degrees rather than in divisions. You can use direct indexing when the number of degrees you desire can be divided into 360° and when the quotient you obtain can be divided evenly into the number of holes on the direct indexing plate. For example, to divide the work into 30° divisions, divide 360° by 30. The quotient 12 indicates that you must divide the work into 12 divisions that are 30° apart. Now divide 24 (the number of holes on the direct index plate) by 12 (the number of divisions). The quotient (2) indicates the number of holes that you use to index 30° on the direct index plate.

3-17. When it is necessary to divide work into degrees by plain indexing, remember that one turn of the index crank will rotate the work $\frac{1}{40}$ of a revolution. Since one revolution of the work equals 360°, one turn of the index crank would revolve the work $\frac{1}{40}$ of 360°, or 9°. Therefore, $\frac{1}{9}$ of a turn of the index crank would revolve or index the work 1°. When you select the dividing head index plate for degree indexing a workpiece, 2 holes in an 18-hole circle index the work 1°; 1 hole in a 27-hole circle indexes $\frac{1}{3}$°; 6 holes in a 54-hole circle indexes 1°; 4 holes in a 54-hole circle indexes $\frac{1}{2}$°; and 2 holes in a 54-hole circle indexes $\frac{1}{3}$°. To determine the number of turns and parts of a turn of the index crank required to index work the desired number of degrees, you should divide the number of degrees to be indexed by 9. The quotient represents the number of complete turns and fraction of a turn that the index crank should be rotated. The sector arms are set for the number of holes that give the desired fraction of a turn. The calculation for indexing work 15° using an index plate with a 54-hole circle is as follows:

$$\frac{15}{9} = 1\frac{6}{9} = 1\frac{36}{54}$$

or one complete turn of the index crank and 36 holes in a 54-hole circle. The calculation for indexing work 13½° using the 18-hole circle of an index plate is as follows:

$$\frac{13\frac{1}{2}}{9} \times \frac{2}{2} = \frac{27}{18} = 1\frac{9}{18}$$

or 1 complete turn and 9 holes in an 18-hole circle.

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 of 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.

4. Plain, Face, and Angular Milling

4-1. Plain and face milling are the two basic milling machine operations. Both are involved in the milling of all flat surfaces. We will begin by discussing plain milling.

4-2. **Plain Milling.** Plain milling is the process of milling a flat surface in a plane parallel to the cutter axis. You obtain the desired size of the work by individually milling each of the flat surfaces on the object. Plain milling cutters, such as the ones shown in figure 9, are used for plain milling. Select a cutter that is slightly wider than the width of surface to be milled, if possible. Make the work setup *before* you mount the cutter. This precaution will prevent you from accidentally striking the cutter and cutting your hands as you set up the work. You can mount the work in a vise or fixture or clamp it directly to the milling machine table. You can use the same methods that you used to hold work in a shaper to hold work in a milling machine. Clamp the work as close to the milling machine column as possible to permit mounting the cutter near the column. The closer you place the cutter and work to the column, the more rigid the setup will be.

4-3 We will explain how you could machine a rectangular work blank (for example, a spacer for an engine test stand). You would first 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 only concerned with machining a surface in a horizontal plane.

Next, place the work in the vise, as shown in figure 47,A. Select the proper milling cutter and arbor. Wipe off the tapered shank of the arbor and the tapered hole in the spindle with a clean cloth. Mount the arbor in the spindle. Clean and position the spacing collars and place them on the arbor in such a way that the cutter is above the work. Place a square key in the arbor keyseat. 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 alined over the key. The bearing should be located as close to the cutter as possible. Make sure that the work and vise will clear all parts of the machine. Install the arbor nut and tighten it fingertight only. Position the overarm and mount the arbor support. Now, tighten the arbor nut with a wrench, after the arbor has been supported.

4-4. Set the spindle directional control lever to give the required direction of cutter rotation. Determine the required speed and feed, and set the spindle speed and feed controls. Set the feed trip dogs for the desired length of cut and center the work under the cutter. Lock the saddle. Engage the spindle clutch and pick up the cut. Pick up the surface of the work by holding a long strip of paper between the rotating cutter and the work and very slowly moving 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.

4-5. Move the work longitudinally away from the cutter and set the vertical feed graduated collar

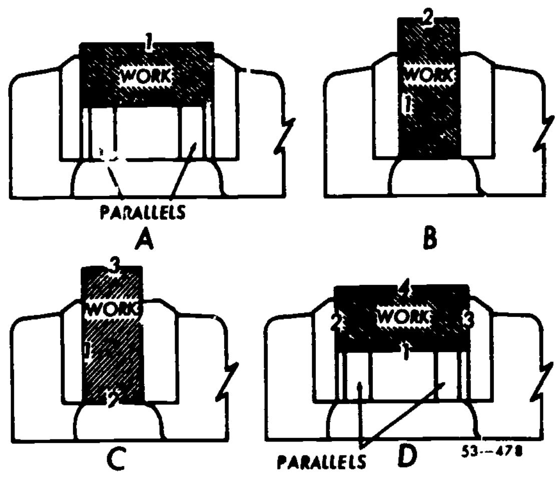


Figure 47. Machining sequence to square a block.

at ZERO. Compute the depth of the roughing cut and raise the knee this distance. Lock the knee, and direct the coolant flow on the work and cutter. Position the cutter to within $\frac{1}{16}$ inch of the work. Use hand table feed until you become proficient. Then engage the power feed. After the cut is completed, stop the spindle. Return the work to its starting point on the other side of the cutter. Then raise the table the distance required for the finish cut. Set the finishing speed and feed, and take the finish cut. When you are done, stop the spindle, and return the work to the opposite side of the cutter. Deburr the work, and remove it from the vise.

4-6. To machine the second side, place the work in the vise as shown in figure 47,B. 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 vise. Place the work in the vise, as shown in figure 47,C, 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 too deep, you will have to remove the blacklash from the vertical feed before taking the new depth of cut. You do this by lowering the knee well past the original depth of the roughing cut. Then raise the knee back to the setting used for the roughing cut. Now you raise the knee the correct distance for the finishing cut and engage the feed. After this cut, stop the spindle and return the work to the starting point on the other side of the cutter. Deburr the work and remove it from the vise. Side 4 is placed in the vise, as shown in figure 47,D, and machined, using the same procedures as for side 3. When you have completed side 4, remove the work from the vise and check it for accuracy.

4-7. This completes the machining of the four sides of the block. If the block is not too long, you can rough and finish mill the ends to size in the same manner in which you milled the sides. This is done by placing the block on end in the vise. Another method of machining the ends is by face milling.

4-8. **Face Milling.** Face milling is the milling of surfaces that are perpendicular to the cutter axis, as shown in figure 48. 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.

4-9. *Cutter setup.* You can use straight shank or taper shank end mills, shell end mills, or face milling cutters for face milling. Select a cutter that is slightly larger in diameter than the thickness of the material that you are machining. If the cutter is smaller in diameter than the thickness of the material, you will be forced to make a series of slightly overlapping cuts to machine the entire surface. Mount the arbor and cutter *before* you make the work setup. Mount the cutter by any means suitable for the cutter you have selected.

4-10. *Work setup.* Use any suitable means to hold the work for face milling, provided that 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 aline the work properly. If you use a vise to hold the work, you can aline the stationary vise jaw with a dial indicator, as shown in figure 49. You can also use a machinist square and a feeler gage, as shown in figure 50.

4-11. *Operation.* To face mill the ends of work, such as the engine mounting block that we discussed previously, you should first select and mount a suitable cutter. Then mount and position a vise on the milling machine table, as shown in figure 48, so that the thrust of the cutter is toward the solid vise jaw. Aline the solid vise jaw square with the column of the machine, using a dial indicator for accuracy. Now mount the work in the vise, allowing the end of the work to extend slightly beyond the vise jaws. Raise the knee until the center of the work is approximately centered with the center of the cutter. Lock the knee in position.

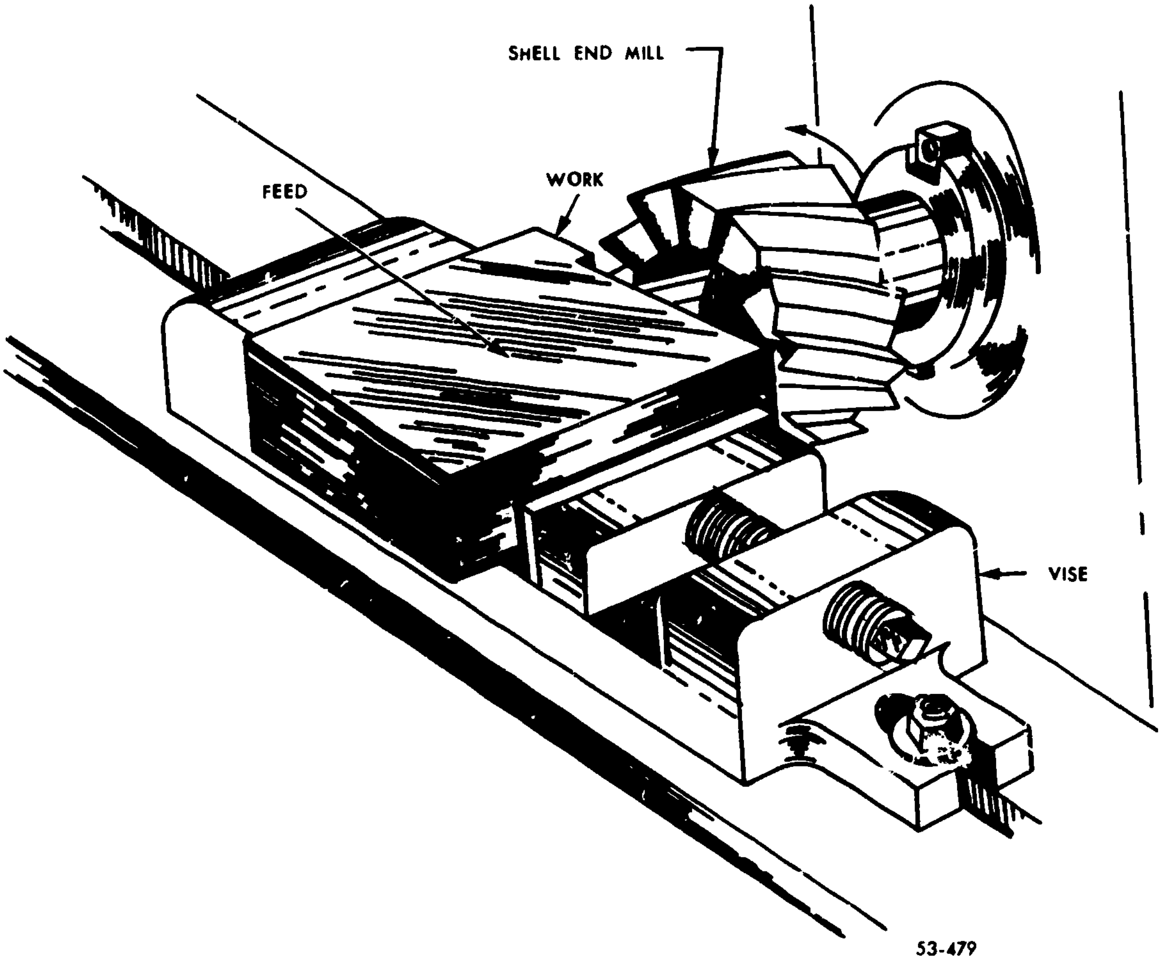


Figure 48. Face milling.

4-12. Set the machine for the proper roughing speed, feed, and table travel. Start the spindle and pick up the end surface of the work by hand feeding the work toward the cutter. Place a strip of paper between the cutter and the work, as shown in figure 51, to help pick up the surface. Once the surface is picked up, set the saddle feed graduated dial at ZERO. Move the work away from the cutter with the table and direct the coolant flow on the cutter. Set the roughing depth of cut, using the graduated dial, and lock the saddle. Position the work to about $\frac{1}{16}$th inch from the cutter, then engage the power feed. After the cut is complete, stop the spindle, and move the work back to the starting point before the next cut.

4-13. Set the speed and feed for the finishing cut, and then unlock the saddle. Move it in for the final depth of cut, and relock it. Engage the spindle and take the finish cut. Then stop the machine and return the work to the starting place. Shut the machine off. Remove the work from the vise. Handle it very carefully to keep from cutting yourself before you can deburr the work. Next, mount the work in the vise so that the other end is ready for machining. Mill this end in the same manner as the first, only now you have 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.

4-14. **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 52. However, you can perform angular milling with a plain, side, or face milling cutter by positioning the work at the required angle.

5. Milling Flat Surfaces on Cylindrical Work

5-1. Many maintenance or repair tasks involve machining flat surfaces on cylindrical work. These tasks include milling squares and hexagons, and milling two flats in same plane.

5-2. **Squares and Hexagons.** You will machine squares and hexagons frequently on the ends of bolts, taps, reamers, or other items that are turned

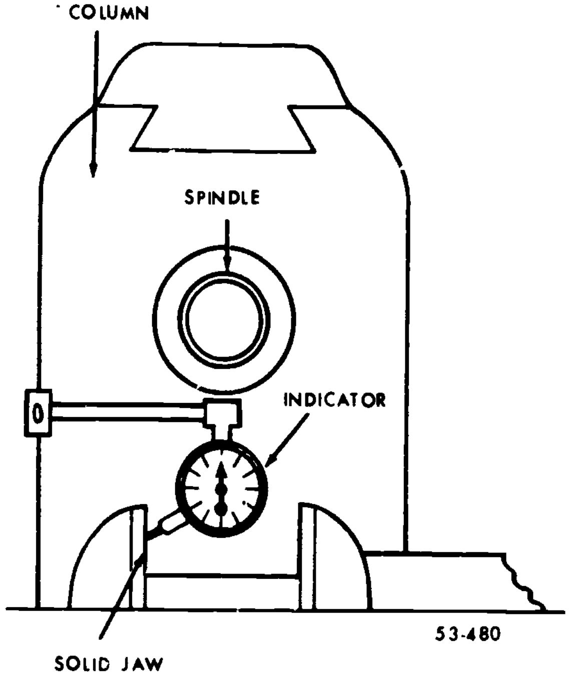


Figure 49. Alining vise jaws using an indicator.

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.

5-3. *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 to machine work that is held in a chuck, and for heavy cutting. You can use end mills for work that is held in a chuck or between centers, and for light cutting. If you use a side milling cutter, be sure that the cutter diameter is large enough to permit machining the full length of the square or hexagon without interference by the arbor. If you use an end mill, it should be 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 53 and 54. The cutter shown in figure 53 rotates in a counterclockwise direction and the work is fed toward the left. The cutter shown in figure 54 rotates in a clockwise direction and the work is fed upward.

5-4. *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 aline the index spindle in either the vertical or horizontal plane. If you are machining work between centers, you must also aline the footstock center. If you use a screw-on chuck, take into consideration the cutter rotary thrust applied to the work. Always cut on the side of the work which 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 55, contains two lock screws. One lock screw clamps the drive plate to the index center and insures 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 55,A. This eliminates any movement of the work during the machining operation. It may be necessary, especially if you are using a

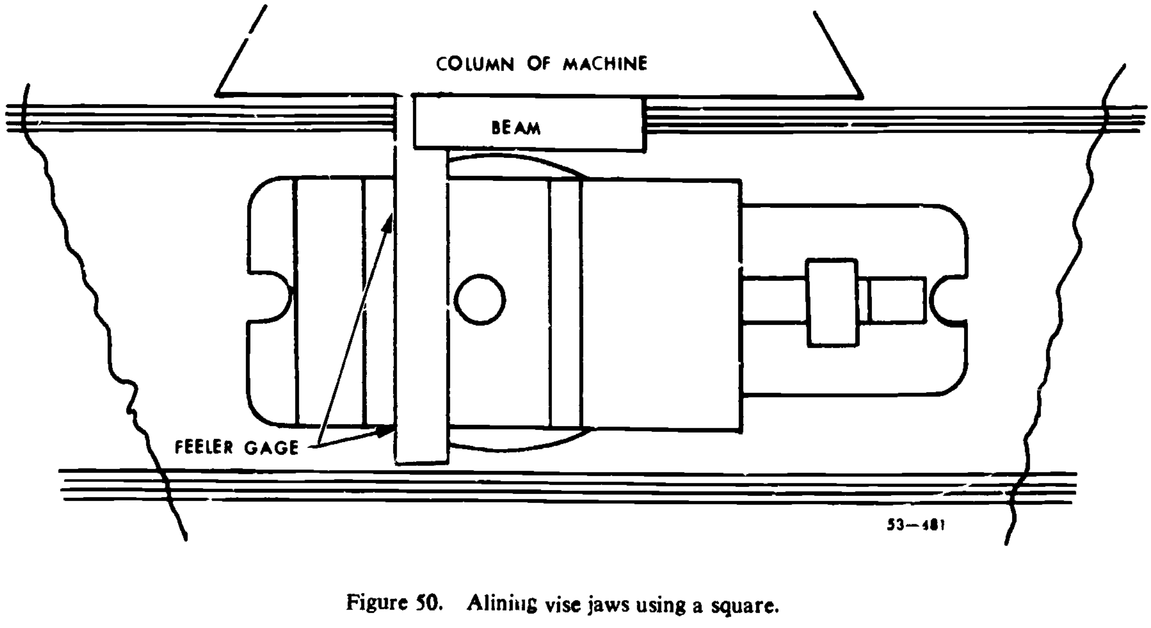


Figure 50. Alining vise jaws using a square.

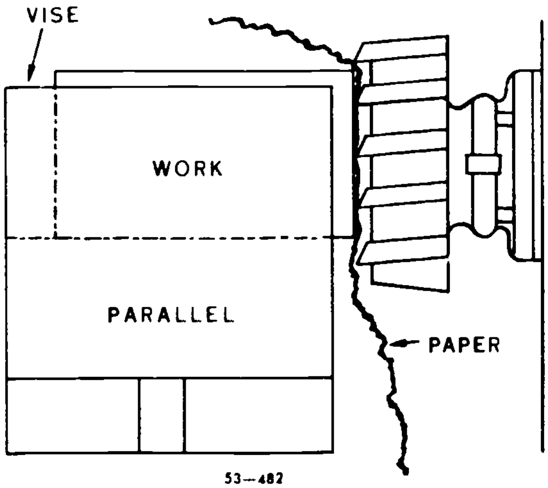


Figure 51. Picking up the work surface.

short end mill, to position the index head near the rear edge of the table to insure that cutter and work make contact.

5-5. *Calculations.* The following information will help you determine the amount of material that 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.

5-6. The size of a square, H, in figure 56, is measured across the flats. The largest square that you can cut from a given size of round stock equals the diameter of the stock in inches (which is also the diagonal of the square) times 0.707. This may be expressed as:

$$H = G \times 0.707$$

The diagonal of a square equals the distance across the flats times 1.414. This is expressed as:

$$G = H \times 1.414$$

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. Thus:

$$I = \frac{G - H}{2}$$

You use the same formula ($I = \frac{G - H}{2}$) determine the amount of material to remove when you are machining a hexagon.

5-7. The size of the largest hexagon that you can machine from a given size of round stock (H in fig. 57) is equal to the diagonal (the diameter of the stock) of the hexagon times 0.866 inch or:

$$G = H \times 0.866$$

The diagonal of a hexagon equals the distance across the flats times 1.155, or:

$$G = H \times 1.155$$

The length of a flat is equal to one-half the length of the diagonal, or:

$$r = \frac{G}{2}$$

5-8. *Operations.* 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. Two examples would be the heads of bolts and the square on taps and hand reamers. We will explain two methods of machining a square or hexagon: machining work mounted in a chuck and machining work mounted between centers.

5-9. Machining a square or hexagon on work mounted in a chuck can be done 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. A thin film of clean machine oil should be spread 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 it on the milling machine table.

After the index head is mounted on the table, position the spindle in the vertical position, as shown

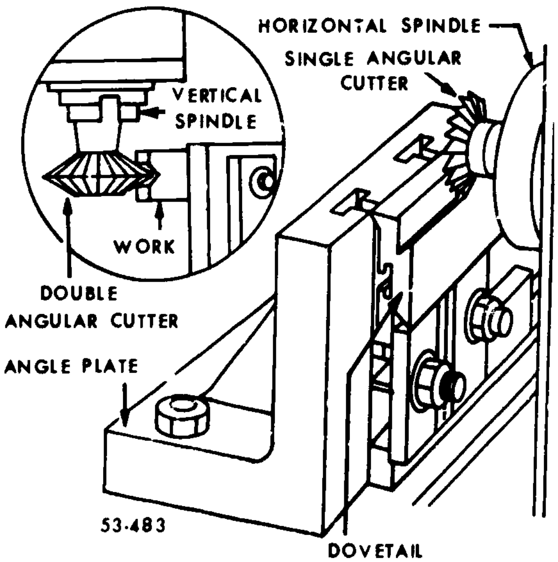


Figure 52. Angular milling.

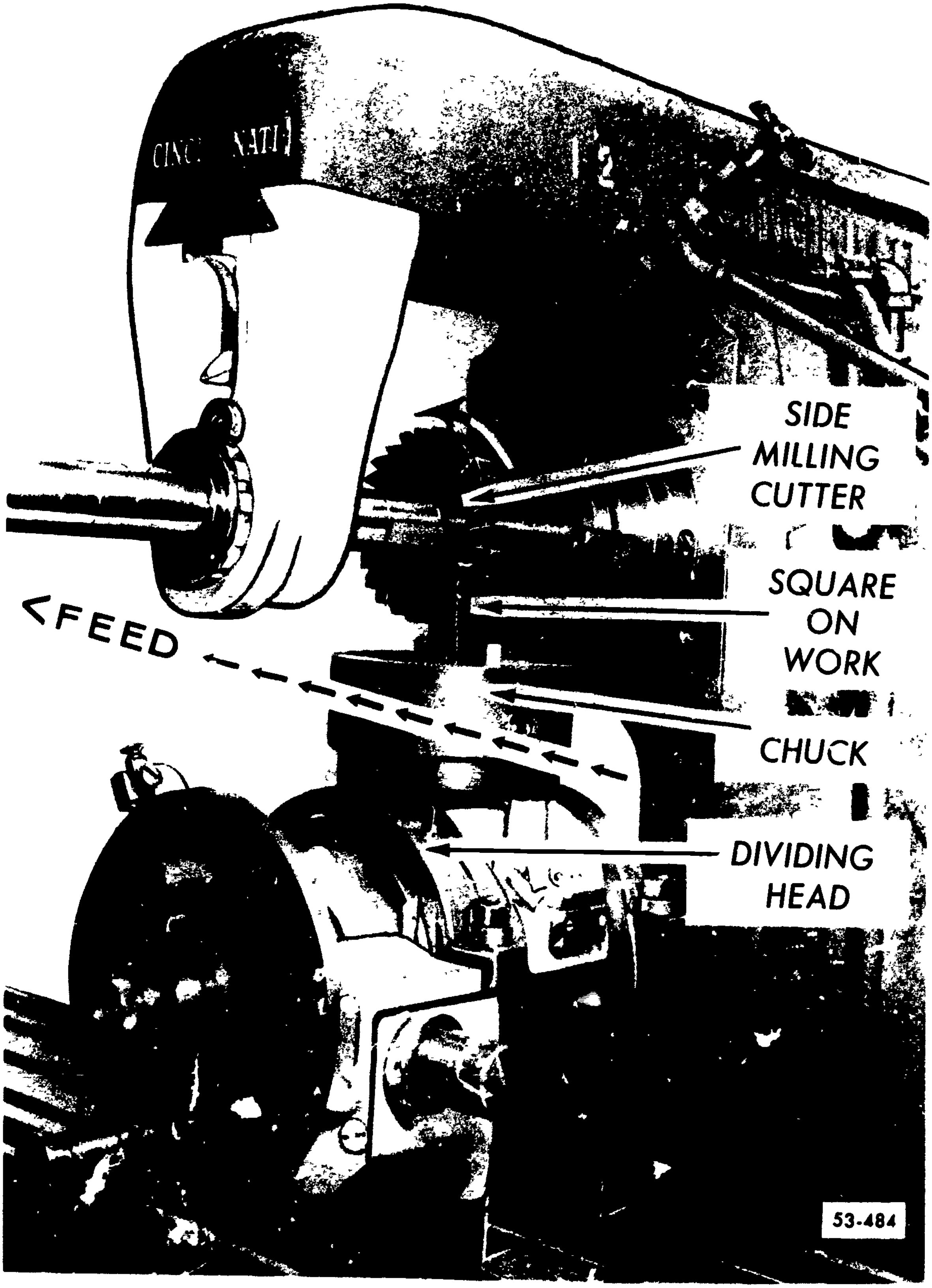
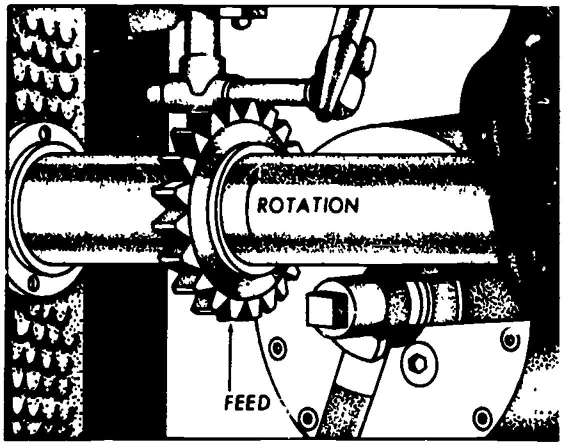


Figure 53. Milling a square on work held ertically.



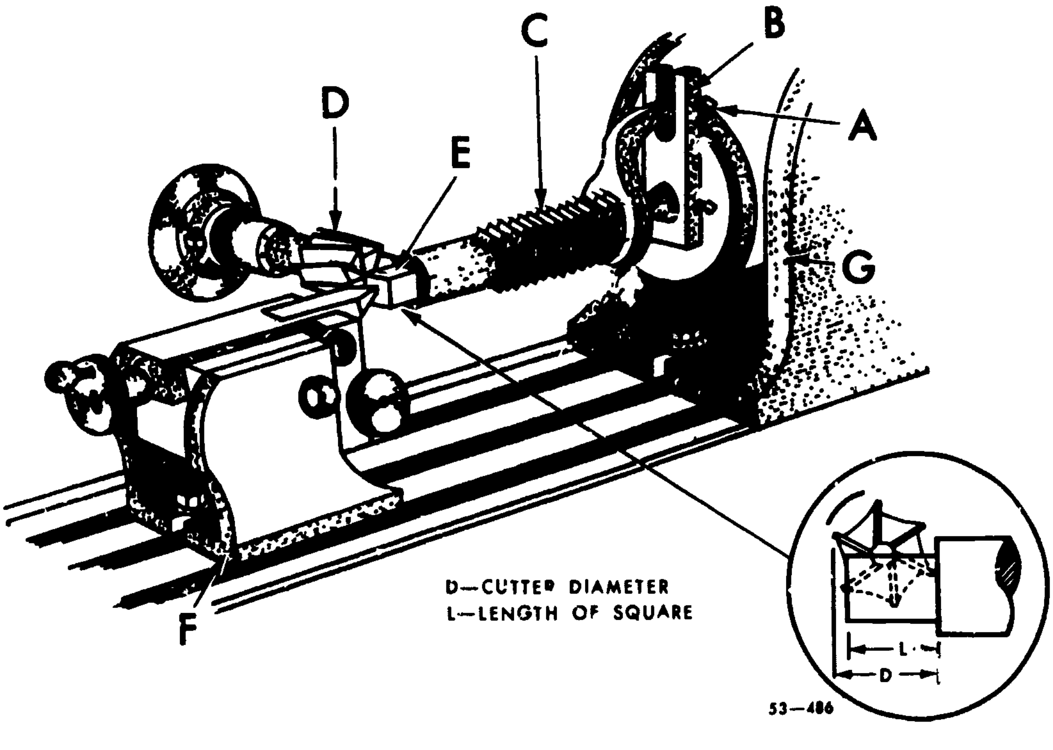
53—485

Figure 54. Milling a square on work held horizontally.

in figure 53. Use the degree graduations on the swivel block. This is accurate enough for most work requiring the use of the index head. The vertical position also means that the work will feed horizontally.

5-10. You should now tighten the work in the chuck to keep it from turning due to the cutter thrust. The arbor, cutter, and arbor support should now be installed. The cutter should be as close to the column as practical. Remember, this is done so that the setup will be more rigid. Set the machine for the correct roughing speed and feed. With the cutter turning, pick up the cut on the end of the work. Then move the work sideways to clear the cutter and raise the knee a distance equal to the length of the flat surfaces to be cut. Now, 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. Set the crossfeed graduated dial at ZERO, and move the work clear of the cutter. Remember, the cutter should rotate so that the cutting action takes place as in "up milling." Feed the table in the required amount for a roughing cut, and engage the power feed and coolant flow. When the cut is finished, stop the spindle, and return the work to the starting point.

5-11. Loosen the index head spindle lock, and rotate the work one-half revolution with the index crank. Tighten the index head spindle lock, and take another cut on the work. When this cut is finished, stop the cutter and return the work to the starting point. 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



53—486

A. LOCK SCREW FOR DOG
B. DRIVE PLATE
C. TAP
D. END MILL
E. TAP SQUARE
F. FOOTSTOCK
G. INDEX HEAD

Figure 55. Milling a square using an end mill.

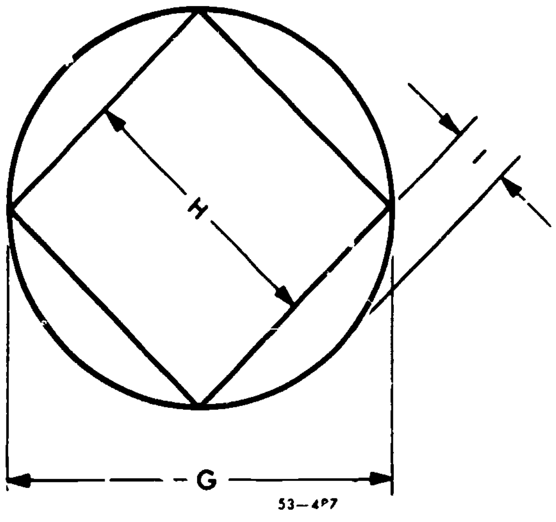


Figure 56. Diagram of a square.

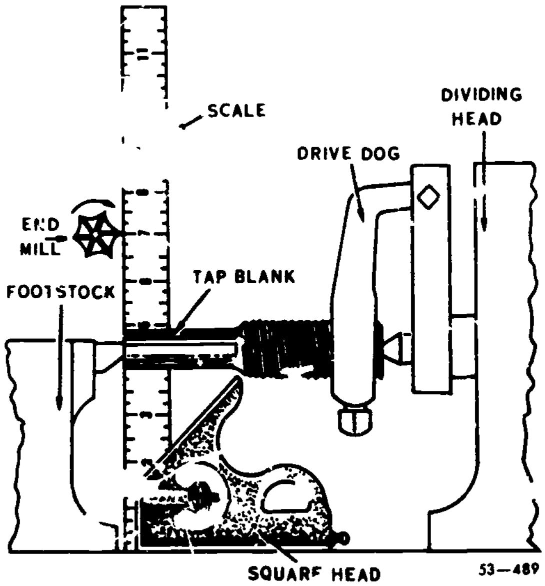


Figure 58. Alining the work and cutter.

setup, as a mistake may have been made. 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. Then return the work to the starting point again, loosen the spindle lock, rotate the work one-half revolution, and take the fourth cut. Return the work again to the starting point, and set the machine for finishing speed and feed. Now, finish machine opposite sides (1 and 3), using the same procedures already mentioned. Check the distance across these sides. If it is correct, finish machine the two remaining sides. Then 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 54 and 55. If you use the horizontal setup, you must feed the work vertically.

5-12. Machining a square or hexagon on work mounted between centers is done in much the same manner as in holding the work in a chuck. The index head is mounted the same way, only with the spindle in a horizontal position. The feed will be in a vertical direction. Insert a center in the

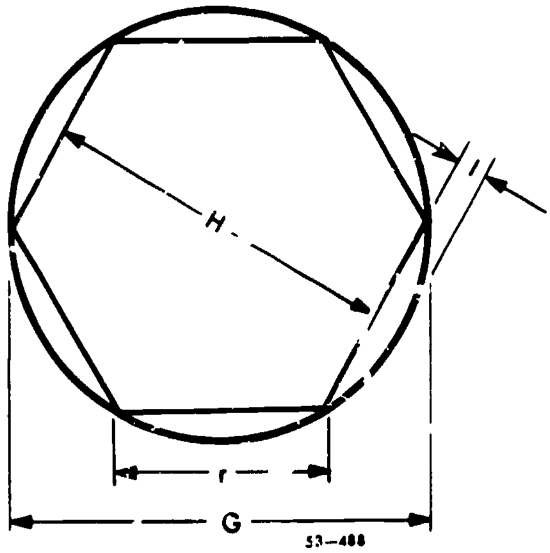


Figure 57. Diagram of a hexagon.

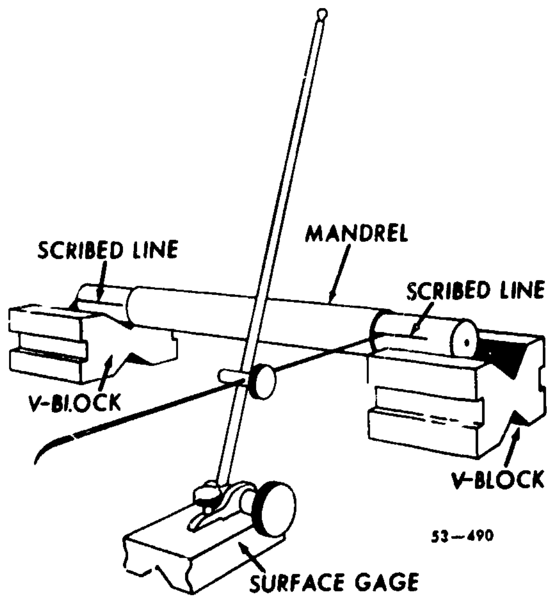


Figure 59. Layout of work.

spindle and aline it with the footstock center. Then 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 55.

5-13. Mount the work between centers. Make sure that the drive dog is holding the work securely. Set the machine for roughing speed and feed. Pick up the side of the work and set the graduated crossfeed dial at ZERO. Lower the work until the cutter clears the footstock and move the work until the end of the work is clear of the cutter. Aline the cutter with the end of the work. Use a square head and rule, as shown in figure 58.

NOTE: Turn the machine off before alining the cutter by this method.

Now move the table a distance equal to the length of the flat desired. Move the saddle the distance required for the roughing depth of cut. Feeding the work vertically, machine side 1. Lower the work below the cutter when the cut is complete. Loosen the index head spindle lock, and index the work one-half revolution to machine the flat opposite side 1. Tighten the lock, and engage the power feed. After the cut is complete, again lower the work below the cutter, and stop the cutter. Measure the distance across the two flats to check the accuracy of the cuts. If it is correct, index the work one-quarter revolution to machine another side. Then lower the work, index one-half revolution, and machine the last side. Remember to lower the work below the cutter again. Set the machine for finishing speed, feeds, and depth of cut, and finish machine all the sides. Deburr the work and check it for accuracy.

5-14. **Two Flats in One Plane.** You will often machine flats on shafts to serve as seats for setscrews. One flat is simple to machine. You can machine it in any manner with a side or end mill, as long as you can mount the work properly. However, machining two flats in one plane, such as the flats on the ends of a mandrel, presents a problem, since the flats must aline 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.

5-15. We will describe the method that is used when the size or shape of the work requires repositioning it in order to machine both flats. You should first apply layout dye on both ends of the work. Then place the work on a pair of V-blocks, as shown in figure 59. Set the scriber point of the surface gage to the center height of the work. Scribe horizontal lines on both ends of the work, as illustrated in figure 59. Now mount the index head on the table with its spindle in the horizontal position. Again, set the surface gage scriber point, but to the centerline of the index head spindle. Insert the work in the index head chuck with the end of the work extended far enough to permit all required machining operations. To aline the surface gage scriber point with the scribed horizontal line, rotate the index head spindle. Then lock the index head spindle in position.

5-16. These flats can be milled with either an end 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.

Now, raise the knee with the surface gage still set at center height until the cutter centerline is alined with the scriber point. This puts the centerlines of the cutter and the work in alinement with each other. 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. After starting the machine, feed the work by hand so that the cutter contacts the side of the work the scribed line is on. Now move the work clear of the cutter and stop the spindle. 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 in order to center the mark on the layout line. Once the mark is centered, take light "cut and try" depth of cuts until the desired width of the flat is reached. Then machine the flat to the required length. When one end is completed, remove the work from the chuck. Turn the work end for end, and reinsert it in the chuck. Machine the second flat in the same manner as you did the first. Deburr the work and check it for accuracy.

5-17. You can check the flats to see if they are in the same plane by placing a matched pair of parallels on a surface plate and placing one flat on each of the parallels. If the flats are in the same plane, you will not be able to wobble the work.

6. Slotting, Parting, and Milling Keyseats and Flutes

6-1. 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.

6-2. **Slotting.** You can cut internal contours, such as internal gears and splines and six- or twelve-point sockets by slotting. Most slotting is

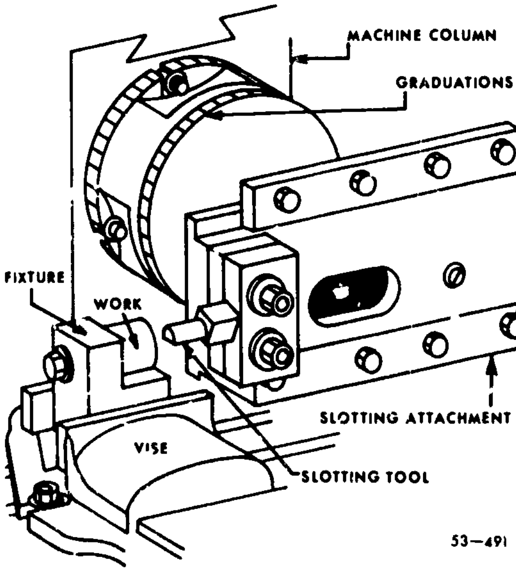


Figure 60. Slotting attachment.

done with a milling machine attachment called a slotting attachment, as shown in figure 60. 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 reciprocat'ng 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:

$$\text{Strokes per minute} = \frac{\text{CFS} \times 4}{\text{length of stroke}}$$

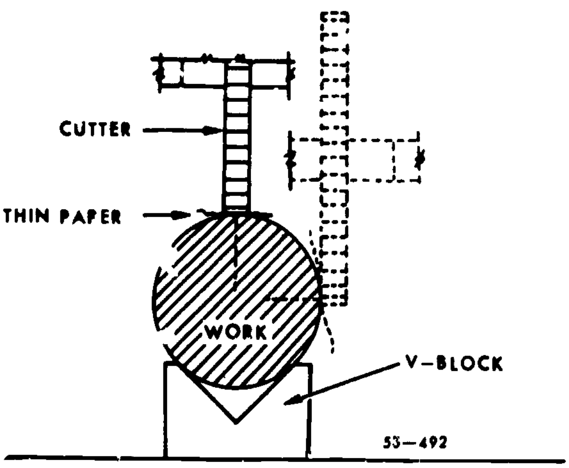


Figure 61 Alining cutte using a paper strip.

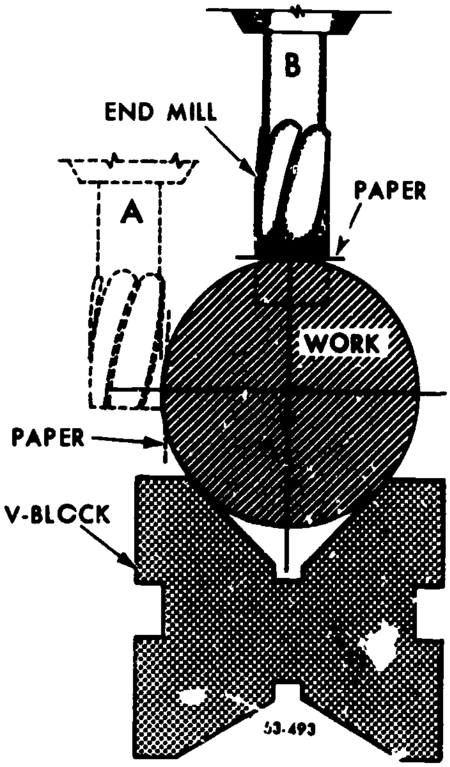


Figure 62. Alining an end mill with the work.

The cutting tools which you use with slotting attachments are ground to any desired shape from high-speed steel tool blanks. These tools are then 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. hen it is possible, position the slotting attachmen' and the work in the vertical position in order to provide the best possible view of the cutting action of the tool.

6-3. **Parting.** Use a metal slitting saw for sawing or par ing operations and for milling deep slots

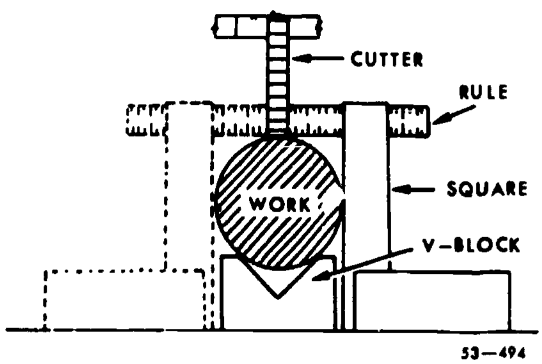


Figure 63. Visual alinement of a cu'ter.

in metals and in a variety of materials. Efficient sawing depends to a large extent upon the slitting saw you select. The work required of slitting saws varies greatly. It would not be efficient to use the same saw to cut very deep narrow slots, part thick stock, saw thin stock, or saw hard alloy steel. Soft metals, such as copper and babbit, or nonmetallic materials, such as bakelite, fiber, or plastic, require their own styles of slitting saw.

6-4. 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.

6-5. A coarse-tooth slitting saw is best for sawing brass and for cutting deep slots. A fine-tooth slitting saw is best for sawing thin metal, and a staggered-tooth slitting saw is best for making heavy deep cuts in steel. You should use slower feeds and speeds to saw steels to prevent cutter breakage. Use conventional milling in sawing thick material. In sawing thin material, however, clamp the stock directly to the table and use down milling. Then the slitting saw will tend to force the stock down on the table. Position the work so that the slitting saw extends through the stock and into a table T-slot.

6-6. **External Keyseat.** Machining an external keyseat on a milling machine is less complicated than machining it on a shaper. In milling, starting an external keyseat is no problem. You simply bring the work in contact with a rotating cutter and start cutting. It should not be difficult for you to picture in your mind how you would mill a straight external keyseat with a plain milling cutter or an end mill. If the specified length of the keyseat exceeds the length you can obtain by milling to the desired depth, you can move the work in the direction of the slot to obtain the desired length. Picturing in your mind how you would mill a Woodruff keyseat should be easier. The secret is to select a cutter that has the same diameter and thickness as the key. How you machine internal keyseats will be explained in the section on internal slotting. If your memory of keyseats and how to lay them out and calculate and measure their depth is a bit hazy, review the material on keyseats in the chapter on shaper work (Chapter 2 of Volume 3).

6-7. *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.

6-8. Before you can begin milling the keyseat, you must aline the axis of the work with the midpoint of the width of the cutter. Figure 61 shows one method of alinement. Let us 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.

6-9. You use a similar method to aline 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 62,A. After the paper tears, lower the work to just below the bottom of the end mill. Then move the work and saddle a distance equal to the radius of the work PLUS the radius of the end mill. Move the work up, using hand feed, until a piece of paper held between the work and the bottom of the end mill begins to tear, as shown in figure 62,B. Then move the table and work away from the bottom of the end mill. Set and lock the graduated dial at ZERO on the vertical feed, and then feed up for the roughing cut. The cutter rpm and the longitudinal feed are computed in the same manner as for conventional milling cutters. Because of the higher speeds and feeds involved, more heat is generated, so the work and cutter should be flooded with coolant.

6-10. When extreme accuracy is not required, you can aline the work with the cutter visually, as shown in figure 63. Position the work as near the midpoint of the cutter as possible by eye. Make the final alinement 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 on the steel rule. This method can be used with both plain milling cutters and end mills.

6-11. Before you begin to machine the keyseat, you should measure the width of the cut. You cannot be certain that the width will be the same as the thickness of the cutter. The cutter may not run exactly true on the arbor or the arbor may not run exactly true on the spindle. The recommended practice is to nick the end of the work with the cutter and then measure the width of the cut.

6-12. Specifications for the depth of cut are usually furnished. When specifications are not available, the total depth of cut for a square keyseat may be determined by means of the following formula and the dimensions in figure 64:

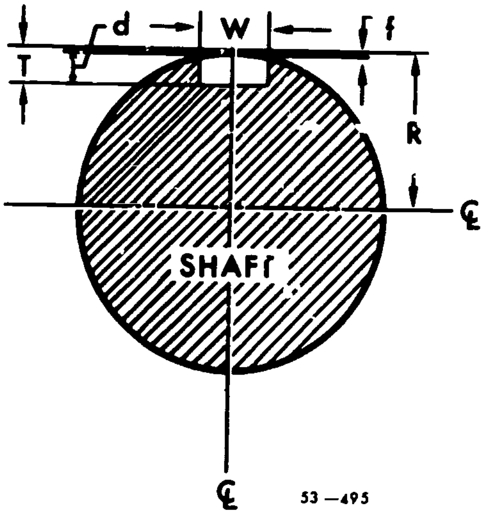


Figure 64. Keyseat dimensions for straight square key.

Total depth of cut (T) $= d + f$,

where

$$d = \frac{W}{2} = \text{depth of keyseat}$$

$$f = R - \sqrt{R^2 - (\frac{W}{2})^2} = \text{height of arc}$$

W = width of key
R = radius of shaft

The height of arc (f) for various sizes of shafts and keys may be found in chart 2. Keyseat dimensions for rounded end and rectangular keys may be obtained from the *Machinery's Handbook.* Keyseats may be checked for accuracy with rules, outside and depth micrometers, vernier calipers. and go-no-go gages. Chart 2 can be used for both square and Woodruff key seats, which will be explained next.

6-13. *Woodruff keyseat.* A woodruff key is a small half-disc 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 aline the work with the cutter and measure the width of the cut in exactly the same manner as for milling straight external keyseats.

6-14. A Woodruff keyseat cutter has deep flutes cut across the cylindrical surface of the teeth. Figure 65 shows a Woodruff keyseat cutter. The cutter is slightly thicker at the crest of the teeth than it is at the center. This feature provides clearance between the sides of the slot and the cutter. Two-inch-diameter and larger cutters 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 provide additional clearance. Also, note that large cutters usually have staggered teeth to improve the cutting action.

6-15. As you may remember, 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 66.

6-16. In milling the keyseat, the cutter is centrally located over the position in which the keyseat is to be cut and parallel with the axis of the work. The work should be raised 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 the graduated dial on the vertical feed should be set at ZERO and the clamp on the table set. With the graduated dial as a guide, the work is raised by hand until the full depth of the keyseat is cut. Should specifications for the total depth of cut not be available, the correct value may be determined by means of the following formula:

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

6-17. The most accurate way to check the depth of a Woodruff keyseat is to insert a Woodruff key of the correct size in the keyseat. Measure over the key and work with an outside micrometer to obtain the distance M in figure 67. The correct micrometer reading for measuring over the shaft and key can be determined by the formula:

$$M = D + \frac{(W)}{(2)} - f$$

Where

M = micrometer reading.
D = diameter of shaft.
W = width of key.
f = height of arc.

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.

CHART 2

VALUES FOR FACTOR (F) FOR VARIOUS SIZES OF SHAFTS

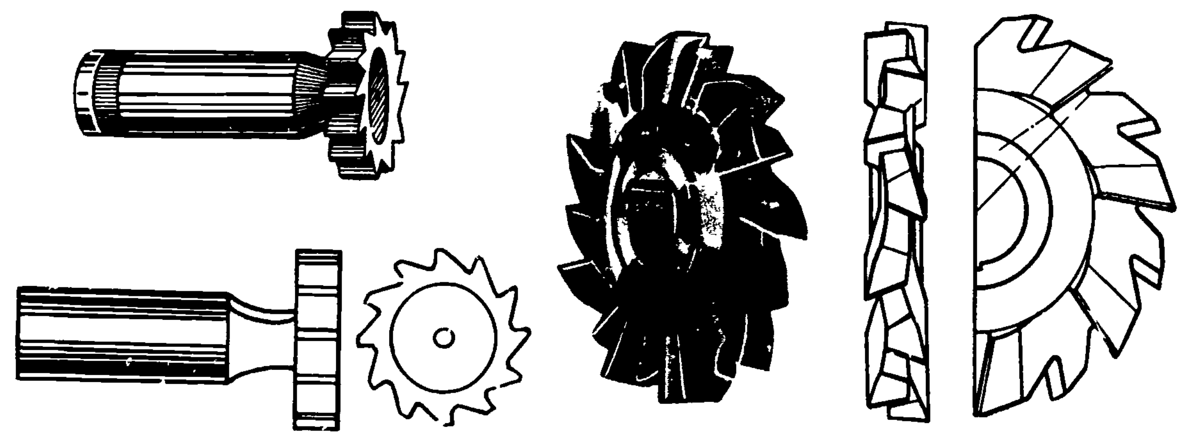
WIDTH OF KEY IN INCHES								
DIAMETER OF SHAFT (INCHES)	1/16	3/32	1/8	5/32	3/16	7/32	1/4	5/16
SHAFT SIZE	FACTOR (f)							
1/2	.002	.004	.008	.013	.018	.025	.033	---
5/8	.001	.003	.006	.010	.014	.019	.025	.042
3/4	.001	.003	.005	.008	.012	.016	.022	.034
7/8	.001	.002	.004	.007	.010	.014	.018	.028
1	.001	.002	.004	.006	.009	.012	.015	.024
1 1/8	----	.002	.003	.005	.008	.011	.014	.022
1 1/4	----	.002	.003	.005	.007	.010	.013	.019
1 1/2	----	.001	.002	.004	.006	.008	.011	.016
1 3/4	----	.001	.002	.003	.005	.007	.009	.014

53·

6-18. **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 enable coolant to reach the cutting edges. The shape of the flute and tooth depends upon the cutter you use to machine the flute. The following information pertains specifically to taps and reamers. Since flutes are actually special purpose grooves, you can apply much of the information to grooves in general.

6-19. *Tap flutes.* You usually use a convex cutter to machine tap flutes. This type of cutter produces a "hooked" flute, as shown in figure 68. The number of flutes is determined by the diameter of the tap. Taps ¼ inch to 1 ¾ inches in diameter usually have four flutes and taps 1 ⅞ 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. Chart 3 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.

6-20. You can mill the flutes on a tap blank in the following manner. Mount and aline the index centers and set the surface gage to center height.

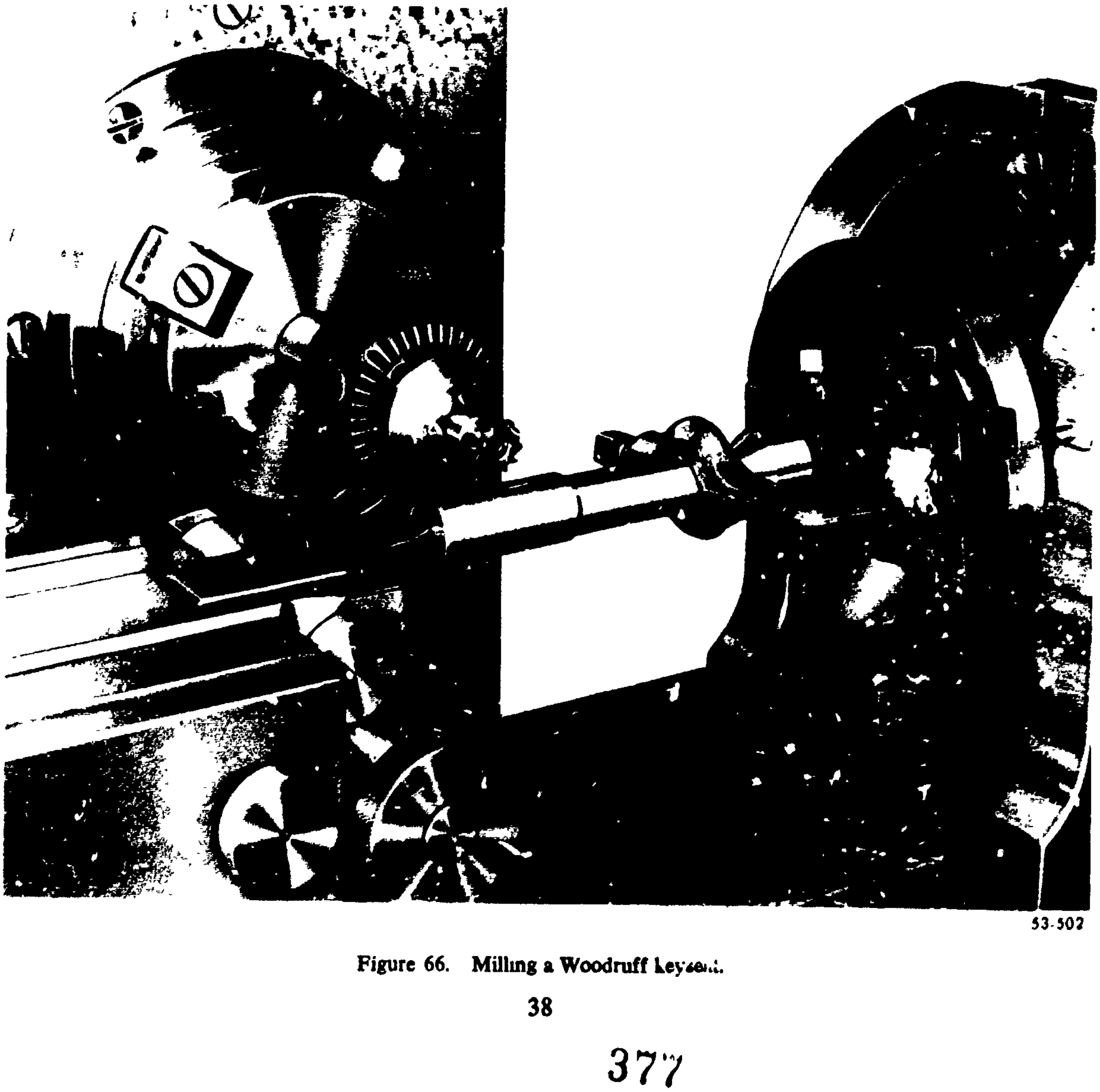


53-501

Figure 65. Woodruff keyseat cutter.

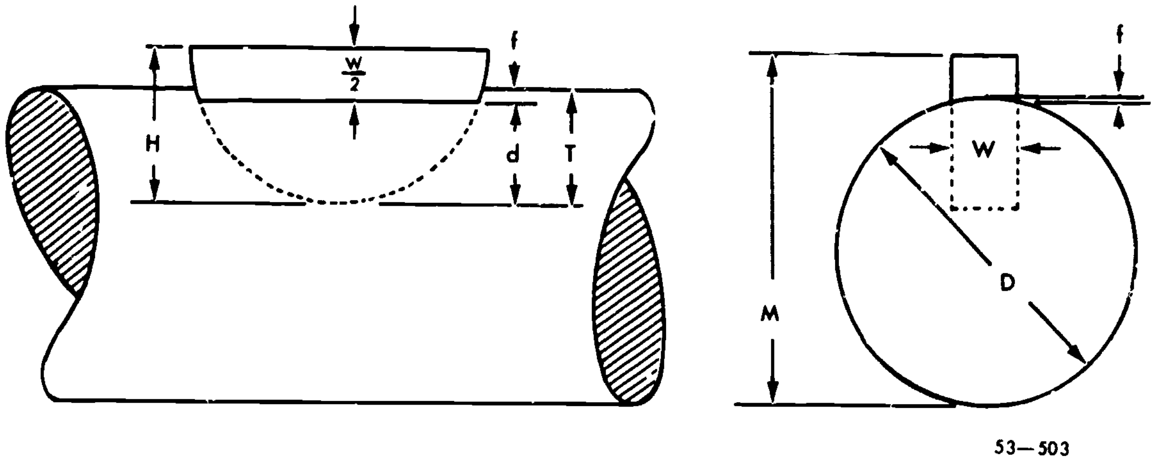
Place the tap blank between the centers with one flat ι the square on the tap shank in a vertical position. You can aline the flat with a square head and blade. Scribe a line on the tap shank.

6-21. Remove the tap blank, place a dog on the shank, and remount the blank between centers Then aline the scribed line with the point of the surface gage scriber. Make sure that the surface



53-502

Figure 66. Milling a Woodruff keyseat.



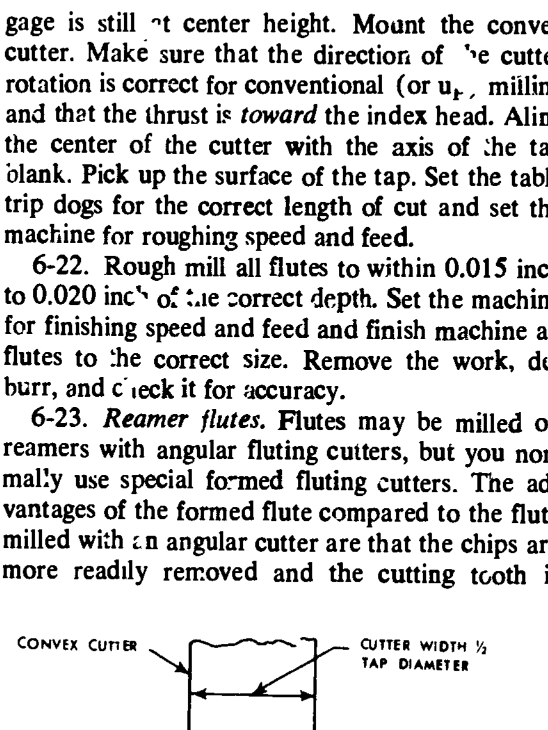
53—503

Figure 67. Dimensions for Woodruff keyseat.

gage is still at center height. Mount the convex cutter. 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. Aline the center of the cutter with the axis of the tap blank. Pick up the surface of the tap. Set the table trip dogs for the correct length of cut and set the machine for roughing speed and feed.

6-22. Rough mill all flutes to within 0.015 inch to 0.020 inch of the correct depth. Set the machine for finishing speed and feed and finish machine all flutes to the correct size. Remove the work, deburr, and check it for accuracy.

6-23. *Reamer flutes.* Flutes may be milled on reamers with angular fluting cutters, but you normally use special formed fluting cutters. The advantages of the formed flute compared to the flute milled with an angular cutter are that the chips are more readily removed and the cutting tooth is



53-504

Figure 68. Hooked tap flutes.

stronger. Also, the tooth is less likely to crack or warp during heat treatment. Formed reamer fluting cutters have a 6° angle on one side and a radius on the other side. The size of the radius depends upon the size of the cutter. Reamer fluting cutters are manufactured in eight sizes. The size of the cutter is identified by a number (1 through 8). Reamers from ⅛ 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 chart 4. You machine reamer teeth with a slight negative rake to help prevent chatter. You obtain the negative rake by positioning the work and cutter slightly ahead of the reamer center, as shown in figure 69. Chart 5 lists the recommended offset for reamers of various sizes. Straight reamer flutes are usually unequally spaced to help prevent chatter. You obtain the unequal spacing by indexing the required amount as each flute is cut. The recommended variation is approximately 2°. Machinists' publications, such as the *Machinery's Handbook,* contain charts that list the number of holes to advance or retard the index crank to machine a given number of flutes when you use a given hole circle. You normally mill the flutes in pairs. After you have machined one flute, index the work one-half revolution and mill the opposite flute.

6-24. The depth of the flute is determined by trial and error. The approximate depth of flute to obtain the recommended width of land is one-eighth the diameter for an eight-fluted reamer, one-sixth the diameter for a six-fluted reamer, etc.

6-25. You can machine the flutes on a hand reamer in the following manner. Mount the reamer blank between centers and the reamer fluting cutter on the arbor. Aline the point of the cutter with the reamer blank axis and just touch the surface of the reamer with the rotating cutter. Re-

CHART 3
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

53-497

move the work blank, and then raise the table a distance equal to the depth of the flute plus one-half the grinding allowance. Rotate the cutter until a tooth is in the vertical position. Then shut off the machine.

6-26. Move the table until the point of the footstock center is alined with the tooth that is in the vertical position. 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 alined with the point of the footstock center. In order 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 obtain the desired amount of offset, then lock it in position. Move the table until the cutter clears the end of the reamer blank. Then remount the blank between the centers. Calculate the indexing required to space the flutes unequally. Set the table feed trip dogs so that the minimum length of the full depth of flute is equal to the length of the reamer teeth. 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.

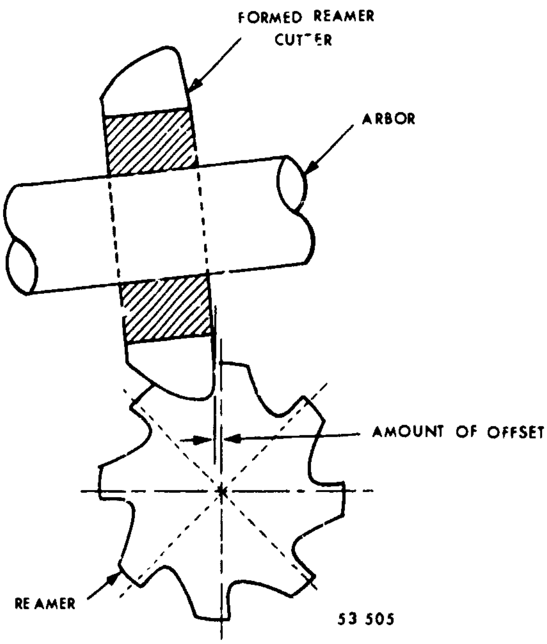


Figure 69 Negative rake tooth.

6-27. *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, provided that 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 32. 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.

7. Drilling, Reaming, and Boring

7-1. Drilling, reaming, and boring are operations that you can do very efficiently on a milling machine. The graduated feed screws make it possi-

CHART 4

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

53-498

ble 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.

7-2. **Drilling and Reaming.** You use the same drills and reamers that you use for drilling and reaming in the lathe and drill press. Drills and reamers are held in the spindle by the same methods that you use to hold straight and taper-shanked end mills. The work may be held in a vise, clamped to the table, held in fixtures or between centers, and in index head chucks, as is done for milling. The speeds used for drilling and reaming are determined in the same manner as for drilling and reaming in the lathe or drill press. The work is fed into the drill or reamer either by hand or power feed. If the cutting tool is held in a horizontal position, the transverse or saddle feed is used.

CHART 5

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

53-499

When the drill or reamer is held in a vertical position, as in a vertical type machine, you use the vertical feed.

7-3. **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 70 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 makes it possible to 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.

7-4. 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 70 that a boring bar holder is not used. The boring bar is inserted into an adapter, and the adapter is fastened in the hole in the adjustable slide. Power for driving the boring bar is transmitted directly through the shank. The elimination of the boring bar holder results in a more rigid boring operation, but the size of the hole that can be bored is more limited than in boring on a lathe or drill press.

7-5. Fly cutters, which we previously discussed, can also be used for boring, as shown in figure 71. A fly cutter is especially useful for boring relatively shallow holes. The cutting tool must be adjusted for each depth of cut.

7-6. The speeds and feeds you should use in boring on a milling machine are comparable to those you would use in boring on a lathe or drill press and depend upon the same factors: hardness of the metal, kind of metal the cutting tool is made of, 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

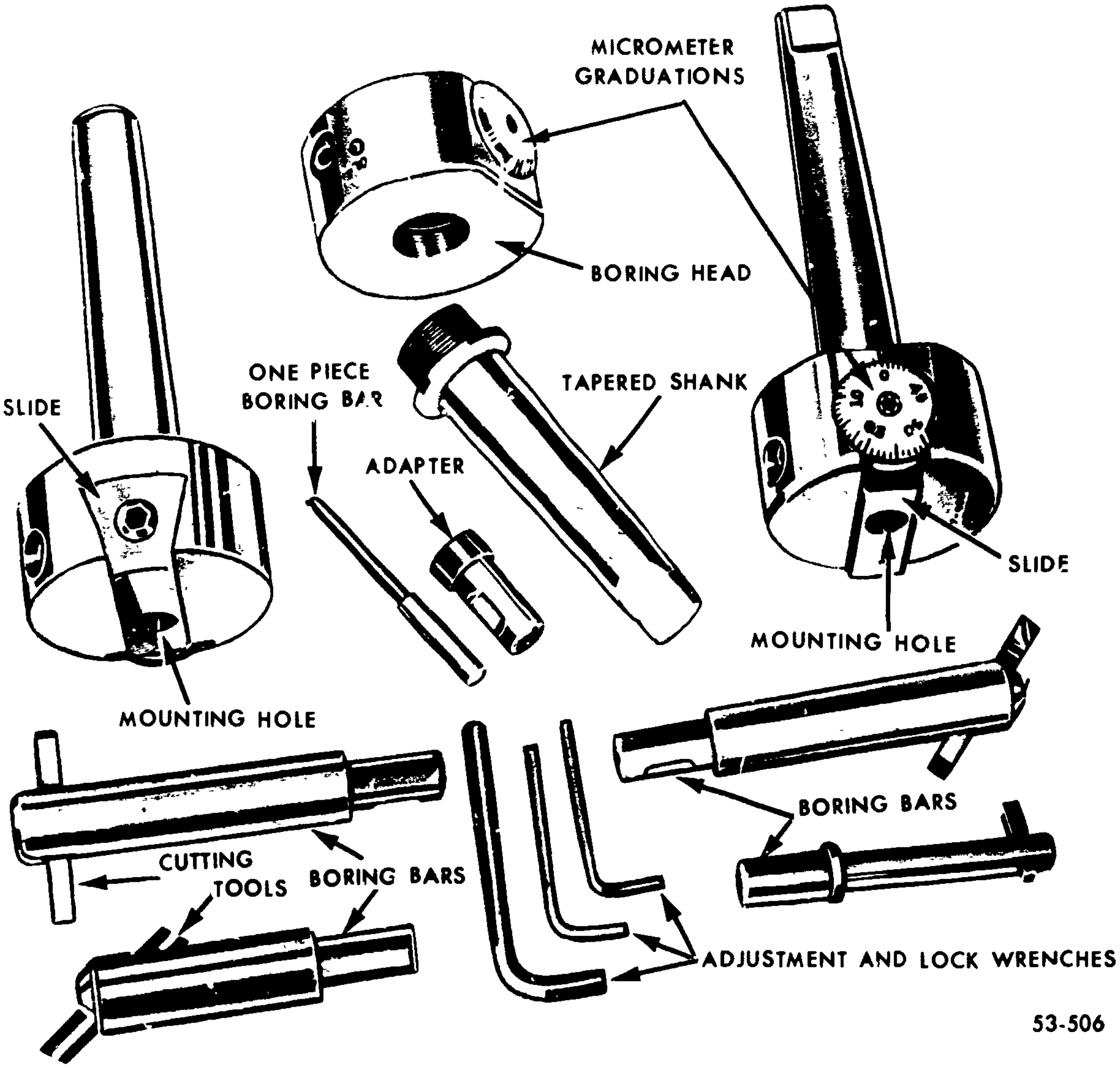


Figure 70. Offset boring head and tools.

all of these reasons, you must guard against operating at too great a speed or vibration will occur.

8. Milling Machine Attachments

8-1. Many attachments have been developed that increase the number of jobs a milling machine can do or that make such jobs easier to do. For instance, by using a vertical spindle attachment, you can convert the horizontal spindle to a vertical spindle machine and swivel the cutter to any position in the vertical plane, as shown in figure 72. By using a universal milling attachment, you can swivel the cutter to any position in both the vertical and horizontal planes. By using a high-speed universal attachment, you can perform milling operations at higher speeds than those for which the machine was designed. These attachments will enable you to do more easily jobs which would otherwise be very complex.

8-2. **High-Speed Universal Attachment.** This attachment is clamped to the machine and is driven by the milling machine spindle, as you can see in figure 73. The attachment spindle head and cutter can be swiveled 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 in order to maintain an efficient cutting action.

8-3. **Circular Milling Attachment.** This attachment, shown in figure 74, 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

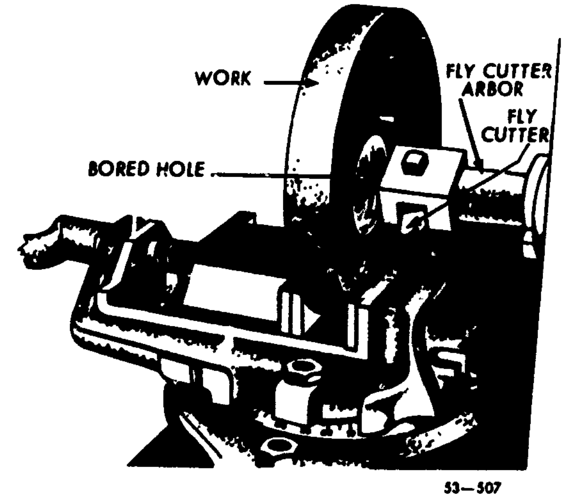


Figure 71. Boring with a fly cutter.

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.

8-4. **Rack Milling Attachment.** The rack milling attachment, shown in figure 75, 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 at-

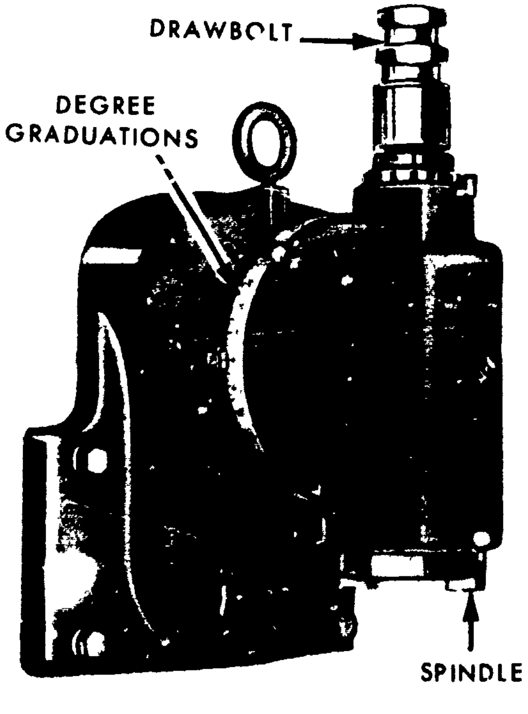


Figure 72. Vertical milling attachment.

tachment parallel to the table T-slots. An indexing arrangement is used to space the rack teeth quickly and accurately.

8-5. **Right-Angle Plate.** The right-angle plate, shown in figure 76, is attached to the table. The right-angle slot permits mounting the index head so that the axis of the head is parallel to the milling machine spindle. With this attachment you can make work setups 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.

8-6. **Raising Block.** Raising blocks, shown in figure 77, are heavy-duty parallels, which usually come in matched pairs. They are mounted on the table and the index head is mounted on the blocks.

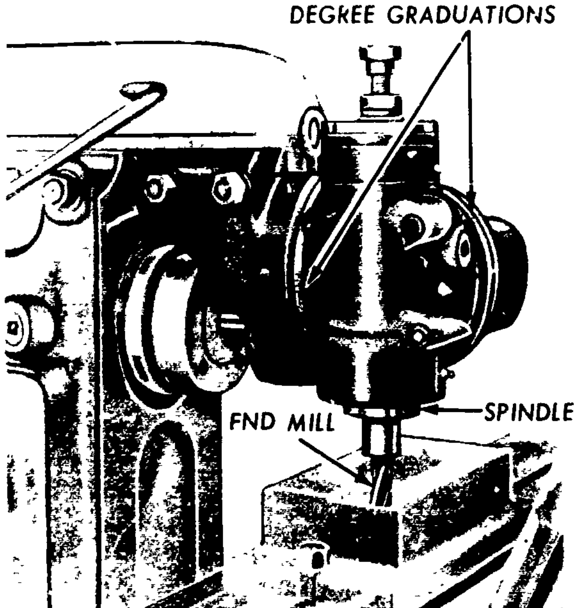


Figure 73. High-speed universal milling attachment.

This arrangement raises the index head and makes it possible to swing the head through a greater range to mill larger work.

8-7. **Toolmaker's Knee.** The toolmaker's knee, shown in figure 78, is a simple but useful attachment for setting up angular work, not only for milling but for shaper, drill press, and grinder operations as well. 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.

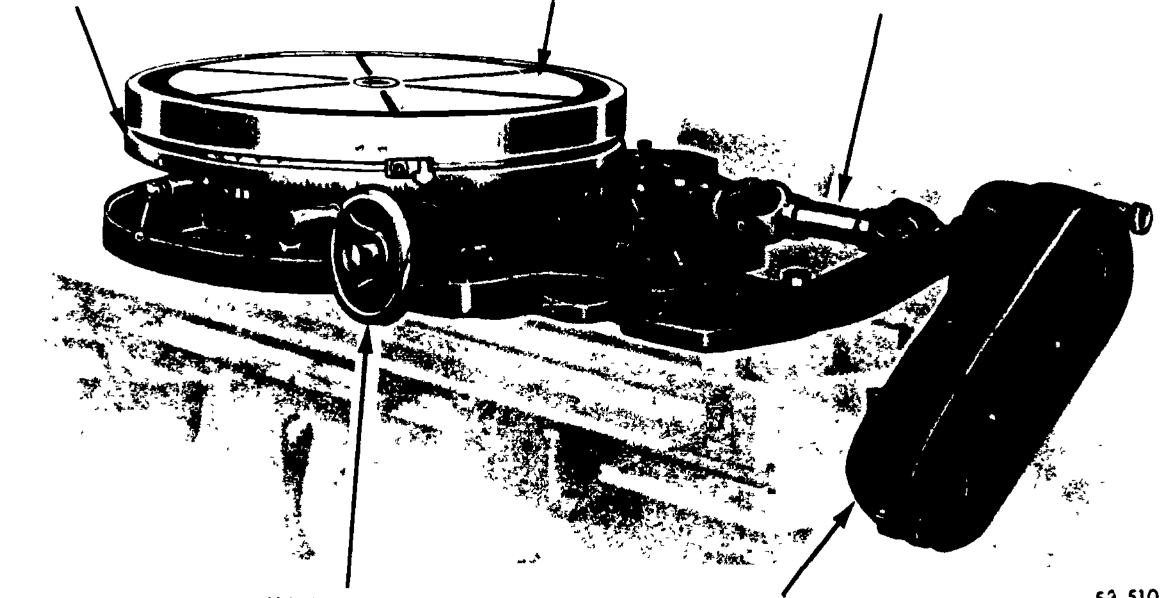


Figure 74. Circular milling attachment.

9. Gearing and Gear Cutting Practices

9-1. Special gear cutting machines are used by industry to manufacture grant quantities of gears. However, a single gear or a relatively small number of gears may be cut on a milling machine. Many different types of gears are used to transmit motion, as shown in figure 79. For now, we will restrict our discussion to the spur gear. The principles of spur gearing apply to a great extent to other types of gearing.

9-2. Spur gears have teeth that are parallel to the axis of the gear blank, as shown in figure 80. Compared to helical gears, spur gears are relatively simple to machine. Spur gears are noisier than helical gears and not as smooth running. You will usually machine spur gears with special form cutters called gear cutters. The cutter has the form of the space between the gear teeth, as shown in figure 80. The most important dimension, as far as milling a spur gear is concerned, is the pitch circle. However, all the dimensions of a gear are related to each other. You must know the other dimensions, or how to calculate them, in order to machine an accurate gear.

9-3. **Spur Gear Terms.** To understand spur gearing and to be able to machine a spur gear blank, you must have a working knowledge of gearing terms. Many spur gear terms apply also to other types of gears. Figure 8' il help you to understand these terms. Refer to it frequently.

NOTE: the smaller of two gears in mesh, or the smallest gear in a gear train, is called a *pinion,* or *pinion gear.*

a. Pitch circle (PC) is a term that is commonly used to represent the pitch cylinders of a pair of gears. The pitch cylinders are the imaginary cylinders that roll one on the other when the gears are in mesh.

b. Number of teeth (N) refers to the teeth on the periphery of the gear. They can be easily counted on the sample gear or the drawing.

c. Pitch diameter (D) is the diameter of the pitch circle.

d. Diametral pitch (P) is the number of teeth per inch of pitch diameter. The diametral pitch determines the size of the gear teeth. Mating gears must have the same diametral pitch.

e. Circular pitch (CP) is the distance from any point on one tooth to a corresponding point on an adjacent tooth measured on the pitch circle.

f. Addendum (a) is the height of that portion of gear tooth that extends above the pitch circle.

g. Clearance (c) is the radial distance between the top of the tooth and the bottom of the mating tooth space when two gears are in mesh.

h. Dedendum (d) is the depth of that portion of gear tooth that extends below the pitch circle.

i. Outside diameter (OD) is the distance from the top of one tooth to the top of the tooth diametrically opposite.

j. Center distance (CD) is the distance from the center of one gear to the center of the mating gear when the gears are in mesh, or the distance between the center of the shafts upon which the gears are mounted.

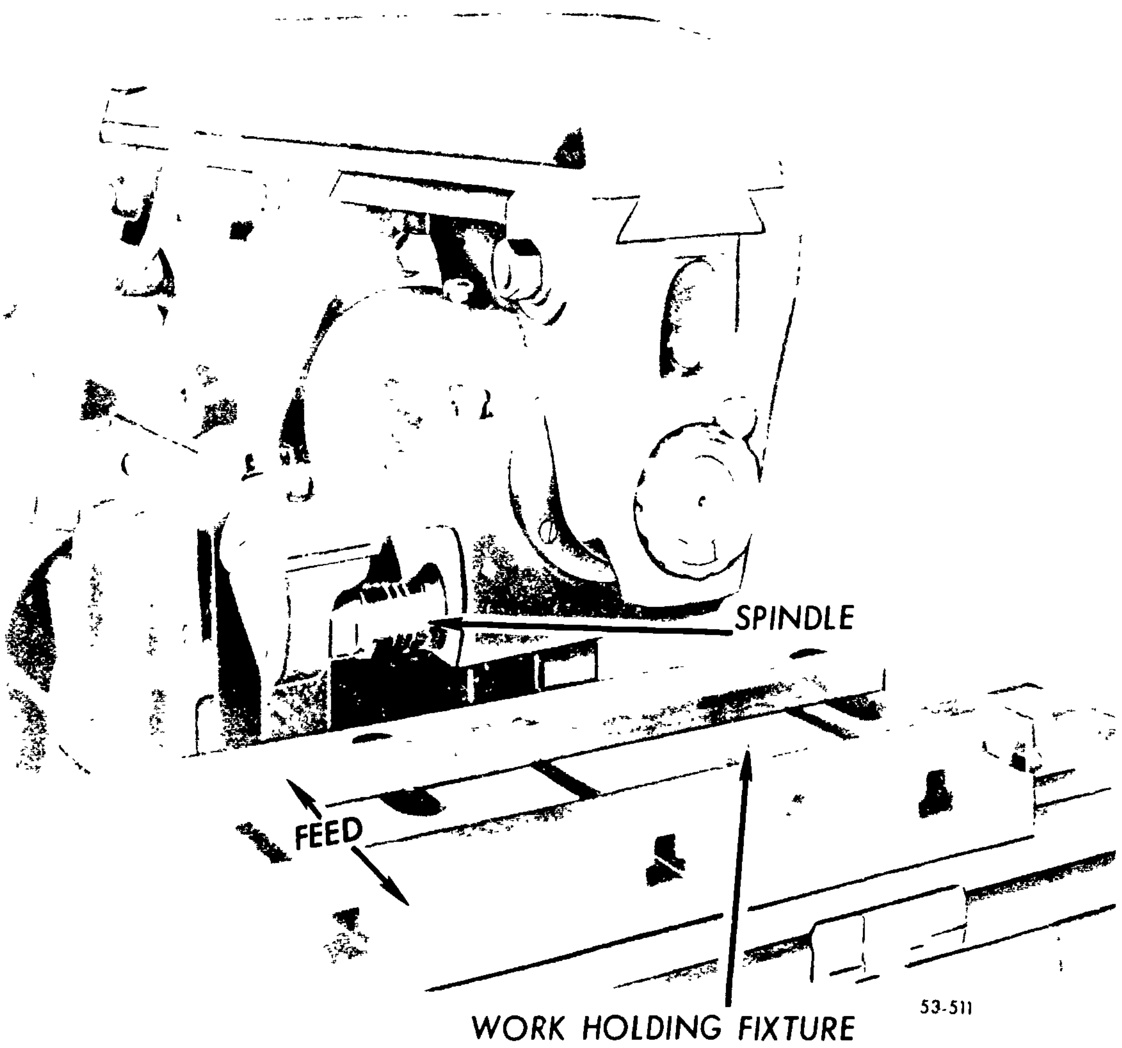


Figure 75. Rack milling attachment.

k. Velocity ratio (VR) is the ratio between the speed of rotation of the mating gears. It is also the ratio between the number of teeth on the driven gear and the driver gear. It is also the ratio of the pitch diameter of the larger gear and the driver gear. It is also the ratio of the pitch diameter of the larger gear to the pitch diameter of the pinion. If the ratio between two mating gears is 2 to 1, it is written as 2:1. More information on gear formulas can be found in various technical publications, such as the *Machinery's Handbook*.

Figure 76. Right-angle plate.

9-4. **Spur Gear Formulas.** You must know the dimensions of a gear in order to machine it. The following formulas will provide you with the information you need. You are already familiar with some of the terms, while some of them will be new to you. Refer to figures 81 and 82 as you study the following definitions:

a. Circular pitch (cp)—The distance from a point on one tooth to a corresponding point on an adjacent tooth measured on the pitch circle.

$$cp = \frac{3.1416}{P}$$

b. Diametral pitch (P)—The number of teeth (N) per inch of pitch diameter.

$$P = \frac{N}{D}, \text{ or } P = \frac{3.1416}{cp}$$

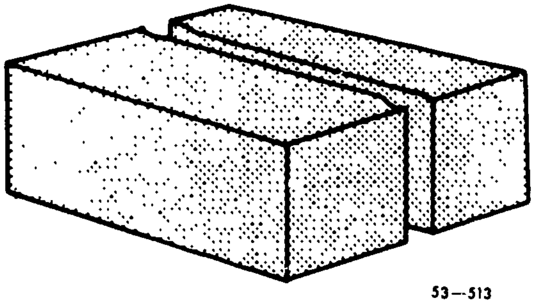


Figure 77. Raising blocks.

c. Pitch diameter (D)—The diameter of the pitch circle.

$$D = \frac{N}{P}$$

d. Addendum (a)—The distance from the pitch line to the top of the tooth.

$$a = \frac{1}{P}$$

e. Dedendum (d)—The distance from the pitch line to the root of the tooth.

$$d = \frac{1.157}{P}$$

f. Clearance (c)—The amount the tooth space is cut deeper than the working depth.

$$C = \frac{0.157}{P}$$

g. Whole depth (W)—The working depth plus the clearance.

$$W = 2a + c, \text{ or } W = a + d, \text{ or } W = \frac{2.157}{P}$$

h. Working depth (Wd)—The depth in the space that the tooth of the mating gear extends.

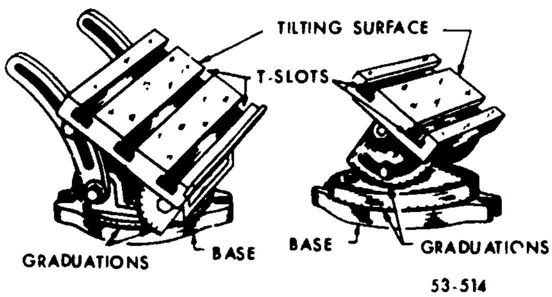


Figure 78. Toolmaker's knees.

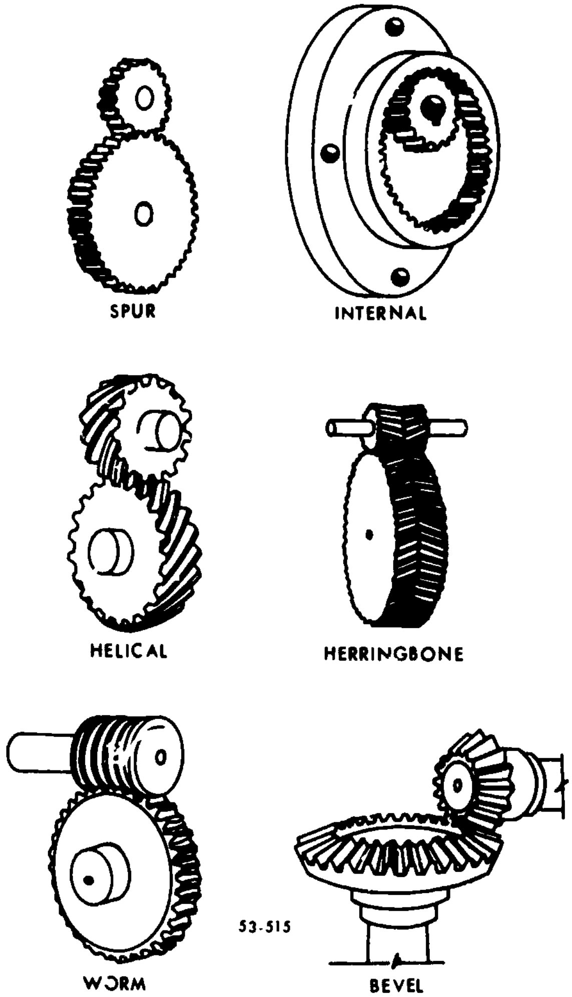


Figure 79. Types of gears.

$$Wd = \frac{2}{P}$$

i. Tooth thickness (T)—The thickness of the tooth measured on the pitch circle.

$$T = \frac{ep}{2}$$

j. Chordal thickness (CT)—The thickness of the tooth on the chord that connects the ends of the pitch circle segment.

$$CT = D \times \text{sine } \theta$$

k. Outside diameter (OD)—The largest diameter of the gear.

$$OD = D + 2a, \text{ or } OD = \frac{N + 2}{P}$$

l. Height of arc (H)—The distance from the chord of the pitch circle segment to the arc formed by the segment.

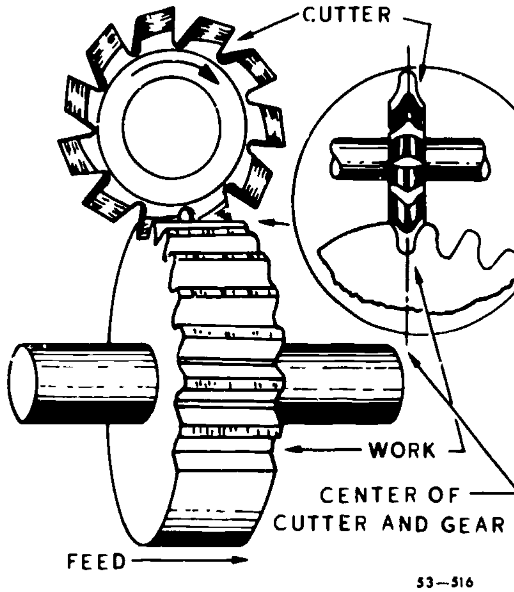


Figure 80. Milling a spur gear.

$$H = D \times \frac{(1 - \text{cosine } \theta)}{2}$$

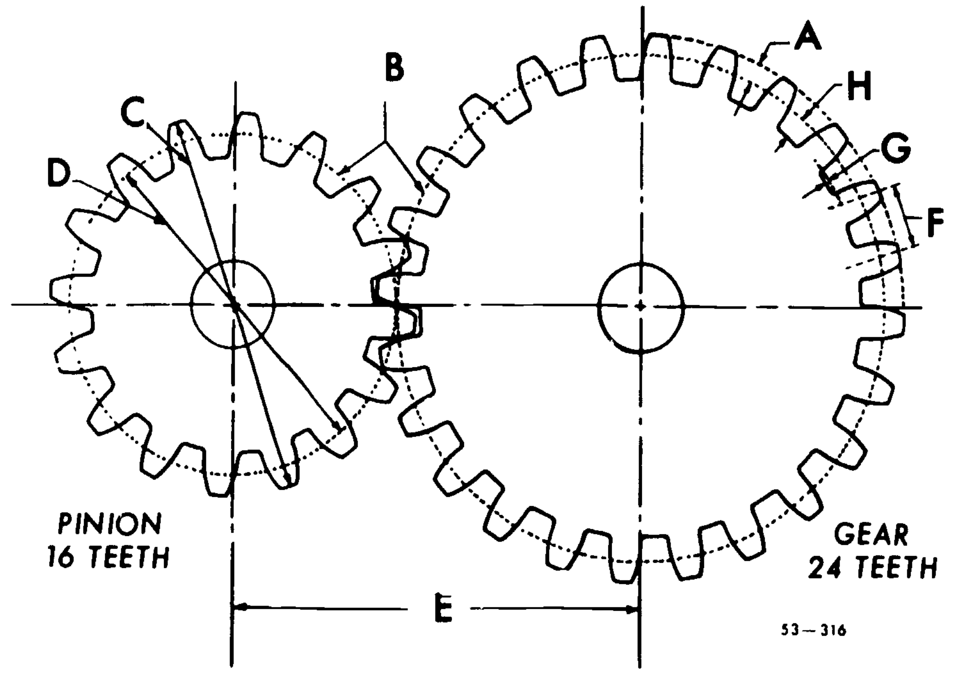
m. Corrected addendum (a′)—The distance from the chord of the pitch circle segment to the top of the teeth.

$$a' = a + H$$

n. Theta (θ)—The angle whose value is one-fourth of the angle subtended by the circular pitch.

$$\theta = \frac{90^\circ}{N}$$

9-5. **Machining a Spur Gear.** You must know the diametral pitch and the number of the teeth on the gear that you are cutting in order to select the proper cutter. Eight standard cutters are manufactured for each diametral pitch. Each cutter is identified by a number. The cutter number and the number of teeth that the cutters machine are shown in chart 6. A cutter is shaped to accurately produce the fewest number of teeth in its range, while it is nearly correct for the other numbers in the range. A series of cutters, sizes 1½ through 7½, which fall between the standard cutter sizes, are available when the shape of the gear teeth must be more nearly perfect. You machine the teeth to size by picking up the surface of the gear blank with the cutter and machining the tooth space to the whole depth of the gear tooth. When more precise dimensions are required, you can measure the tooth thickness with a vernier gear tooth caliper, such as the one shown in figure 82. The gear tooth caliper measures the thickness of the gear tooth at the pitch circle. Since the caliper measures a chord of the pitch circle, you must set the vertical scale for the corrected addendum (a′), as shown in figure 82. The corrected addendum allows for the height of the arc (H). After setting the vertical



A. ADDENDUM
B. PITCH CIRCLES
C. OUTSIDE DIAMETER
D. PITCH DIAMETER
E. CENTER DISTANCE (RATIO OF GEARS 1½ TO 1)
F. CIRCULAR PITCH
G. CLEARANCE
H. DEDENDUM

Figure 81. Spur gear terms.

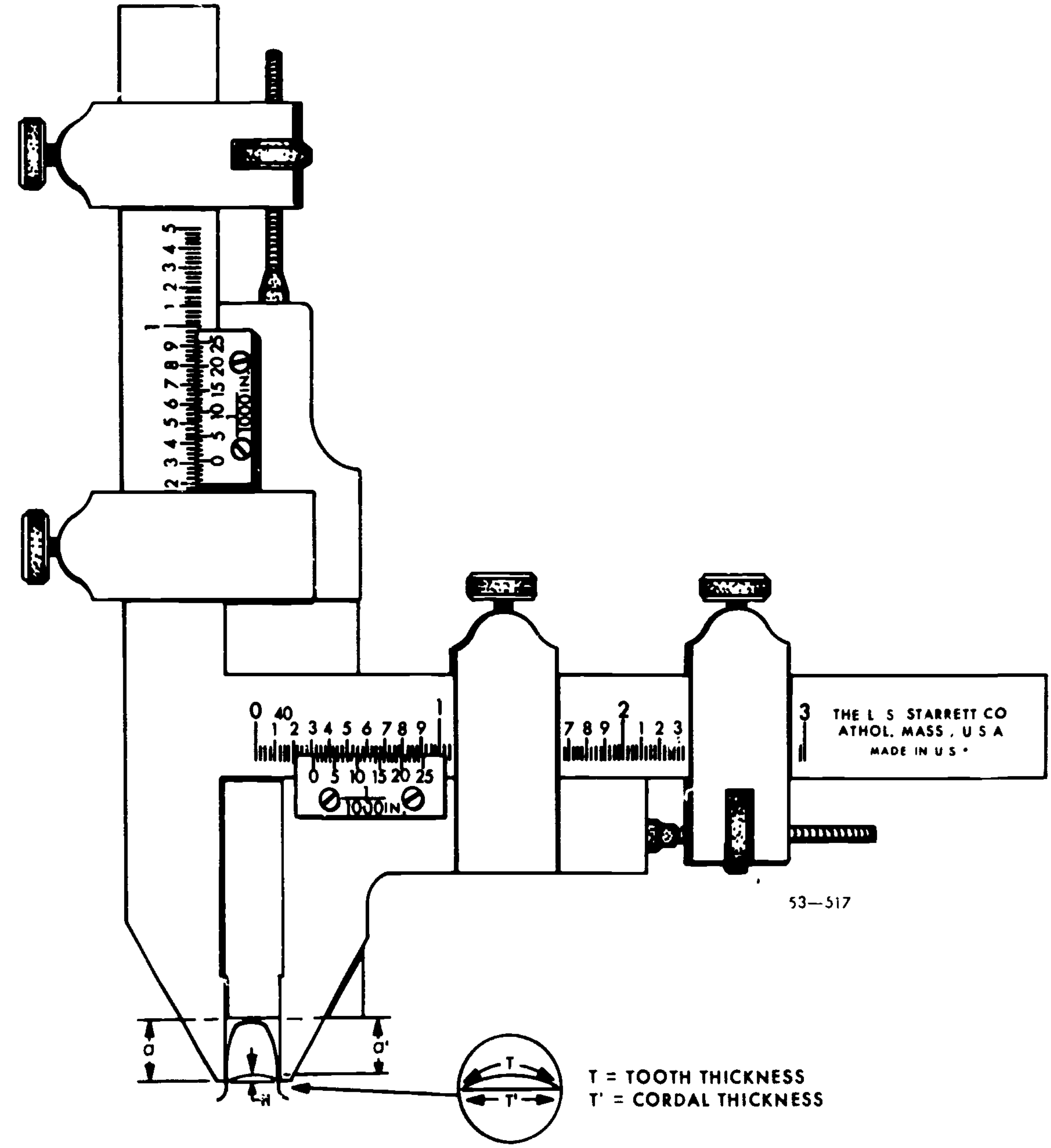


Figure 82. Vernier gear tooth caliper.

scale for the corrected addendum, you measure the thickness of the tooth with the horizontal scale. You read the vernier scales in the same manner that you read a vernier caliper except that the subdivisions represent 0.020 inch instead of the usual 0.025 inch. Take cuts on adjacent sides of a tooth until you obtain the correct thickness and machine the remaining teeth with the same setting.

9-6. Assume that you are to machine a gear blank for a 24-tooth 6-diametral-pitch spur gear. The calculations necessary to machine the teeth on a blank are as follows:

a. Number of teeth (N) (given) 24.

b. Diametral pitch (P) (given) 6.

c. Outside diameter (OD).

$$OD = \frac{N + 2}{P} = \frac{24 + 2}{6} = 4.333 \text{ inches}$$

d. Pitch diameter (D).

$$D = \frac{N}{P} = \frac{24}{6} = 4 \text{ inches}$$

e. Addendum (a).

$$a = \frac{1}{P} = \frac{1}{6} = 0.1666 \text{ inch}$$

CHART 6

INVOLUTE GEAR CUTTER NUMBERS

Cutter number	Number of gear teeth
1	From 135 to rack
2	55 to 134
3	35 to 54
4	26 to 34
5	21 to 25
6	17 to 20
7	14 to 16
8	12 to 13

53 500

f. Working depth (Wd).

$$Wd = 2a = 0.3332 \text{ inch}$$

g. Circular pitch (cp).

$$cp = \frac{3.1416}{6} = 0.5236 \text{ inch}$$

h. Tooth thickness (T).

$$T = \frac{cp}{2} = \frac{0.5236}{2} = 0.2618 \text{ inch}$$

i. Whole depth of tooth (W).

$$W = \frac{2.157}{P} = 0.3595 \text{ inch}$$

j. Angle theta (θ).

$$\theta = \frac{90^\circ}{n} = \frac{90^\circ}{24} = 3^\circ\, 45'$$

k. Cosine θ (trig table) = 0.99786.

l. Height of arc (H).

$$H = \frac{D \times (1 - \text{cosine } \theta)}{2} = 6 \times \frac{(1 - 0.99786)}{2}$$

$$= \frac{6 \times 0.00214}{2} = \frac{0.01284}{2} = 0.00642 \text{ inch}$$

m. Corrected addendum (a′).

$$a' = a + H = 0.1666 + 0.0642 + 0.2308 \text{ inch}$$

n. Sine θ (trig table) = 0.06540.

o. Chordal thickness (CT).

$$CT = D \times \text{sine } \theta = 6 \times 0.06540 = 0.39240 \text{ inch.}$$

p. Cutter number (24 teeth) 6 diametral pitch, number 5 (21 to 25).

q. Indexing $\frac{40}{N} = \frac{40}{24} = 1\frac{2}{3}$ = 1 turn and 12 holes in an 18-hole circle.

9-7. If you are required to manufacture a spur gear that will have 24 teeth and a 6-inch diametral pitch, you could do it in the following manner. If you are to use a universal milling machine, make sure that the table is set for zero taper. Get someone to help you mount the index head on the table. Then mount the footstock and aline the centers. Press the gear blank on a mandrel. Position the surface gage scriber point at center height and set it aside, taking care not to disturb the setting. Mount the gear blank and the mandrel between centers. Make sure that the thrust of the cutter is toward the larger diameter of the mandrel.

9-8. Before proceeding further, you should check the concentricity of the gear blank with a dial indicator. Set up the indicator with the plunger contacting the circumference of the gear blank. Then rotate the blank and mandrel by hand. Using the centers as pivot points, find the location midway between the maximum and minimum runout. This location is then used as the reference point for all measurements. The teeth on either side of this point will vary slightly in depth, but this variation will not normally affect the use of the gear.

9-9. Cover the circumference of the gear blank near the reference point with layout dye. Then turn the index crank, rotating the work enough for the reference point to be at center height. Using the already set surface gage, scribe a horizontal line across the gear blank circumference. Index the blank a quarter turn so that the reference line is at top dead center. Then lock the index head spindle.

9-10. Mount the correct arbor and gear cutter and place the cutter over the gear blank. Move the saddle until the cutter is alined by eye over the scribed line. Set the speed and feed for a roughing cut. The feed rate may have to be reduced, as the method of mounting the blank on the mandrel may cause it to slip when a heavy feed is used. While the cutter is rotating, slowly raise the table with hand power until the work lightly touches the cutter. Set the vertical feed dial to ZERO. Then lower the work from the cutter so that you can determine on which side of the layout line the cutter made contact. If the cutter is not quite on center, move the saddle in the required direction until it is. Until you become more experienced, it may take several tries to get the cutter centered.

9-11. Once the work is centered under the cutter, move the work clear of the cutter, and raise the knee to the depth of the required roughing cut. Lock the knee of the milling machine in this position. Then slowly feed the rotating cutter into the gear blank just enough to nick the blank. Back the cutter off from the blank; then index for the next tooth space, and nick the blank again. Continue this operation until all the tooth spaces have been nicked. After shutting off the machine, you should count the *spaces between* the nicks to be sure the proper number of teeth will be cut. When you are sure that everything is correct, then take roughing cuts in each tooth space, leaving from .015 to .020 for finishing cuts.

9-12. When the roughing cuts are complete, set the machine for finishing speeds and feeds, and raise the knee and work to a few thousandths less than the finishing depth of cut. With this setting, machine two adjacent tooth spaces as close as possible to the reference point. Then measure the chordal thickness, using a vernier gear tooth caliper. Make the necessary corrections to obtain the correct depth of cut. Take light trial cuts in adjacent teeth spaces until you obtain the correct chordal thickness. Once you have the correct setting, mill the remainder of the teeth at this setting. Finally, check the gear for accuracy before removing it from the machine.

Grinding Machine Work

GRINDING MACHINES ARE used to dress, shape, or finish work surfaces by means of a rotating abrasive wheel. You perform two types of grinding in a machine shop. Precision grinding is grinding work on a machine to close tolerances. Hand grinding is holding the wheel to the work or the work to the wheel and grinding by hand manipulation. Grinding lathe and shaper tools, which we discussed in Chapter 1 of Volume 2, is an example of hand grinding. This chapter will cover basic grinding information and operations which can be performed on surface grinders, cylindrical grinders, and tool and cutter grinders.

10. Basic Grinding Information

10-1. You can do a wide variety of work with a few general purpose grinding wheels. It has been estimated that there are 12,000 possible combinations of abrasive, bond, grade, and grain size (grit) for every wheel. It is, therefore, more economical in most machine shops to do grinding work with a relatively small number of general purpose wheels. In this section we will discuss grinding wheels, the preparation of grinding wheels, and speeds.

10-2. **Grinding Wheels.** You cannot base your selection of a grinding wheel solely on the kind of material that you want to grind. Selecting a wheel which will grind efficiently depends upon these additional factors:

(1) Amount of stock to be removed.
(2) Accuracy and finish required.
(3) Area of contact between the wheel and the work.
(4) Type and condition of the wheel.
(5) Nature of the operation.
(6) Work speed.
(7) Wheel speed.

We will discuss these factors wherever they are applicable throughout the chapter. We will discuss, specifically, in this section: abrasives, bond, grain, area of contact, shape, and markings.

10-3. *Abrasives.* When you select a grinding wheel for a particular operation you must consider the qualities of the abrasive. The most important are the hardness, toughness, and the nature of the fracture. Hardness is the ability of abrasive to scratch or cut another substance and resist being scratched or cut. The diamond, as you may know, is the hardest substance. You use it to dress and true grinding wheels. The two principal abrasives in grinding wheels are aluminum oxide, which is tough and not easily fractured, and silicon carbide, which is harder and more brittle. You should use an aluminum oxide wheel for most grinding operations involving high tensile strength materials, such as carbon steels, alloy steels, high-speed steels, etc. Use silicon carbide for materials that are hard and brittle or of low tensile strength, such as cast iron. The grinding of nonferrous metals is not recommended as they tend to clog the pores of the grinding wheel.

10-4. *Bond.* Bond is the material that holds the abrasive particles together and supports them as they cut. The "hardness" of a grinding wheel with a given type of bond depends upon the amount of bond. The greater the amount and strength of the bond the harder the grinding wheel will be. Abrasives are held together by one of five bonds: vitrified, silicate, shellac, resinoid, and rubber. Three of every four wheels have a vitrified bond. It is used with both aluminum oxide and silicon carbide abrasives. Vitrified wheels are limited to slower wheel speeds, but they are strong enough for heavy-duty work. You can use vitrified wheels for the rapid removal of stock and for precision grinding operations. Silicate bonds release grains readily. They have a mild cutting action and dissipate heat easily and are thus excellent for grinding edged cutting tools. Shellac bonds are used in wheels to produce a high finish and for light-duty operations, such as cutting off. They are not suitable for heavy-duty grinding. Resinoid- and rubber-bonded wheels are less brittle than vitrified wheels. Thin wheels ($\frac{1}{32}$ inch and less) and

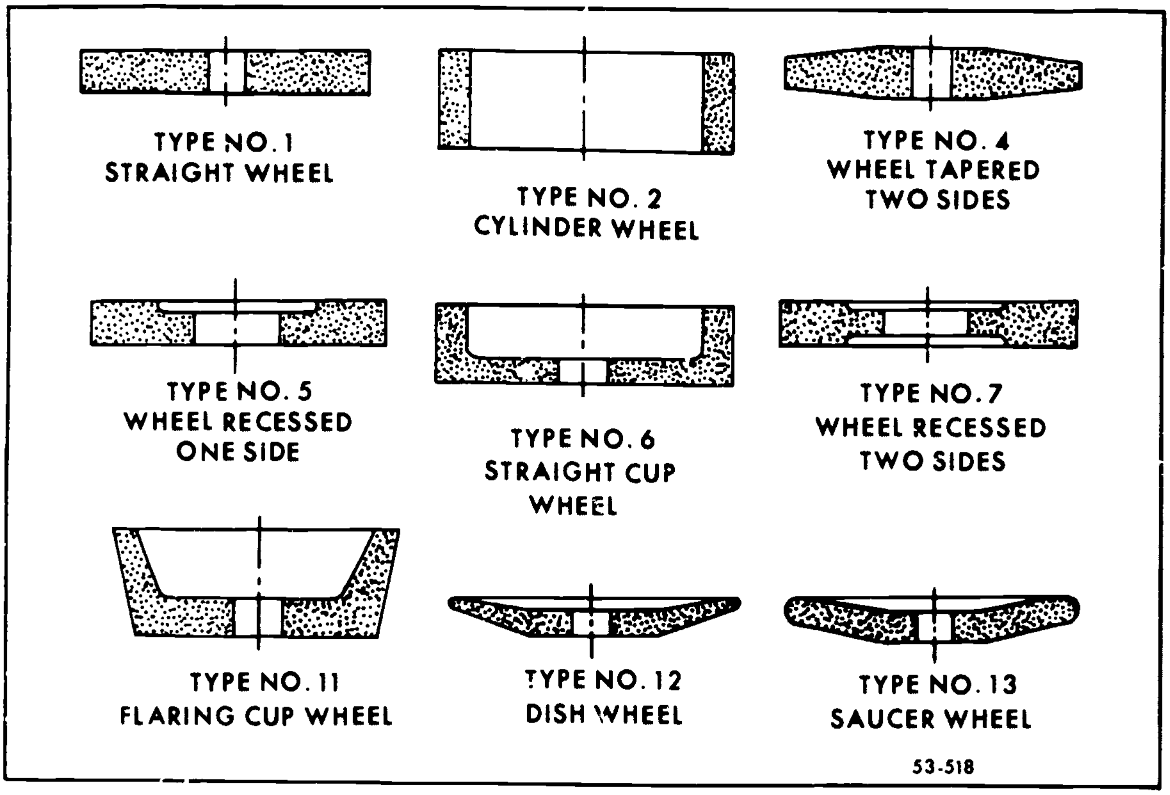


Figure 83. Standard grinding wheel shapes.

wheels subject to lateral strains by the spindle are usually rubber bonded.

10-5. *Grain*. The size of the abrasive grain in the wheel you want to use depends upon the amount of material to be removed, the finish desired, and the physical properties of the material. You should use a coarse-grained wheel for rapid removal of stock and a fine-grained wheel to produce a fine finish. For soft, ductile materials you should use a wheel with widely spaced coarse grains because they penetrate deeply. They also provide clearance between the grains for large chips. Use a closely spaced, fine-grained wheel to grind hard, brittle materials.

10-6. *Area of contact*. You must consider the area of contact between the wheel and the work when you select a grinding wheel. For instance, you generally use a straight wheel in cylindrical grinding, as shown in figure 83, because the area of contact is small. You will need a relatively fine-grained, closely spaced wheel to bring a greater number of cutting edges to the work. To prevent rapid wheel wear, you should use a medium hard grade wheel. However, when you grind a surface with the rim of a cylinder wheel, as shown in figure 83, the area of contact is quite large. You will then need a widely spaced coarse-grained wheel to distribute a smaller number of cutting edges to the work. Also, the grade should be soft to permit the grains to be torn out and released as they become dull.

10-7. *Shape*. There are many different shapes of wheels. Figure 83 shows some of the more common standard shapes. You can do practically all the grinding that will be required in a machine shop with these shapes of wheels. Grinding wheels also come in a wide variety of wheel faces, as shown in figure 84. Standard wheel shapes are identified by a number and wheel faces by a letter. Figure 85 illustrates some common shapes of mounted wheels.

10-8. *Markings*. Every grinding wheel is marked by the manufacturer with a stencil or a small tag. The manufacturers have worked out a standard system of markings, shown in chart 7. The chart is self-explanatory and should be studied for future reference. You should note the information contained in the various positions:

(1) Kind of abrasive.
(2) Number of grain size.
(3) Grade.
(4) Structure.
(5) Bond.
(6) Manufacturer's modification number.

As an example, let us use a wheel marked A60–L6–V11. The A refers to the abrasive, which in this case, is aluminum oxide. The 60 represents

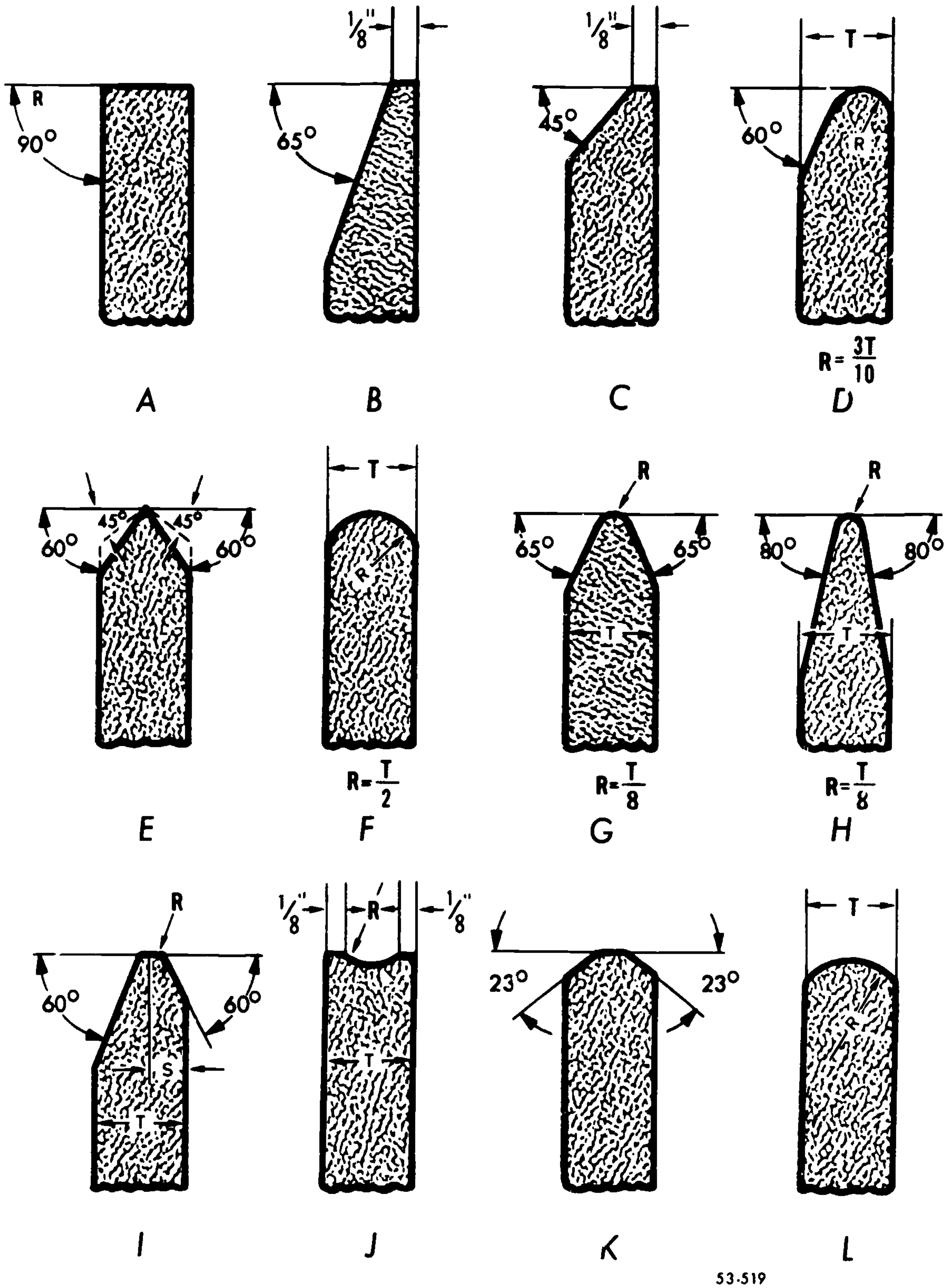


Figure 84. Standard grinding wheel faces.

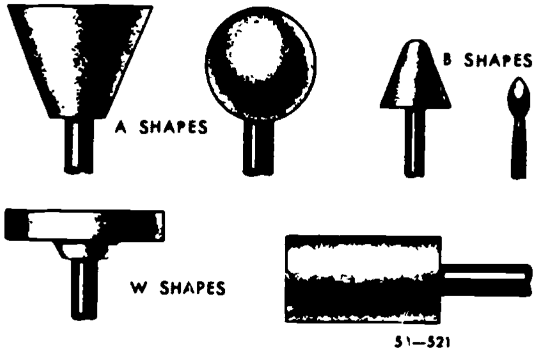


Figure 85. Mounted wheels.

the grain size. The letter L is the grade, which is the degree of hardness. The 6 refers to the structure, which is medium for this wheel. The bond type is vitrified, as indicated by the V, while the 11 is a manufacturer's code.

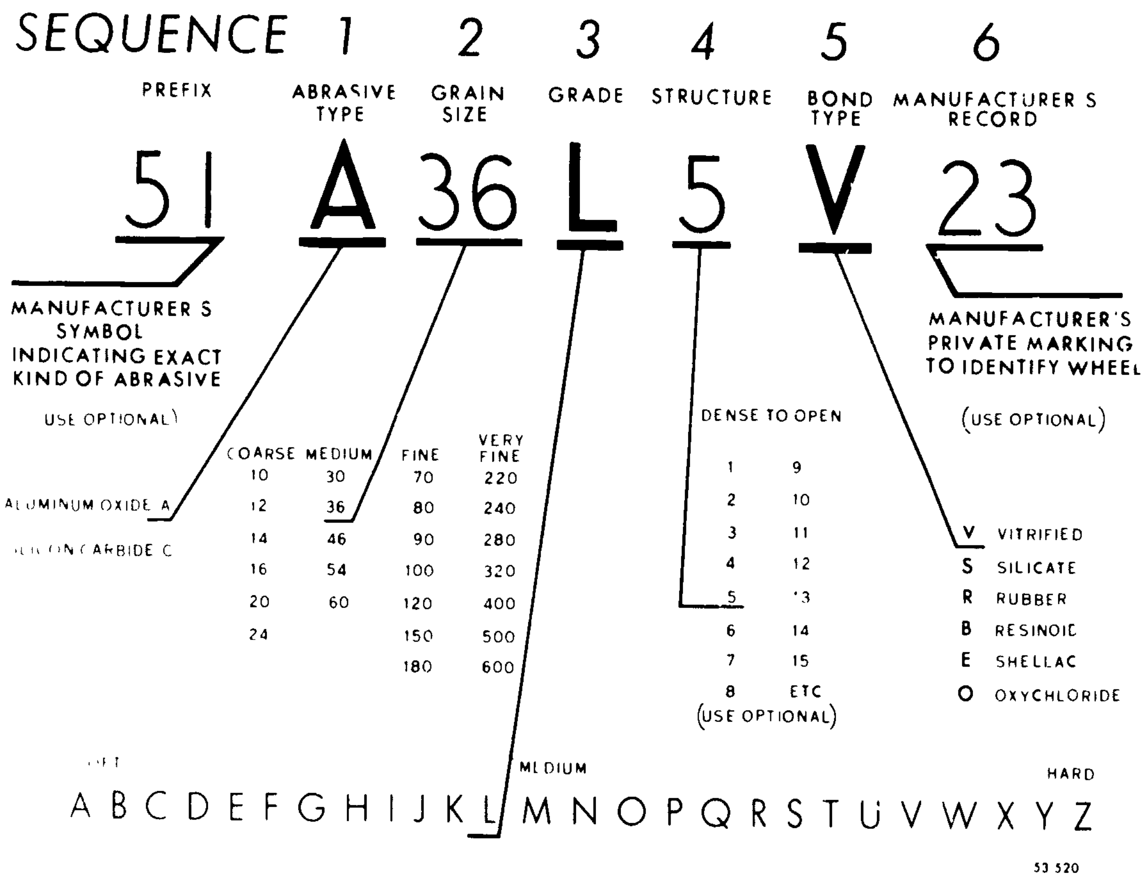
10-9. **Preparation of Grinding Wheels.** Prior to a grinding operation, you must prepare the wheel for grinding. The preparation of a grinding wheel involves inspection, balancing, mounting, and dressing and truing.

10-10. *Inspection.* When a wheel is received in the shop or removed from storage, you should inspect it closely for damage and cracks. Check a small wheel by suspending it on one finger or with a piece of string. Tap it gently with a light nonmetallic instrument, such as the handle of a screwdriver, as shown in figure 86. Check a larger wheel by striking it with a wooden mallet. If the wheel does not emit a clear ring, examine it for cracks. Discard the wheel if it is cracked. All wheels do not produce the same tone when they are rung. A low tone does not necessarily indicate a cracked wheel. Wheels are often filled with various resins and greases to modify their cutting action. Resin or grease deadens the tone. Vitrified and silicate wheels emit a clear metallic ring. Resinoid-, rubber-, and shellac-bonded wheels emit a tone that is less clear. You can readily identify the sound of a cracked wheel.

10-11. Carefully inspect a wheel that has been in storage before you use it. All wheels should be stored in a dry place. Straight wheels 6 inches in diameter and larger should be placed on edge in such a way that they cannot tip or roll. Store wheels less than 6 inches in diameter in racks, as

CHART 7

GRINDING WHEEL MARKINGS



53-522

Figure 86. Checking for cracks.

illustrated in figure 87. Lay thin wheels, regardless of size and bond, on a flat surface to prevent warping. Flaring cup and dish wheels should be stacked flat, with cushioning material between them. Straight cup wheels can be stored either on edge or flat.

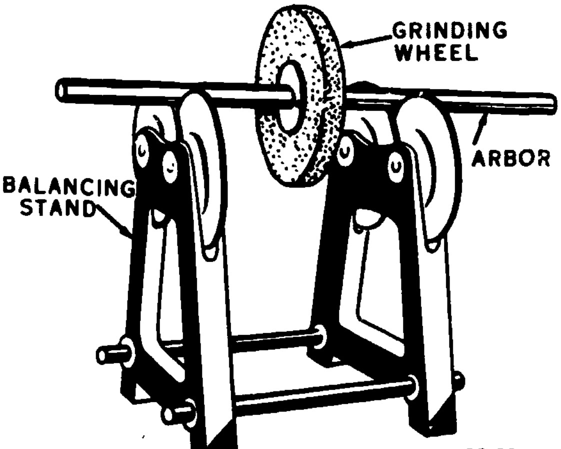
10-12. *Balancing.* A grinding wheel under 12 inches in diameter seldom needs balancing. Larger wheels, especially those that are to be used in precision grinding, must be balanced. To balance a wheel, you mount it on an arbor and allow it to slowly revolve on a balancing stand, as shown in figure 88. The wheel will come to rest with the heaviest part down. You balance the wheel by shifting the position of the weights, either two or four in number, in a circular groove cut in the wheel mount bushing, as shown in figure 89. The weights are secured by a jam screw. If a wheel does not have weights, you can balance it by carefully chiseling out some of the wheel next to the bushing and filling the space with lead. After mounting the wheel, recheck the balance with the wheel rotating. Do not permit wheels with which you perform wet grinding to remain stationary with a portion of the wheel immersed in the coolant. This will cause the wheel to absorb coolant and be thrown out of balance. Also, for the same reason, do not permit coolant to flow on a stationary wheel. After mounting a wheel, stand to one side and allow it to run at full operating speed for at least 1 minute prior to using it.

NOTE: A wheel may fly apart. ALWAYS wear eye protection when you're grinding and stand to one side to avoid possible injury. Your eyesight is not expendable!

53-523

Figure 87. Grinding wheel storage.

10-13. *Mounting.* You mount a grinding wheel on the wheel spindle by means of wheel flanges or a collet. Power is transmitted through the flange or collet to the wheel. Figure 90 shows a flange mounting and figure 91 shows a collet mounting. Tighten the wheel between the flanges, or with the



53-524

Figure 88. Wheel balancing stands.

collet, enough to prevent wheel slippage and to transmit the driving torque. Do not tighten enough to crush the wheel. The safety guard should cover from one-half to three-quarters of the wheel diameter. The wheel guard should not expose more of the upper portion of the wheel than is required.

10-14. In flange mounting you mount the wheel directly on the wheel spindle. There is a flange on each side of the wheel. These flanges must be equal in diameter and the center portion must be relieved, as illustrated in figure 90. The outer portion of the flanges provides the bearing surface. The diameter of a flange should be about one-third of the diameter of the wheel. Other design features, such as the radial width of the flange bearing surface and the thickness of the flange bore and bearing surface, are given in chart 8.

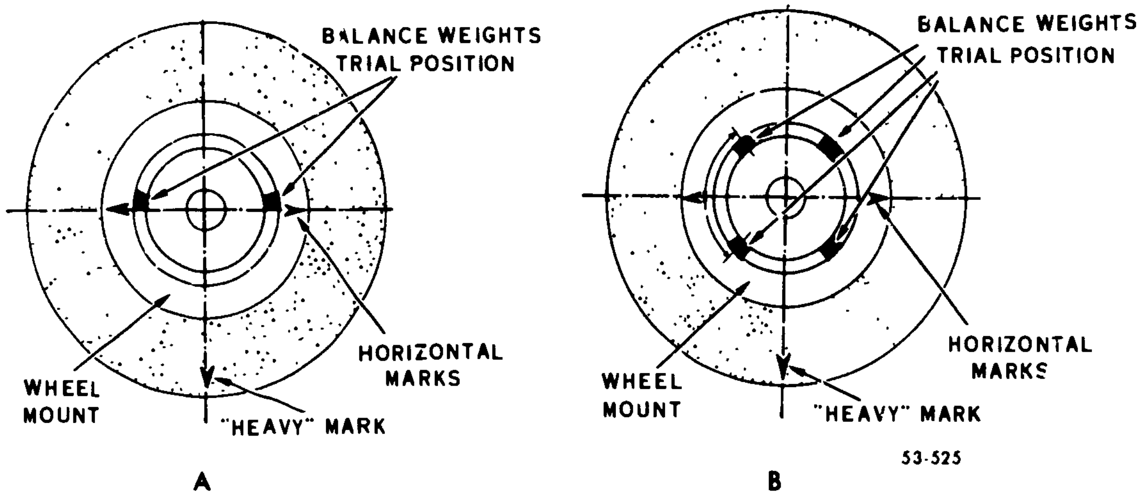


Figure 89. Use of balance weights.

Normally, you need only to be concerned about flange diameter. Some flanges are keyed to the spindle shaft. Others are pressed on the shaft. Insert a paper biotter no thicker than .025 inch and no smaller than the flange diameter between each flange and the wheel. Hold the spindle to prevent it from turning and tighten the spindle nut against the outer flange just enough to hold the wheel firmly.

10-15. Some grinding wheels are designed to be mounted on a collet, as shown in figure 91. Small screws that pass through the bore of this type of wheel tighten the flanges of the collet against the wheel. Tighten one screw and then the one directly opposite, etc., to equalize the pressure against the wheel. You can mount the wheel on the collet with the collet either off or on the spindle. If the collet is already in place, it is easier to follow the latter practice. Otherwise, it is necessary to remove the collet from the spindle with a puller.

10-16. *Dressing and truing.* As you may remember from Volume 1, a grinding wheel is *dressed* to improve or alter the cutting action of the wheel. The wheel is *trued* to restore a concentric surface to the wheel cutting face. You can expect a grinding wheel to perform efficiently only if it is properly dressed and trued. Within limits, a grinding wheel is self-sharpening. The forces acting at the point of contact tend to fracture and dislodge the dulled abrasive grains. This action results in new and sharp cutting grains contacting the work. In time, however, a grinding wheel will require dressing in order to clean out the metal-clogged pores.

10-17. There are several types of dressing and truing tools, as shown in figure 92. The hand-held Huntington *mechanical dresser* has alternate pointed and solid discs, which are losely mounted

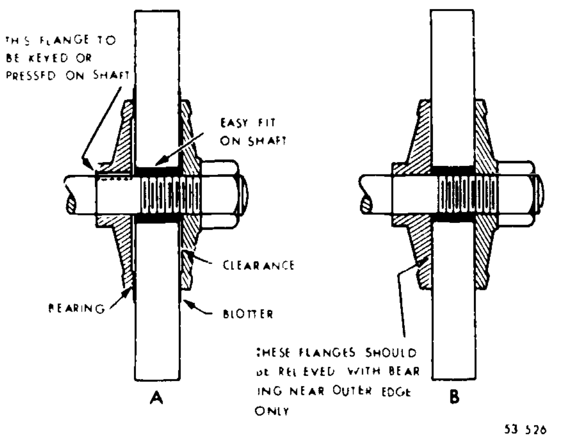


Figure 90. Flange mounting.

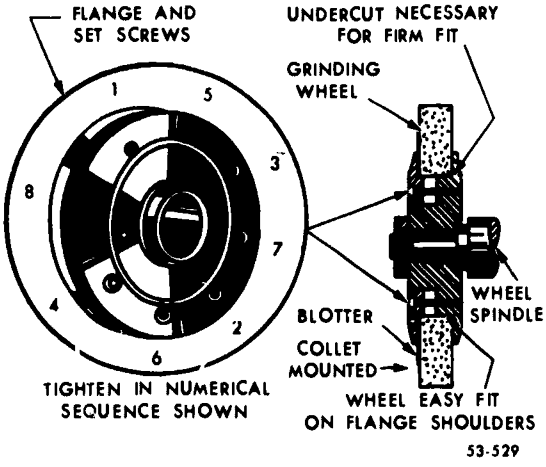
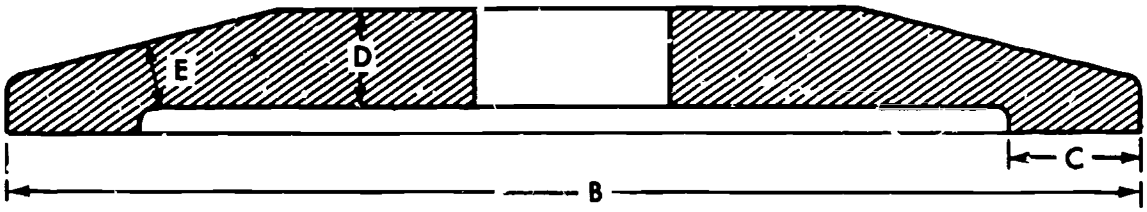


Figure 91. Collet mounting.

CHART 8

FLANGE DIMENSIONS



A	B	C		D	E
Diameter of wheel (inches)	Minimum outside diameter of flanges (inches)	Radial width of bearing surface (inches)		Minimum thickness of flange at bore (inches)	Minimum thickness of flange at edge of recess (inches)
		Minimum	Maximum		
1	3/8	1/16	1/8	1/16	1/16
2	3/4	1/8	3/16	1/8	3/32
3	1	1/8	1/4	3/16	3/32
4	1 1/4	3/16	3/8	3/16	1/8
5	1 1/2	3/16	3/8	1/4	1/8
6	2	1/4	1/2	3/8	3/16
8	3	1/4	1/2	3/8	3/16
10	3 1/2	5/16	5/8	3/8	1/4
12	4	5/16	5/8	1/2	5/16
14	4 1/2	3/8	3/4	1/2	5/16
16	5 1/2	1/2	1	1/2	5/16
18	6	1/2	1	5/8	3/8
20	7	5/8	1 1/4	5/8	3/8
22	7 1/2	5/8	1 1/4	5/8	7/16
24	8	3/4	1 1/4	5/8	7/16
26	8 1/2	3/4	1 1/4	5/8	1/2
28	10	7/8	1 1/2	3/4	1/2
30	10	7/8	1 1/2	3/4	5/8
36	12	1	2	7/8	3/4

53-577

on a pin. Use this dresser to dress coarse-grit wheels and for wheels used in hand grinding. This type is the most efficient in picking the metal particles out of the wheel without causing a big loss of abrasive. You do not need to use a coolant.

10-18. The *abrasive stick dresser* comes in two shapes, square, for hand use, and round, for mechanical use. The abrasive stick dresser is often used instead of the more expensive diamond dresser for dressing shaped and form wheels. It is also used for general grinding wheel dressing.

10-19. The *abrasive wheel dresser* is a bonded silicon carbide wheel that is fastened to the machine table at a slight angle to the grinding wheel and is driven by contact with the wheel. The dresser produces a smooth, clean-cutting face that leaves no dressing marks on the work. You do not usually need to use a coolant.

10-20. The *diamond dresser* is the most efficient for truing wheels used for precision grinding where accuracy and high finish are required. A dresser may have a single diamond or multiple

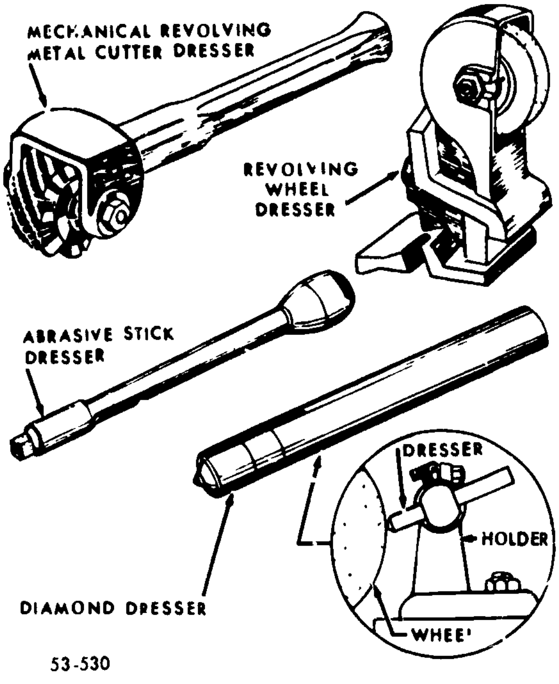


Figure 92. Dressing tools.

diamonds mounted in the end of a round steel shank. You should inspect the diamond point frequently for wear. The point is the only usable part of the diamond. Once the point is worn away, it will not properly dress the wheel. *Always* use a large flow of coolant for diamond dressing, since diamonds are adversely affected by heat. As shown in figure 93, you should hold the dresser with the diamond point positioned 30° to the plane of the wheel. Also, you should slant the diamond 3° to 15° in the direction of rotation to prevent chatter and gouging. Rotate the diamond slightly in its holder between dressing operations to keep it sharp. A dull diamond will press the wheel cuttings into the bond pores and load the face of the wheel, which in effect increases the hardness of the wheel.

10-21 When you use a diamond dresser to dress or true a grinding wheel, the wheel should be turning at or slightly less than normal operating *speed—never* at a higher speed. For wet grinding, the wheel should be flooded with coolant when you dress or true it. For dry grinding, the wheel should be dressed dry. The whole dressing operation should simulate the grinding operation as much as possible. Whenever possible, hold the dresser by means of some mechanical device. It is a good idea to round off wheel edges with a handstone after dressing. This is especially true of a fine finishing wheel, to prevent chipping the wheel edges. You do not round off the edges if the work requires sharp corners. The grinding wheel usually wears more on the edges, leaving a high spot towards the center. When you start the dressing or truing operation, be certain that the point of the dressing tool contacts the highest spot of the wheel *first,* to prevent the point from digging in. Regardless of the type of grinding machine you are using, the grinding wheel must be rotating when you dress it. Depending upon the type of grinding machine, the face of the grinding wheel either moves across the stationary point of the dresser or the point of the dresser moves across the face of the wheel.

10-22. You should progressively feed the dresser tool point .001 at a time into the wheel until the sound indicates that the wheel is perfectly true. The rate at which you move the point across the face of the wheel depends upon the grain and the grade of the wheel and the finish desired. A slow feed gives the wheel a fine finish; however, if the feed is too slow, the wheel may glaze. A fast feed makes the wheel free cutting. However, if the feed is too fast, dresser tool marks may be left on the wheel. You can determine the correct feed only by trial, but you should always maintain a *uniform* rate of feed during any one pass.

10-23. **Speeds.** Grinding operations are subject to many variables. For this reason, wheel, work,

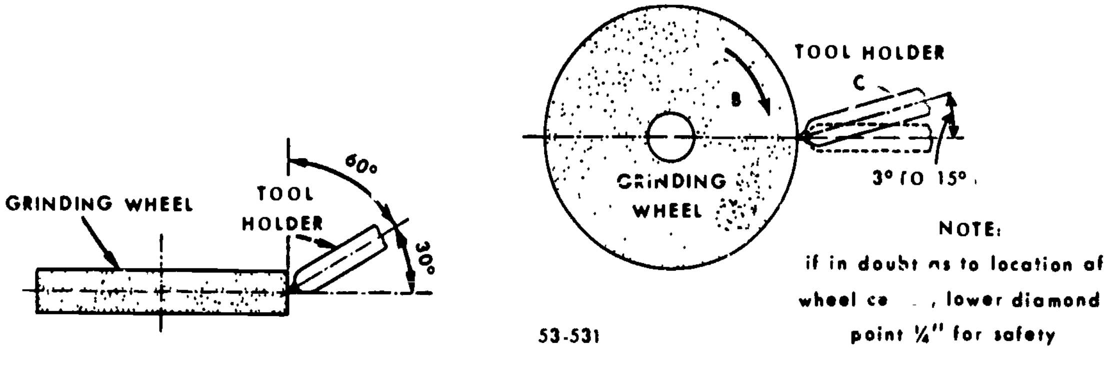


Figure 93. Position of diamond dresser.

and machine speeds must be varied to give suitable results.

10-24. *Spindle speed.* You do not have fine spindle speed control in grinding work. For most operations the surface foot speed (SFS) is between 5500 and 6500. Since you cannot control the spindle speed directly, you have to adjust either feed or depth of cut to improve the grinding operation. When you are grinding steel with a vitrified- or silicate-bonded wheel, the SFS for the following grinding operations are recommended:

Cylindrical	5500—6500
Face	5000—6000
Internal	2000—6000
Cutter	4000—5000

10-25. It will be necessary, sometimes, to calculate the SFS of a grinding wheel. You can use this formula:

$$\text{SFS} = \text{diameter of wheel} \times 0.2618 \times \text{rpm}$$

Example: You want to grind a cylindrical piece of work and you do not know the spindle rpm. The diameter of the grindng wheel is 8 inches. What is the SFS? First, you determine spindle rpm with a tachometer or by reading the rpm from the drive pulley. Assume that it is 2500:

$$\text{SFS} = 8.00 \times 0.2618 \times 2500$$
$$\text{SFS} = 5236$$

A more convenient formula is

$$\text{SFS} = \frac{\text{dia wheel}}{4} \times \text{rpm}$$
$$\text{SFS} = 8/4 \times 2500$$
$$\text{SFS} = 5000$$

10-26 The recommended SFS for [illegible]ical grinding is 5500 to 6500. Assume th[illegible]t to obtain a SFS of 6000. What spindle [illegible] produce this SFS? You can use the formula:

$$\text{Spindle rpm} = \frac{\text{SFS}}{\text{dia wheel} \times 0.2618}$$
$$\text{Spindle rpm} = \frac{6000}{[illegible] \times 0.2618}$$
$$\text{Spindle rpm} = 2865$$

or by approximation:

$$\text{Spindle rpm} = \frac{\text{SFS} \times 4}{\text{dia wheel}}$$
$$\text{Spindle rpm} = \frac{6000 \times 4}{8}$$
$$\text{Spindle rpm} = 3000$$

10-27. If spindle rpm is fixed by the manufacturer, you will have to select a larger diameter wheel to increase the SFS. Otherwise, you select the drive pulley that corresponds most closely to the desired spindle rpm.

10-28. **Work Speed.** In addition to the wheel speed, you must also consider the work speed in operations such as cylindrical grinding, because the work also rotates. Work speed is the speed at which the surface of the work rotates as it passes the contact point with the wheel face. The recommended SFS for grinding a plain cylinder is 60 to 100. You have to consider several factors when you select work speed: size and shape of the work, type of material, amount of stock to be removed, and desired finish. Irregular or out-of-balance work must be turned more slowly. If the grade of the grinding wheel is not exactly correct, you can improve the grinding efficiency by varying the work speed. You can change the work speed by means of the headstock pulley. Use the following formula to obtain the desired spindle rpm:

$$\text{rpm} = \frac{\text{SFS}}{0.2618 \times \text{dia of work}}$$

Example: You want to obtain a work SFS of 80 for grinding a shaft 1 inch in diameter. What is the desired spindle speed?

$$\text{rpm} = \frac{80}{0.2618 \times 1}$$
$$\text{rpm} = 305$$

10-29. **Machine Speed.** Machine speed refers to the movements of certain parts of the grinding machine. These movements can be traverse, cross-feed, and infeed, depending on the type of grinding machine being used. These speeds can also be varied to improve grinding efficiency. They will be discussed in more detail later in this chapter, as specific grinding machines and their operations are explained.

11. Surface Grinding Machine Operations

11-1. Surface grinding is the grinding of flat surfaces. In actual use the surface may be in a horizontal, vertical, or angular position, but you grind it in the horizontal position. You can compare surface grinding to machining a flat surface on a milling machine. Remember that the milling machine uses a multitooth cutting tool. A grinder uses a grinding wheel, which could be considered a multi-tooth cutter with a great number of teeth. In this section we will discuss types of surface grinders, the action of the wheelhead and table, and a typical surface grinding operation.

11-2. **Types of Surface Grinders.** Horizontal spindle surface grinders are designed for the production of flat surfaces where precision, fine finish, and rapid removal of stock are requirements.

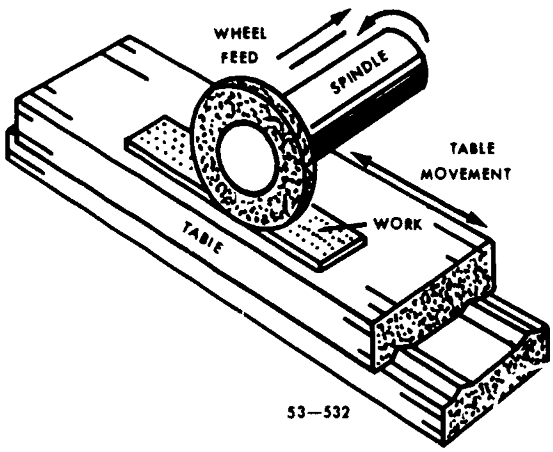


Figure 94. Reciprocating table.

They may be divided into two types according to table movement. On the reciprocating table type you mount the work on a reciprocating table which passes the work back and forth under the wheel face. Transverse wheel feed takes place at each end of the table movement. Figure 94 illustrates surface grinding on a reciprocating table. On a rotating table type you mount the work on a circular table which rotates the work under the wheel face, as shown in figure 95. The wheel moves in a horizontal plane across the work from the outer to the inner circumference and back.

11-3. **Wheelhead and Table.** Before you can do surface grinding, you must understand the action of the wheelhead and table. Figure 96 shows a typical horizontal spindle surface grinder. The grinder in your shop may differ somewhat from this one but the principles of operation apply to all types.

11-4. *Wheelhead.* The wheelhead contains the spindle motor, spindle, and grinding wheel. The wheelhead is mounted on the vertical ways of the machine and may be moved either vertically or horizontally. There are usually two spindle speeds; 1800 rpm for 12-inch and larger wheels and 3600

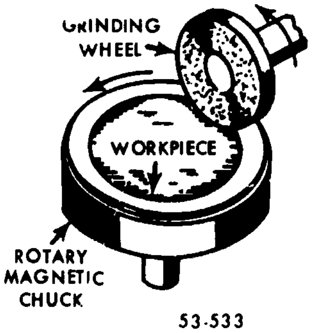
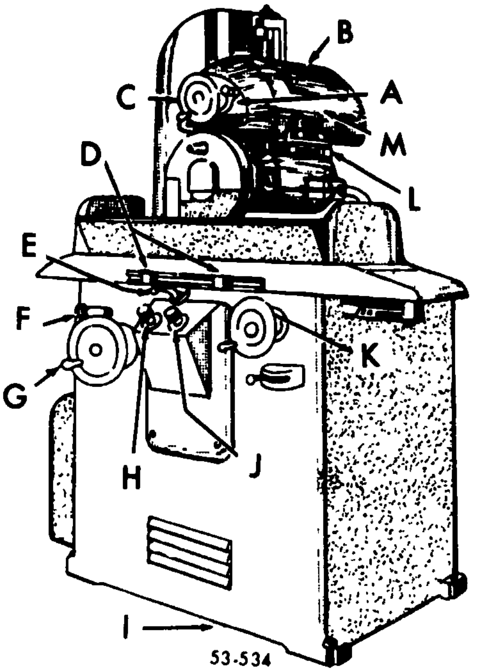


Figure 95. Rotating table.

rpm for 8-inch and smaller wheels. A high-speed control located behind the grinding wheel makes it impossible to operate a wheel over 8 inches in diameter at the higher speed.

11-5. You can move the wheelhead vertically by turning the vertical feed handwheel. This handwheel is graduated in 0.005-inch increments and has a micrometer stop for adjustments as fine as 0.001 inch.

11-6. You can move the wheelhead horizontally either by power or by turning the head crossfeed handwheel. This handwheel is graduated in 0.005-inch increments. One full turn advances the



A. CROSSFEED SELECTOR LEVER
B. WHEELHEAD
C. HEAD CROSSFEED HANDWHEEL
D. TABLE TRIP DOGS
E. TABLE REVERSE LEVER
F. MICROMETER STOP
G. VERTICAL FEED HANDWHEEL
H. TABLE THROTTLE
I. BASE
J. HEAD THROTTLE
K. TABLE TRAVERSE HANDWHEEL
L. HEAD TRIP DOGS
M. HEAD REVERSE LEVER

Figure 96. Horizontal spindle surface grinder.

head 0.100 inch. The crossfeed selector lever, located in back of the hand wheel, must be engaged for hand feed and disengaged for power feed. The power crossfeed is hydraulically operated and you control it with the head throttle. You can select either intermittent feed or continuous feed. When it is set for intermittent, the table feeds traversly each time the longitudinal travel reverses. This movement can be set to any amount desired, from a few thousandths to ⅜ of an inch. You should se-

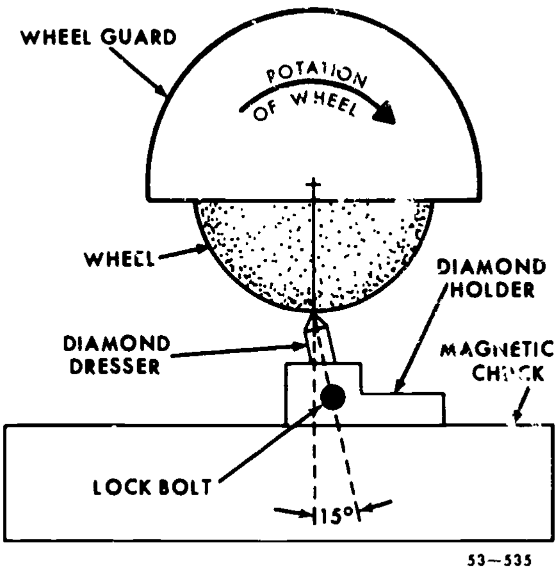


Figure 97. Surface grinder wheel dressing setup.

lect the rate of crossfeed prior to beginning the grinding operation. The maximum crossfeed rate should not exceed one-half the width of the wheel face per revolution. Continuous feed is used only for setting the position and length of the crossfeed, and in dressing and truing the wheel. It feeds regardless of whether there is any table movement.

11-7. *Table.* You mount the work on a magnetic chuck, which is bolted to the table. You can feed the table by hand by turning the table traverse handwheel or by hydraulic power by turning the table throttle. You can vary the table feed from 0 to 50 feet per minute. Position the table and adjust the amount of movement by means of the table trip dogs.

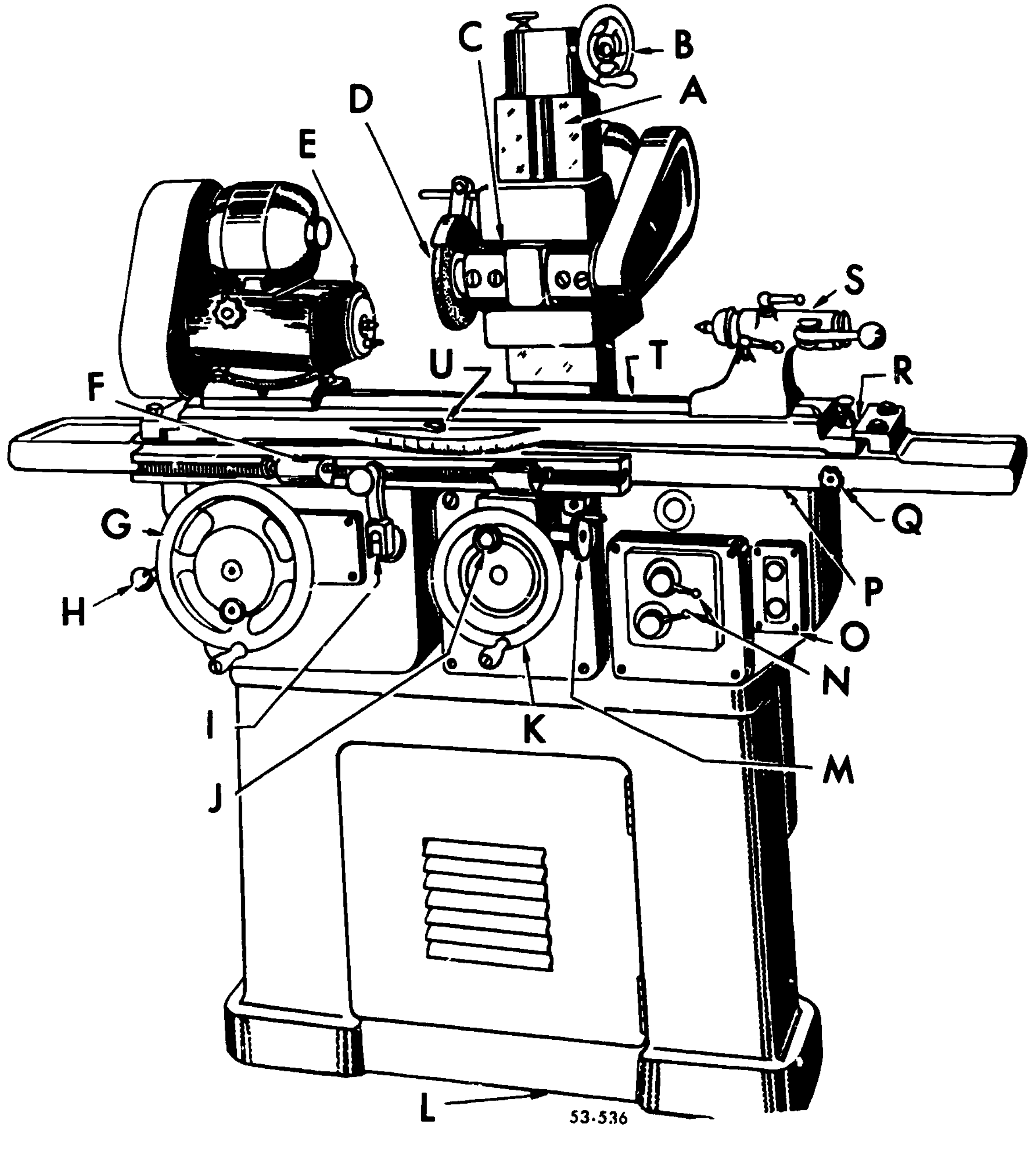
11-8. In surface grinding you often grind several small workpieces at the same time. The pieces are held by a magnetic chuck and the main problem is to maintain parallelism between the opposite sides of the work. Make sure that the workpieces rest solidly on the chuck and that they have been properly packed or shimmed before you magnetize the chuck. If the workpieces have been heat treated, you should first rough grind all the surfaces that are to be finished in order to relieve the internal stresses. The rough grinding may leave the work slightly distorted. If this occurs, shim the work and finish grind one surface until it is flat. Then finish grind the opposite surface parallel to, and the other surfaces square with, the original surface. One method you can use to stress relieve a small heat-treated workpiece is to immerse it in boiling water for about 5 hours between rough and finish grinding. If a rectangular workpiece is slightly out of square, you can correct the condition by clamping it tightly between two hardened steel parallels and regrinding it.

11-9. The work must be held rigidly. When you are grinding the edge of rectangular-shaped work, the area of contact between the magnetic chuck and the work is small and rigidity becomes a problem. You can overcome this difficulty by placing a steel parallel on either side of the work. The parallels, as well as the work, will be held by magnetic attraction and a rigid setup will be assured.

11-10. **Surface Grinding.** Assume that you have a hardened steel parallel to grind to a specified size. We will discuss the procedures that could be used to grind it. You first mount the proper wheel on the wheel flange assembly. Then mount the flange assembly on the wheelhead spindle. Tighten the spindle and flange nuts as required. Then place the guard over the wheel. Wipe off the magnetic chuck with a clean cloth, then with your hand. If you can feel any burrs, remove them by rubbing the face of the chuck with a fine-grain abrasive stone. Then wipe the chuck off again. Place the diamond dresser and its holder, as shown in figure 97, on the magnetic chuck and turn the magentism on.

11-11. The diamond should tilt in the direction of wheel rotation. Now, position the wheel directly over the diamond and start the wheel rotating. After it has been rotating for several minutes, feed the wheel down until it just touches the diamond. After turning on the coolant, dress and true the wheel, using the hand crossfeed. Turn off the coolant. Let the wheel rotate for a few minutes, and then turn it off. Move the wheelhead assembly away from the chuck enough to allow safe and easy access to the chuck face. After the wheel has stopped rotating, turn off the magnetic chuck and remove the diamond and the holder. Once again, clean off the face of the magnetic chuck with a clean cloth and your hand, removing all abrasive residue.

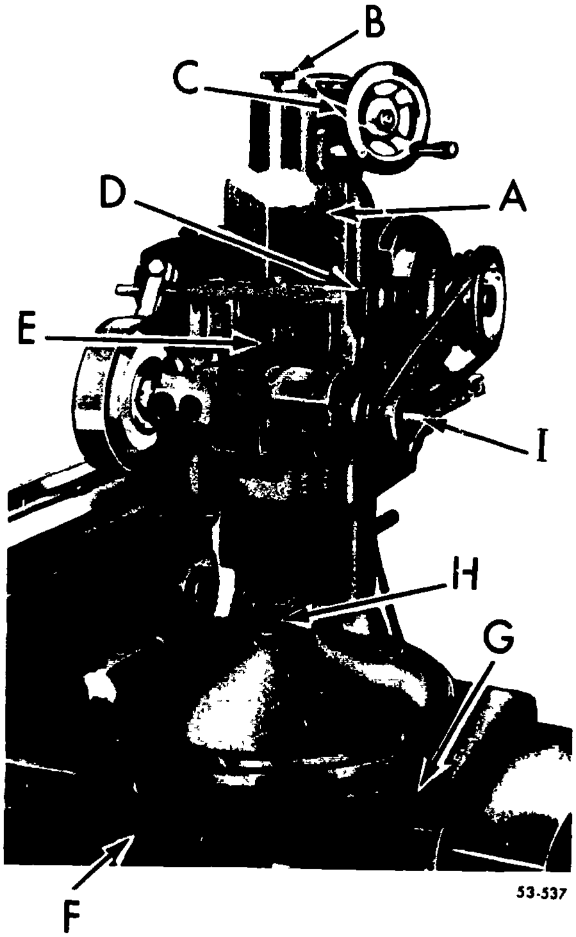
11-12. The loose scale caused by the heat treatment of the parallel should be removed by using abrasive cloth. Place the work on the magnetic chuck and shim it to keep it from moving during the grinding operation. Turn the magnetism on and position the reversible stop dogs so that the grinding wheel will run off both ends of the work. Also, position the wheelhead trip dogs so that the wheel will run off both sides of the work. The wheel motor, hydraulic system, and coolant pump should now be turned on. Using continuous power feed, check the wheelhead and table for correct wheel overrun and make any necessary adjustments. Next, with the table in motion, position the wheel directly over the parallel and pick up the cut by using hand feed. Position the coolant nozzle to



A. WHEEL SLIDE WAYS
B. HANDWHEEL
C. SPINDLE AND WHEEL SLIDE
D. GRINDING WHEEL
E. HEADSTOCK
F. TABLE TRIP DOGS
G. LONGITUDINAL HANDWHEEL
H. TABLE TRAVERSE ENGAGING LEVER
I. TABLE TRAVERSE LEVER
J. FINE CROSSFEED ENGAGEMENT KNOB
K. CROSSFEED HANDWHEEL
L. BASE
M. FINE CROSSFEED HANDWHEEL
N. TABLE TRANSVERSE SPEED LEVERS
O. ON-OFF SWITCH
P. SLIDING TABLE
Q. FINE ADJUSTMENT KNOB
R. TAPER SCALE
S. FOOTSTOCK
T. SWIVEL TABLE
U. CLAMP NUT

Figure 98. Brown and Sharpe universal tool and cutter grinder.

supply an adequate volume of coolant to the wheel and the work. The volume of coolant is more important than the pressure. Now, turn on the wheelhead power feed and rough grind the work. Remember not to exceed a .002 inch depth of cut. After rough finishing the first surface, stop the table motion. Shut off the coolant, turn off the wheel motor, and move the wheelhead away from the work. Turn off the magnetic chuck, and remove the work. Wipe off the chuck very carefully again, and then place the parallel in the same location, but with the ground surface down. Turn on the magnet, pick up the cut, and grind this side in the same manner as you did the first side. Do not grind to final size, but leave .002 inch for finish grinding. Grind the two remaining sides in the



A. WHEEL SLIDE UPRIGHT
B. SWIVEL CLAMPING KNOB
C. ELEVATING HANDWHEEL
D. ZERO MARK
E. WHEEL SPINDLE SLIDE
F. CROSS SLIDE
G. UPRIGHT SWIVEL PLATE
H. CLAMP
I. WHEEL SPINDLE (GUARD REMOVED)

Figure 99. Wheelhead and column.

same manner as the first two. Remember to leave .002 inch for finishing. Before finishing, dress the wheel. The depth of cut for finish grinding should not exceed .0005 inch. The work should be allowed to "spark out." This is the proces of letting the work move under the rotating wheel until there are no more sparks visible. After the work is checked for accuracy, deburr it with a hand abrasive stone.

12. Cylindrical Grinding Machine Operations

12-1. In cylindrical grinding the work is driven by a spindle while it is being held in a chuck, on an arbor, or between centers. In this respect cylindrical grinding is similar to a lathe machining operation.

12-2. In this section we will describe a cylindrical grinder. We will discuss the preparation of the work for grinding between centers and explain several typical cylindrical grinding operations.

12-3. **Description.** There are several types of cylindrical grinding machines but all of them have many features in common. A description of one machine will in many ways fit the description of all types. To be able to cylindrically grind any type of work, it is necessary for you to know something about the machine. We will explain the essential features of the Brown and Sharpe No. 13 Universal Tool and Cutter Grinder, which is shown in figure 98.

12-4. *Wheelhead.* The wheelhead, shown in figure 99, is mounted on the column or vertical ways of the machine. The spindle is mounted in self-adjusting bronze bearings. Both ends of the spindle are tapered to receive the wheel sleeves upon which the grinding wheels are mounted. This makes it easier and quicker to change wheels and helps assure a true running wheel. The wheelhead is also used to mount attachments, such as wheel guards, coolant guards, and the internal grinding attachment. You can move the wheelhead up and down the vertical column by means of a handwheel located at the top of the column. You can rotate the column on its base 360° in either direction.

12-5. *Crossfeed.* The column has a transverse movement, which moves the wheel away from or toward the work for a depth of cut. You adjust the coarse crossfeed by turning the large handwheel, shown in figure 98, which is graduated in .001-inch increments. This causes the table to move .100 inch per handwheel revolution. You make fine crossfeed adjustments by first tightening the

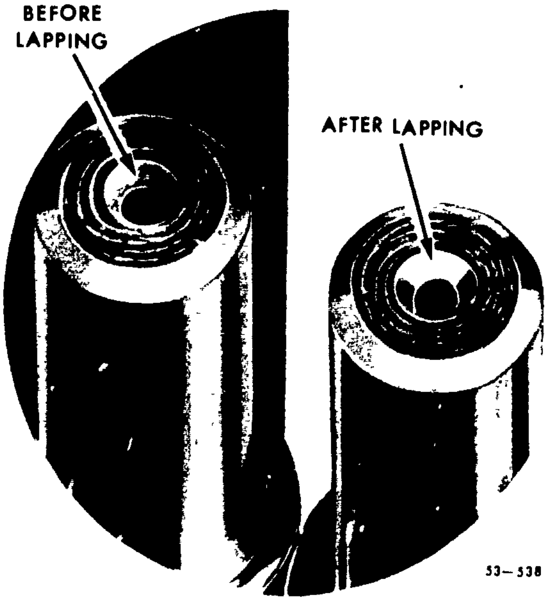
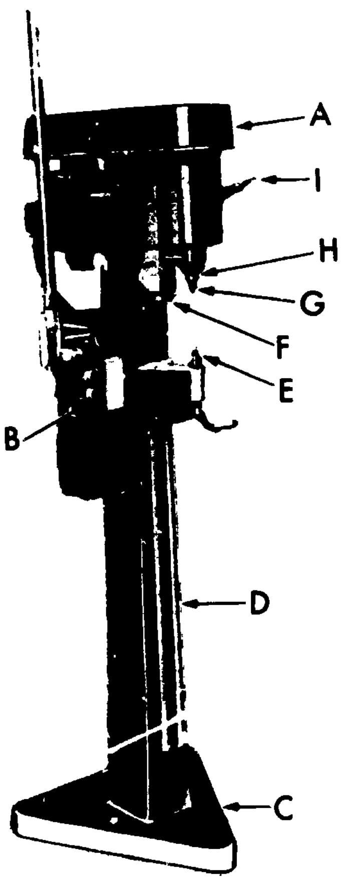


Figure 100. Rough and lapped center holes.



53-539

A. CHANGE SPEED BOX
B. SWITCH
C. BASE
D COLUMN
E. ADJUSTABLE CENTER
F. TRUING DEVICE
G. ABRASIVE STICK
H. CHUCK
I. HAND LEVER

Figure 101. Center lapping machine.

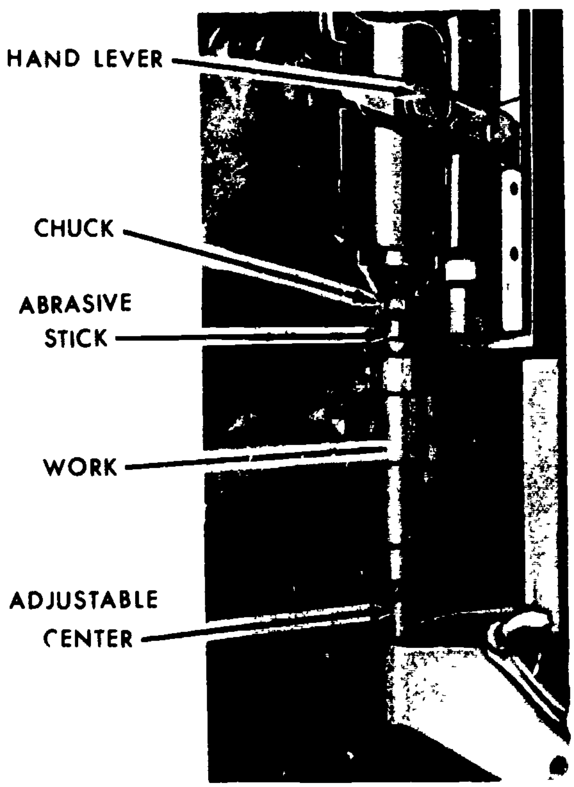
thumbscrew on the crossfeed handwheel and then turning the small fine crossfeed handwheel, which is graduated in .001-inch increments. One revolution of the small handwheel moves the crossfeed .001 inch.

12-6. *Motor*. The motor is mounted on the back side of the column and drives the spindle by means of a V-belt. By shifting the V-belt to different sets of pulleys, you can obtain a wheel speed of 2730, 3600, or 5150 rpm.

12-7. *Table*. The swivel table is locked to its base by five bolts, one in the middle and two on each end. After loosening these five bolts, you swivel the table on a pivot by turning a fine adjusting screw. This permits you to grind a maximum taper of 3 inches per foot. To make coarse adjustments, raise the spring knob at the right of the table and swivel the table by hand. By using the graduations on the front of the table, you can swivel the table through a range of 90° (45° to each side).

12-8. The automatic table travel is driven independently of the wheel spindle. You can move the table by hand by turning the handwheel on the left side of the machine. You can obtain a coarse or fine hand feed by positioning the lever located behind the handwheel which controls both the hand feed and the power travel. A safety mechanism located on this lever prevents engaging the longitudinal power travel while the lever is locked in position for hand feed. Also, when the lever is unlocked for power travel, you cannot engage the hand feed. The length and location of table travel is controlled by the position of the trip dogs on the rack on the front of the table. You can move the trip dogs along the rack by lifting the catch, and make fine adjustments with a thumbscrew on the dog.

12-9. *Headstock*. The headstock, which is mounted on the left end of the table, supports and rotates the work. You can swivel the headstock on its base as much as 100° to the right or to the left of ZERO. It is clamped to the table T-slots by two T-headed bolts. The headstock spindle is driven by a motor and has two types of drive: live center and dead center. You use live center drive to grind work that is held in a chuck or on a faceplate. You



53—540

Figure 102. Lapping setup.

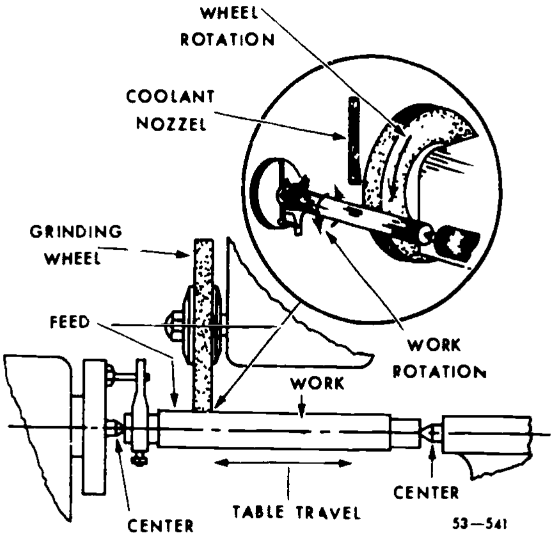


Figure 103. Cylindrical grinding.

use dead center drive to grind work that is held on dead centers. Dead center drive is more accurate, since it eliminates any possible spindle runout present in the live center drive.

12-10. *Footstock.* The footstock is adjustable along the table T-slot and is held in place by a T-headed bolt. A constant spring tension on the center provides an even pressure on the work. The wpring tension compensates for expension or contraction of the work due to temperature change during the grinding operation. The center has a locking device for heavy work or for work that is likely to be forced from between centers during grinding.

12-11. **Preparing Work for Grinding Between Centers.** Before you grind work held between centers, you must lap the center holes. This insures precise limits for roundness, straightness, and concentricity and increases the life of the machine centers. Lapping removes the scale and distortion left by heat treating and corrects inaccurately or roughly drilled holes. Figure 100 shows rough and lapped center holes.

12-12. If your shop does not have a center lapping machine, such as the one shown in figure 101, you can lap the center holes in a lathe or drill press. To do this, you mount a round piece of hard wood, one end of which is turned to a point with a 60° angle, in a chuck. Cover the rotating pointed end of the wood with lapping compound and insert it into the center hole of the work. Be sure to lap the center hole at each end of the work.

12-13. Lapping a center hole by machine, as shown in figure 102, is a simple operation. You hold one end of the work on an adjustable center. Pulling down on a hand lever, you bring the rotating lapping stone into contact with the center hole. By changing the belt on the pulleys, you can obtain speeds of 720, 1300, 2400, and 4500 rpm. You can move the work rest up and down on the ways to accommodate work up to a length of 36 inches. The maximum width that can be held is 10 inches. A diamond dressing device is mounted on the spindle bracket. There is a micrometer adjustment for positioning the diamond dresser for each dressing cut. You perform the dressing operation by swinging the dresser into position and passing the diamond dresser across the lapping stone. When you are not using the dresser, it should be swung back 90° out of the way. Always dress the lapping stone when the spindle is in the retracted position, never in the extended position. The lapping stone is a bonded abrasive wheel cemented on a ½-inch steel spindle. It should be dressed frequently. A loaded or blackened stone does not cut freely, and it generates excessive heat. Lapping stones are often treated with oil to improve their cutting action. Some shops, however, prefer to use untreated stones.

12-14. **Cylindrical Grinding.** Cylindrical grinding is done to remove the warpage caused by heat treatment, reduce the work diameter to exact size, and improve the finish. The work can be held and rotated by mounting it:

a. Between centers and driving it with a drive dog, as shown in figure 103.
b. In a chuck and supporting it with a footstock.
c. On a live center and supporting it with a footstock.
d. On a faceplate.

The revolving grinding wheel provides the cutting action that takes place at the area of contact, as shown in figure 104. The area of contact will vary when the dimensions of the wheel or the work are

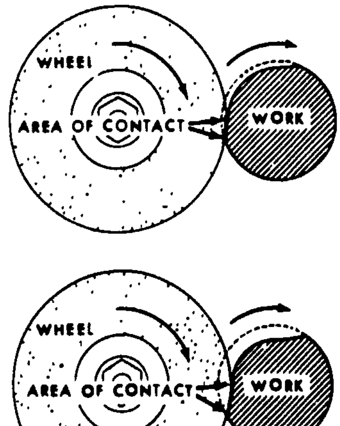


Figure 104. Area of contact.

increased or decreased and when the depth of cut is increased or decreased.

12-15. The wheel and the work are usually set to revolve in opposite directions at the area of contact, as shown in figure 104. This provides a shearing type of cutting action between the wheel and the work. Grinding is subject to more variables than other machining operations. Therefore, in order to grind work efficiently and economically, you must use the correct combination of wheel speed, work speed, and table traverse.

12-16. The typical grinding operations that we will discuss are straight cylindrical grinding, conical (tapered) grinding, the grinding of shoulders and grooves, face grinding, and internal grinding.

12-17. *Straight cylindrical grinding.* Assume that you have the job of grinding a straight-fluted hand reamer to a specified size. After preparing the center holes, you should select the proper wheel and mount it on the wheel flange assembly. Then mount the flange assembly on the wheel head spindle and tighten the spindle nut.

NOTE: The spindle nut uses left hand threads.

Next, place the wheel guards in position. They must cover the wheel and not interfere with any other parts of the machine. Position the column so that the graduations on the base and front of the table indicate a ZERO setting. Then the lock bolts should be snugged down. When the column and table are at the ZERO setting, the wheelhead is parallel to the table. Wipe the table off and attach the diamond dresser and the holder in the proper positon to dress the wheel. The proper position is shown in figure 93. Using the formulas discussed in Section 10 of this chapter, calculate the wheel and work speed.

12-18. Turn on the spindle drive motor to start the wheel rotating and leave it on until done with the entire grinding operation. Letting the wheel run keeps the bearings at their operating temperature and prevents the wheel from becoming unbalanced through the absorption of coolant. After placing the splash guards in position, turn on the coolant pump motor and position the nozzle, as shown in figure 103. There should be an adequate flow of coolant to the area of wheel contact. The center of the spindle (and thus the wheel) should be alined with the headstock center. You can do this by using the elevating handwheel to adjust the wheelhead spindle until the lower cricket mark on the vertical slide corresponds with the wheelhead cricket mark. Turn the crossfeed handwheel to bring the revolving wheel towards the diamond dresser. The dresser should touch the wheel very lightly. The depth of cut should not exceed .001 per pass during the dressing and truing operation.

NOTE: As in surface grinding, the wheel is dressed wet if the work is to be ground wet, and dry if the work is to be ground dry.

After the dressing and truing operation is complete, move the wheel away from the dresser far enough to allow you to safely remove the dresser and holder and the splash guards.

12-19. Thoroughly wipe the table off and mount the headstock on the left end with its graduated base set to ZERO. Mount and secure the footstock in such a manner that the reamer will be supported with some spring tension on the center. Attach the proper size drive dog to the fluted end of the reamer. Place the work between the centers of the headstock and footstock.

NOTE: Be sure the center holes of the reamer are lapped clean and smooth, then coat them with clean white lead to act as a lubricant between the reamer and the centers.

The end of the work with the drive dog on it should be positioned on the headstock dead center with the dead center drive stud placed between the two tails of the dog. The reversible stop dogs should now be positioned to allow minimum table traverse. Allow the grinding wheel to run off the shank end of the reamer into the gap of the footstock half center. Not more than one-half of the wheel width should run off the shank into the undercut between the shank and blades. Bring the revolving wheel to within 1/8 inch of the reamer shank, and engage the work head clutch to start the reamer revolving. Remember, the wheel and the reamer should revolve in opposite directions at the area of contact, as shown in figure 104. Try to pick up the cut at the highest point of warp by hand manipulation of the crossfeed and table traverse. Once the cut has been picked up, set the splash guards in place, turn on the coolant, and engage the table traverse. The table traverse feed for rough and finish grinding is calculated by using the information in chart 9 and the formula:

Table traverse = width of wheel × fraction for finish × rpm of work.

After working out this formula, set the traverse speed levers to correspond as closely as possible to these calculations.

12-20. Grind the reamer shank, using light depths of cut until you have a ground surface from end to end. When the shank is ground to specifications, stop the table traverse with the wheel positioned off the shank of the reamer in the gap of the footstock half center. Remove the reamer from between the centers. To check the shank for taper, measure each end with a vernier micrometer. If taper is present, loosen the five locking bolts on

CHART 9
RECOMMENDED TRAVERSE RATES

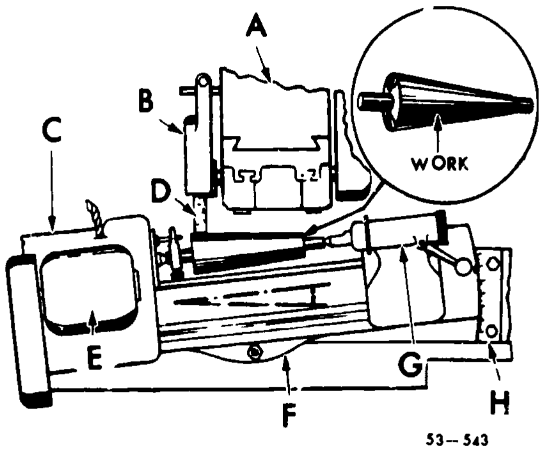
FINISH	TABLE TRAVEL PER WORK REVOLUTION
ROUGH	3/4 OF WHEEL WIDTH
MEDIUM	1/3—1/2 OF WHEEL WIDTH
FINE	1/4- 1/2 OF WHEEL WIDTH

53-528

the table, and swivel the table by turning the fine adjustment screw. This screw can compensate for any taper up to 3 inches of taper per foot.

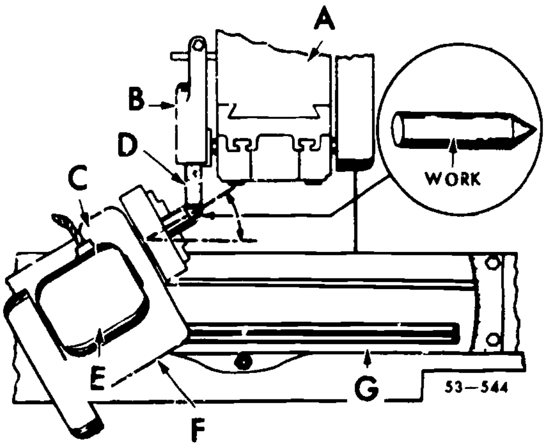
CAUTION: Any time the table is swiveled toward the wheelhead, you must back the wheel away from the work and pick up the cut again. Moving the table toward the wheel also moves the work toward the wheel. This will produce too deep a depth of cut and could result in destroying the wheel and damaging the workpiece.

After swiveling the table to eliminate the taper, snug down the table lock bolts. Pick up the cut, and take another cut. Measure the shank again. If the taper is still out of limits, readjust the swivel table again and take another cut. Repeat these operations until the taper is within the limits of the specifications. Once the taper is eliminated or within limits, rough grind the shank to within .001 of finished size. Do not forget that the body size of the shank of a hand reamer is .002 to .005 less than the nominal size of the reamer. This clear-



A. WHEELHEAD
B. WHEEL GUARD
C. HEADSTOCK
D. GRINDING WHEEL
E. MOTOR
F. SWIVEL TABLE
G. FOOTSTOCK
H. TAPER SCALE

Figure 105. Conical grinding setup for slow tapers.



A. WHEELHEAD
B. WHEEL GUARD
C. HEADSTOCK
D. GRINDING WHEEL
E. MOTOR
F. SWIVELED HEADSTOCK
G. TABLE

Figure 106. Conical grinding setup for steep tapers.

ance enables the shank to pass through the reamed hole without binding.

12-21. Remove the reamer from between centers and prepare to finish grind the shank. Dress the wheel and calculate the traverse feed for finish grinding. Pick up a light depth of cut and finish grind to size. Remember, you have only .001 to remove, so be careful not to take too deep a cut. When you are finished, remove the reamer from between centers.

12-22. Place the drive dog on the finished shank with shim under the end of the screw to prevent damaging the finished surface. Repositon the reversible stop trip dogs for the longer length of the reamer blades in the same manner as for the shank. Reset the table traverse for rough grinding. Use the same calculations and steps that you used to grind the shank.

> NOTE: To insure clearance behind the cutting edge, the reamer should rotate so that the heels of the blades strike the grinding wheel first.

Now, rough grind the blades and check for possible taper after several passes of the wheel. When the diameter measures .001 to .0015 over the final size, remove the reamer from the machine, and dress the wheel. Calculate the finishing feed, pick up the cut, and finish grind the blades to the designated size. When the final size is reached, remove the work and turn off all motors on the machine.

12-23. *Conical grinding*. Conical grinding is the name given to the grinding of cylindrically tapered surfaces, such as the shank and the point of a lathe center or the tapered portion of a taper plug gage. The reason that we refer to the operation as conical, and not just as taper grinding, is that flat work

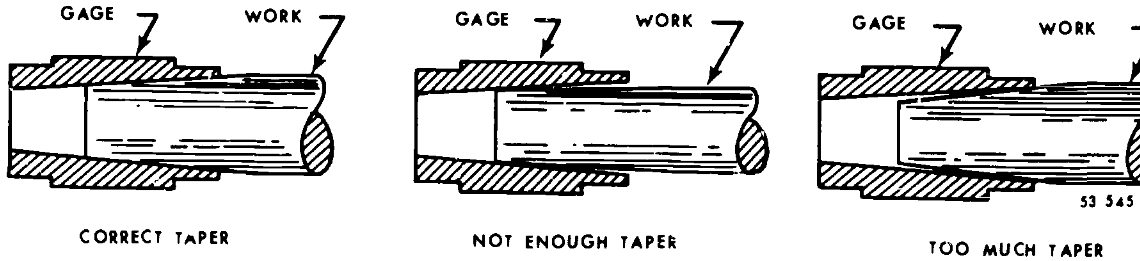


Figure 107. Checking taper using a taper gage.

may also be ground with a taper. You can grind either external or internal work conically. You will usually grind long, slow cylindrical tapers by setting the swivel table to correspond to the inches of taper per foot or to the desired number of degrees of taper. However, short cylindrical tapers are ground by swiveling the headstock on its base to the number of degrees of taper specified for the work.

12-24. You grind conical work in a manner similar to grinding straight cylindrical work, provided that the taper is not too steep or abrupt. After placing the work between the centers of the grinding machine, swivel the table to the required taper by means of the graduations on the end of the table. The correct work setup is illustrated in figure 105. This setup locates the axis of the work at an angle with the line of motion of the table. As the work moves across the face of the wheel, a taper is ground. The angle or taper depends upon how far you swivel the table from its central position. The correct angle or taper also depends directly upon the relation of the wheel to the work. In lathe work you will remember that in order to turn a true taper you need to set the cutting tool exactly at center height or even with the axis of the work being machined. In a similar manner, the grinding wheel axis must be exactly at center height or even with the axis of the work to grind a conical taper. If you position the wheel above or below the center of the work, the taper will be different from that which the table setting indicates.

12-25. You can usually grind steep tapers by swiveling the headstock to the desired angle of the taper, as shown in figure 106. Again, be sure that the axis of the grinding wheel is exactly at center height with the axis of the work. You can also grind internal conical tapers with the aid of the internal grinding attachment, which we will discuss later. When you grind conical surfaces, you can dress and true the grinding wheel either before or after swiveling the table. This is because the face of the wheel is always true and parallel to the ways regardless of the angle to which you swivel the table.

12-26. We will discuss the cylindrical grinding of a conical tapered plug, as shown in figure 105. You should first lap both centers so that they are smooth and clean. Then lubricate them with white lead. Select and mount the proper grinding wheel, and position the spindle at center height to correspond with the work center axis. Mount the tailstock where it will support the work and install a drive dog on the large end of the work. Compute and set the proper spindle and work speeds and

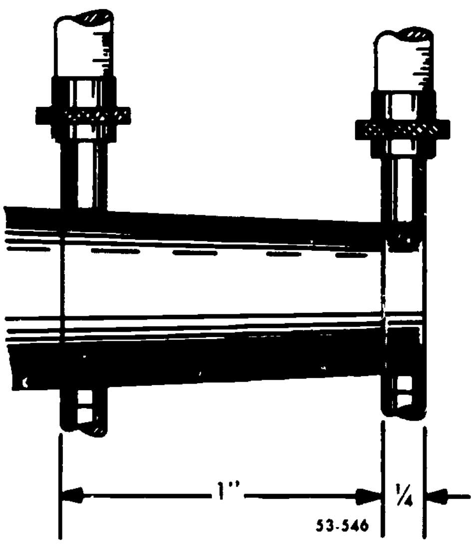


Figure 108. Measuring taper per inch (TPI).

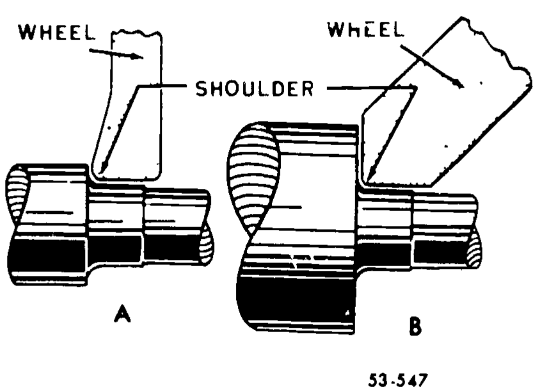


Figure 109. Shoulder grinding setups.

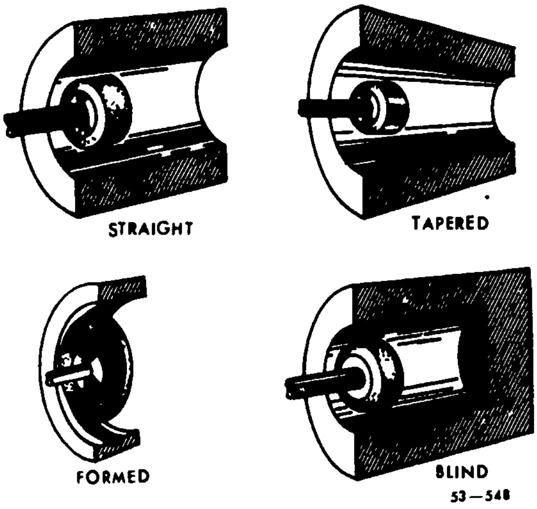
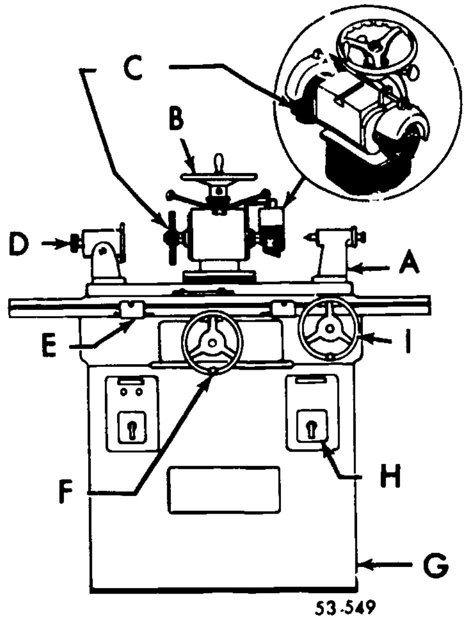


Figure 110. Typical internally ground surfaces.

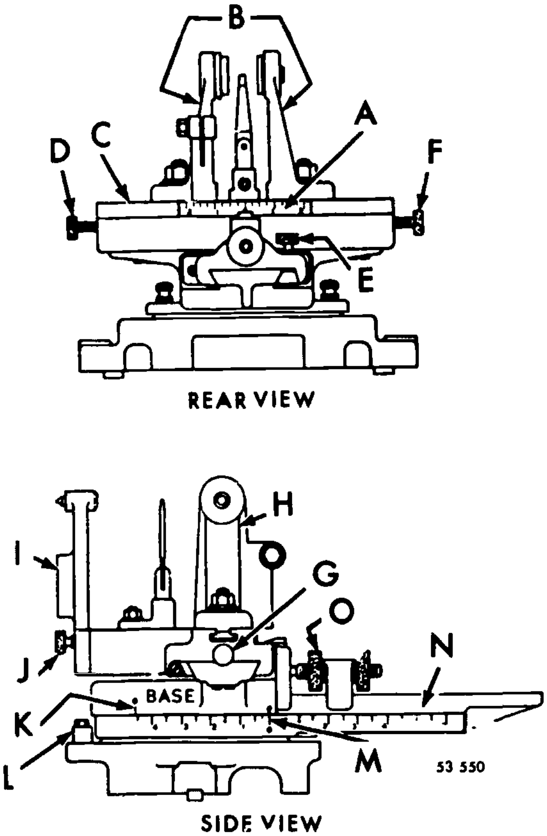
table feed. Using the table taper-per-foot scale or a test bar and dial indicator, swivel the table the required distance to grind the desired taper. Mount the diamond dresser and dress and true the abrasive wheel. Remember, let the wheel continue to rotate during the entire operation.

12-27. Mount the work between centers and pick up the cut. The wheel, work, and table traverse should all be in motion during the picking up of the cut. Rough grind the work surface until it is clean enough of heat treating residue to permit checking the taper. Once the taper is correct, rough grind the work to within .001 to .0015 of the specified diameter if you are using the micrometer method of checking taper. Rough grind to within 1/16 to 1/8 inch of the bottom of the gage if you are using the ring gage method. Then remove the work and dress the wheel. Set the finish speed and feed and finish grind the work to the desired size.



A. FOOTSTOCK
B. HEAD CONTROL WHEEL
C. GRINDING WHEEL
D. UNIVERSAL WORK HEAD
E. TABLE TRIP DOG
F. CROSSFEED HANDWHEEL
G. BASE
H. WORK HEAD MOTOR SWITCH
I. TABLE DRIVE HANDWHEEL

Figure 111. Typical tool and cutter grinder.



A. TAPER SHANK REFERENCE SCALE
B. WORK HOLDERS
C. SLIDE
D. SIDE ADJUSTMENT SCREWS
E. FINE ADJUSTMENT SCREW
F. SIDE ADJUSTMENT SCREWS
G. SIDE ADJUSTMENT SCREWS
H. WORKHOLDER
I. DIAMOND REFERENCE SURFACE
J. DIAMOND CLAMP SCREW
K. DIAMOND REFERENCE LINE
L. SWIVEL STOPS
M. WORKHOLDER REFERENCE LINE
N. SWIVEL
O. FINE ADJUSTMENT SCREWS

Figure 112. Radial grinding attachment.

12-28. There are many methods of checking tapers. The two most common methods are to measure the taper per inch with a vernier micrometer and to check the taper with a gage. When you check a taper with a tapered ring or plug gage, you should use Prussian blue or white lead to check the contact of the surface being ground with the mating surface of the gage. To check an external taper, first apply a very thin coat of Prussian blue or white lead on the work surface. Insert the taper into the proper ring gage, as shown in figure 107. After inserting the work in the gage, turn it with a wringing motion, and then withdraw it. The pattern of the blueing or white lead as it smears will show if the correct taper is being ground. If you don't have the correct size ring gage, you can check for correct taper with a micrometer, as shown in figure 108. Scribe two lines on the taper 1 inch apart, and measure the diameters. Determine the difference, and compare with the desired taper per inch. No matter which way the taper is checked, you adjust the swivel table to get the correct taper.

12-29. *Shoulders and grooves.* Many operations require the grinding of cylindrical pieces that have two or more diameters, radii, fillets, or shoulders. You can sometimes perform these operations by swiveling the wheelhead and using an angular-faced wheel. You can also use a straight wheel with the side recessed to reduce the area of contact with the shoulder. These two methods are illustrated in figure 109. Either of these methods assure a freer cutting action than would be possible with a straight-sided wheel. When you grind straight shoulders with the side of the wheel that is not recessed, you should use a softer grade of wheel to obtain a freer cutting action on the shoulder. The area of contact on the cylindrical position, as shown in figure 109,A, is quite large. Grinding the cylindrical portion of the shoulder will result in some reduction of wheel life. If you must hold to close tolerances on radii or fillets, you will have to use a harder grade wheel with fine grit to prevent rapid breakdown of the wheel face. Naturally, this will reduce the rate of stock removal. When you use the straight wheel method to grind a shoulder, it is desirable to recess the side of the wheel that will grind the shoulder. You can recess a wheel, if a fine finish is required, by using a diamond dresser. If you are not able to move the wheel in close enough to the table to permit the use of the diamond, you can use an abrasive stick and recess the wheel by hand. If a radius is required on the corner of the wheel, you can form it by using the radial attachment or by using the abrasive stick. If you use the abrasive stick, check the radius with a radius gage, *after* stopping the wheel.

12-30. When you use the angular wheel method to grind a shoulder, as shown in figure 109,B, swivel the wheelhead 30° or 45° off center. Both faces of the wheel must be at the same angle and of equal length. The horizontal face of the wheel grinds the diameter, and the vertical face of the wheel grinds the shoulder. If a radius is required at the shoulder, you will have to use the radial grinding attachment or the abrasive stick to obtain the correct radius. You must be very careful to prevent wheel breakdown at the point that grinds the shoulder. If you find it difficult to move the wheelhead crossfeed close enough to the table for the correct depth of cut, you can overcome this difficulty by using the extension spindle. You should always use as narrow a wheel as possible, and it should be at least 7 inches in diameter, hard grade, and fine grain.

12-31. When possible, it is desirable to undercut or recess the work slightly at the shoulder during fabrication. However, some workpieces are so designed that undercutting before being hardened would weaken them or cause cracks to develop during heat treatment. You can grind such a shoulder or a groove by plunge grinding, which is straight-in feed in which no table traverse is involved. You can use plain straight-faced or form-faced wheels to produce any desired form on cylindrical work. The ability of a wheel to hold its form is more important than the rate of stock removal during this type of grinding. For this reason a harder grade of wheel is normally required.

12-32. Assume that you are required to grind a shoulder to specified dimensions on a cylindrical piece of work. You can grind the work on the cylindrical grinding machine, driving it between centers and using an angular wheel. First, mount and aline the extension spindle on the wheelhead. Then select and mount the proper grinding wheel and position the wheel guard. Swivel the wheelhead to the desired number of degrees and aline the table to ZERO with a test bar and dial indicator. Prepare the work for grinding between centers. Compute and set the proper wheel and work speed and set up the diamond dresser. Dress and true the wheel so that both face angles are of equal degrees and lengths. Then remove the dresser and holder. Install a drive dog on the work and mount it between the centers. Turn on the headstock motor, making sure that it revolves the work opposite the wheel direction at the area of contact. Pick up the cut by using the hand crossfeed and then move the wheel slowly to the shoulder with the fine hand feed. Check and adjust for possible taper after making several cuts. Continue using a fine hand feed until you arrive at the specified diameter and length.

12-33. *Internal grinding.* Internal grinding is the grinding of internal circular surfaces. The application of this type of grinding is quite extensive. The range of hole sizes and types of work, as shown in figure 110, is limited only by the capacity of the machine. Internal grinding is a widely used method of finishing internal surfaces, because it is accurate, economical, and produces a good surface. In many instances this method of grinding

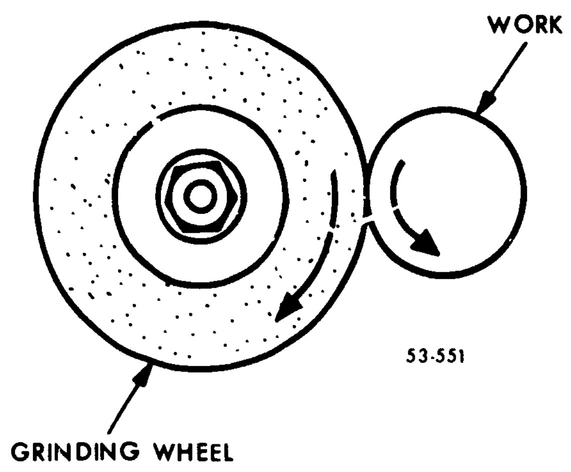


Figure 113. Wheel and cutter rotation for cylindrical grinding.

has taken the place of reaming and boring holes. You will be called upon many times to finish a hole in a hardened metal part because the heat treating process causes a certain amount of distortion. It is then necessary to internally grind the hole to secure an accurate diameter and true surface. Some classes of internal grinding are done on a lathe by using a tool post grinder, which we discussed in Chapter 4 of Volume 2. Internal grinding speeds and feeds are calculated in the same manner as for external cylindrical grinding.

13. Tool and Cutter Grinding Machine Operations

13-1. Tool and cutter grinders are general toolroom grinders that are used primarily to sharpen such items as milling cutters, reamers, and forming tools and to grind a great variety of general machine shop work. As the name implies, they are used primarily for tool and cutter grinding. There are no definite features that distinguish the plain tool and cutter grinder, as shown in figure 111, from the universal tool and cutter grinder which we discussed previously. Most of the features of both machines are identical; however, the universal machine generally has a larger selection of accessories and attachments.

13-2. In this section we will discuss four basic tool and cutter grinder attachments and the grinding of cutting tools.

13-3. **Attachments.** There are many attachments for the tool and cutter grinding machines which permit you to perform numerous operations. There are so many of these attachments that we will not be able to discuss each of them in detail. However, we will discuss four basic grinding attachments: surface, cylindrical, internal, and radial. These attachments will greatly assist you in grinding operations.

13-4. The *surface* grinding attachment can be used to grind flat forming tools, lathe tools, planer tools, flat thread chasers, drifts, chisels, etc. This attachment has a swivel vise with an intermediate support between the vise and the base which permits you to swivel the vise in two planes. This permits the grinding of compound angles.

13-5. The *cylindrical* grinding attachment can be used for all types of straight or taper cylindrical grinding, such as the grinding of reamers, lathe centers, mandrels, and tap or drill shanks. It can also be used for face grinding.

13-6. The *internal* grinding attachment can be used to grind holes in cutters, bushings, arbor collars, etc.

13-7. The *radial* grinding attachment is the most complex and versatile of the many attachments. A typical radial attachment is shown in figure 112. This attachment has a base that is bolted to the machine table. The base supports a work slide on a pivot. You can rotate the slide 360°; however, it is provided with stops, which limit the movement in either direction. You can move the work slide back and forth by thumbscrews located on the back of the attachment. A scale on the side of the slide is graduated for a length of 6 inches in one direction and for 4¼ inches in the other direction. This allows the attachment to accommodate up to a 6-inch convex radius and up to a 1¼-inch concave radius. The workholders will accommodate work up to 5 inches in width and up to 8 inches in diameter. The attachment also has a detachable bracket that you can clamp in front of the work slide to hold a diamond dresser. The dresser can be positioned by the work slide to produce the same radii on the wheel that you can produce on a workpiece.

13-8. **Grinding Cutting Tools.** The working efficiency of a cutter is largely determined by the keenness of its cutting edges. Therefore, it is important to sharpen a cutter at the first sign of dullness. A dull cutter leaves a poorly finished surface. Continued use of a dull cutter leaves it in a condition that will make it necessary for you to grind away a large portion of the teeth to restore the cut-

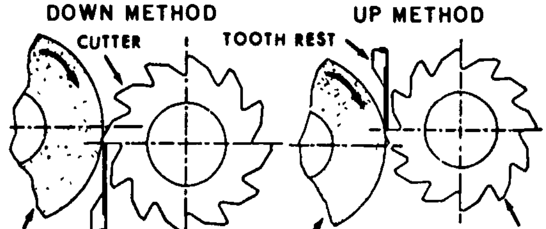


Figure 114. Cutter sharpening positions.

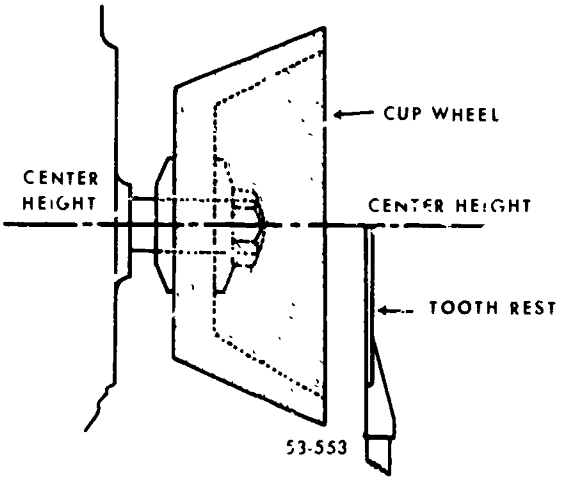


Figure 115. Alinement of tooth rest with axis of cup wheel.

ting edge. When you maintain a cutter in good working condition by frequent sharpening, it will cut rapidly and effectively at all times. When such a cutter does need sharpening, you will have to grind the teeth only a very small amount to insure keen cutting edges. In this section we will discuss grinding cutting tools, cutting tool clearance, grinding form cutters, and grinding convex and concave radii.

13-9. *Grinding cutters cylindrically.* Various types of cutting tools, such as reamers and milling cutters, are ground cylindrically. This is done to remove warpage from heat treatment, remove nicks, obtain a specific diameter, or to produce a finish and a slight clearance on the cutting edges of the teeth. When you grind these tools cylindrically, the work is rotated in the opposite direction from that ordinarily used in cylindrical grinding. If a clearance is desirable on the cutting edges, the movement of the wheel and the work should be in the same direction at the area of contact, as shown in figure 113. Mount the cutter so that the heel of the tooth strikes the wheel first. In theory this will cause a slight spring between the work and wheel, which in turn will cause the heel of the tooth to be ground slightly lower than the cutting edge. The clearance will vary in amount, depending upon the rigidity of the cutting tool being ground and the work setup. The work can be held for the cylindrical grinding operation in three ways: between centers, on a mandrel, or on a stub arbor mounted in the headstock spindle. You should normally select a medium grade grinding wheel for the cylindrical grinding of hardened high carbon steel and high-speed steel cutters.

13-10. After you have cylindrically ground cutters or reamers to restore concentricity, you may use either of two methods to sharpen the cutting edges of the teeth and to provide clearance. These methods depend upon the rotation of the grinding wheel in relation to the cutting edge. Figure 114 illustrates two methods of straight grinding wheel setup. In method A the wheel rotation is from the body of the tooth off the cutting edge. The wheel rotation holds the cutter on the tooth rest but will raise a burr on the cutting edge, which must be removed by stoning. More seriously, this method may have a tendency to draw the temper from the cutting edge. In method B the wheel rotation is from the cutting edge toward the body of the tooth. In this method there is less danger or burning the tooth, but you must exercise greater care to hold the cutter on the tooth rest. If the cutter turns while you are grinding it, the tooth will be ruined.

13-11. Cup wheels, as shown in figure 115, are also used extensively to grind cutters and reamers in a manner similar to that used with straight wheels. You must exercise greater care when you use a cup wheel because the area of contact between wheel and work is larger.

13-12. *Cutting tool clearance for profile cutters.* Correct clearance in back of the cutting edge of any tool is highly essential. Insufficient clearance will cause the teeth to drag, resulting in friction and slow cutting. Too much clearance will result in chatter and in rapid dulling of the teeth. The cutting edge must have strength; the correct clearance will produce this strength. Figure 116 shows a typical cutter tooth and the various angles which are produced by grinding. A secondary clearance of 9° to 30°, depending upon the design of the cutter, produces a strong tooth and provides easy control of the width of the cutting land. In this case, the width of land, which is the area immediately behind the cutting edge should be $1/32$ inch to $1/16$ inch, depending upon the diameter of the cutting tool. When the cutting land becomes too wide from many sharpenings, you must grind secondary clearance to restore the land width to its correct dimension. The secondary clearance is produced by properly locating the wheel, cutter, and tooth rest. There are several setup methods, depending upon the type of wheel used, the shape of the work, and the location of the tooth rest. The wheel may be either a plain straight wheel or a cup wheel. The work may be straight or tapered and may have straight or helical teeth. The tooth rest may be located on either the wheelhead or on the table. The ends of the tooth rest will vary in shape for different cutters. When you use a straight wheel, the clearance angle depends upon the diameter of the grinding wheel. When you use the cup wheel, the diameter of the *cutter* is the determining factor. To determine the setting for a cutter when you use the straight wheel method, multiply the clearance angle of the cutter in degrees times the

Material	Clearance angle A (degrees)	Clearance angle B (degrees)	Width of Land D (inches)
Low carbon steel	5 to 7	30	
High carbon steel and tool steel	3 to 5	30	
Steel castings	5 to 7	30	
Cast iron	4 to 7	30	Small cutters 1/64
Cast brass	10 to 12	30	Medium cutters 1/32
Soft bronze	10 to 12	30	Large cutters 1/16
Medium bronze	6 to 7	30	
Hard bronze	4 to 5	30	
Copper	12 to 15	30	
Aluminum	10 to 12	30	

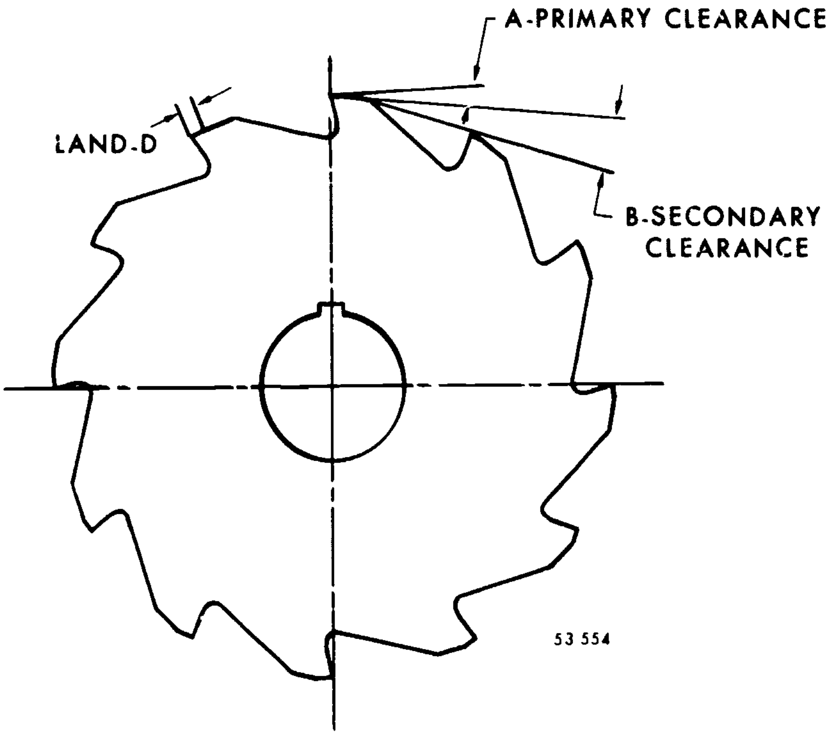


Figure 116. Recommended clearance angles.

wheel diameter in inches times the constant 0.0088. The constant 0.0088 is derived by meas urement of the distance of 1° on the circumference line of a 1-inch circle. Tne result will be the amount in thousandths of an inch that you raise or lower the cutter and tooth rest to obtain the proper clearance. When you use the cup wheel method, you obtain the setting by multiplying the clearance angle in degrees times the *cutter* diameter in inches times the constant 0.0088.

13-13. The tooth rest is normally fastened to the table when you sharpen straight-tooth cutters. However, when you sharpen helical-tooth cutters mounted between centers, the tooth rest must be mounted on the wheelhead so that the work will revolve and follow the angle of helix on the teeth. You can calculate the distance to raise or lower the wheelhead or tooth rest by referring to machinists' publications, such as the *Machinery's Handbook,* which normally have charts from which you can obtain the recommended clearance angles for most cutters.

13-14. *Sharpening formed cutters.* Formed or eccentrically relieved cutters, such as gear cutters and convex or concave cutters, cannot be sharpened in the same manner as the profile type cutters. Formed cutters have a definite shape that must be retained through many sharpenings. To retain this shape, you must grind the face of the teeth with a radial rake. Any positive or negative

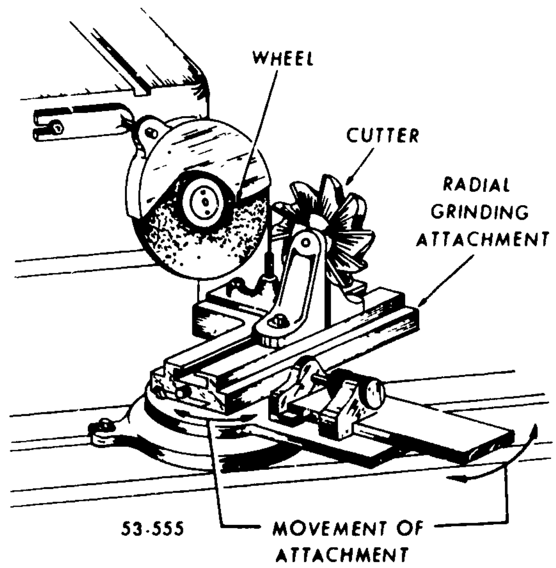


Figure 117. Radial grinding setup.

rake on formed cutter teeth will change their shape. Radial rake means that the faces of the teeth are in a plane passing through the axis of the cutter. You may sharpen formed cutters by using the infeed type of formed cutter attachment. Before you can grind a cutter by the infeed method, you may have to grind the back of the teeth so that the tooth width remains the same for each of the teeth. You could also mount the formed cutter on a mandrel or a stub arbor and sharpen the cutting edges while you use the index head to accurately position the cutter head to help grind the teeth on the solid- or inserted-tooth side of face milling cutters. This is done by setting the head at a slight angle to make the cutter teeth taper toward the center of the cutter. This angle provides clearance and prevents the cutting edge from dragging. You can use either a straight or cup wheel. The cup wheel will produce better results.

13-15. *Grinding convex and concave radii.* As previously mentioned, you can grind convex or concave surfaces with the aid of the radial grinding attachment shown in figure 112. You can use this attachment with the workholders to produce a convex radius on milling cutters, as shown in figure 117. In addition to holding work for grinding, you can use the attachment to form the circumference of a grinding wheel into a convex or concave radius. You can do this by mounting a diamond dresser on the attachment, which will then move the diamond through the desired arc or radius and dress and true the grinding wheel to shape. When you are forming a wheel to a convex shape, the wheel width should be twice the desired radius. If the wheel is wider than twice the radius, a shoulder will result. If it is narrower, you cannot produce a full radius. When you are forming a wheel into a concave radius, the wheel width should be twice the desired radius plus ⅛ inch on each side to give added strength to the edges. A typical grinding operation in which you would use a convex-shaped wheel is the sharpening of tap flutes.

Fitting and Assembly

YOU MUST BE able to fit and assemble many of the items that you will be required to make. Polishing and buffing are often necessary in fitting and assembling parts. Sometimes machined parts will require reworking. In addition to these subjects we will discuss nondestructive inspection methods in this chapter.

2. The fitting and assembly of machined parts involves many different operations. Primarily, it includes all the machine operations that go into the fabrication of the various parts making up an assembly. However, in this text we are assuming that all parts have been previously machined. You are now ready to put together the assembly by lapping or scraping parts to fit, or by fastening it together with fastening devices, such as nuts, bolts, and screws. When you are assembling only one device or mechanism, you are not too interested in the interchangeability of parts. But when several like devices are assembled, you would be interested in the interchangeability of the parts making up the device. Therefore, you should inspect the parts for conformity to the blueprint, not only as to dimensions, but also as to the notes on Rockwell hardness, magnaflux inspection, or plating. Often a part may be wrong to a minor degree in a way that will not interfere with its interchangeability and it can still be used. This is a point at which good judgment enters into the rejection or acceptance of the part. Too much stress cannot be applied to the checking of such things as chamfers, countersunk or counterbored holes, reliefs, radii, and measurements. Another factor to consider in checking parts that are for close dimensions is the material of which they are made. Aluminum, magnesium, and brass have different contraction and expansion characteristics. High-grade steel is the least susceptible to temperature changes. The other grades of steel are much more susceptible to changes in temperature. Parts to be inspected for close dimensions should be inspected in normal room temperature of about 68° F. for best results.

14. Machine Fits

14-1. The fitting of machined parts requires a knowledge of the types of fits and the reworking of parts. Many factors must be considered in selecting fits, because one particular application may not be effective in all situations. Some of these factors include bearing load, temperature, lubrication, materials used, and the speed of moving parts. At times, the length of engagement between the workpieces must also be considered.

14-2. **Types of Fits.** You will use various types of fits for mating parts. The type of fit that you select will depend upon the intended use of the parts. The following information pertains to some of the common fits and their uses. Refer to chart 10 for the allowances which are recommended for the various fits.

a. Standard. The standard fit is a general purpose fit. Use a standard fit when parts must be assembled easily and when a special purpose fit is not required.

b. Revolving or running. Use a revolving or running fit when an internal part revolves within an external part or when an external part revolves around a stationary internal part. The internal part is always smaller in diameter than the external part.

c. Sliding. Use a sliding fit when a part slides within another part. The tailstock spindle of a lathe is an example of a sliding fit.

d. Drive. Use a drive fit to secure bushings in sleeves, pulleys, etc., and to assemble parts when other holding methods are impractical. The internal part is slightly larger than the hole in which it fits. The mating parts can be assembled with light hammer blows or with the aid of a vise.

e. Force. A force fit is similar to a drive fit except that the difference between the mating parts is greater. Use an arbor press or other mechanical device to assemble the parts.

f. Shrink. Use a shrink fit to obtain the maximum grip between two parts, such as a flywheel and the ring gear that fits around it. The opening

CHART 10

ALLOWANCE FOR VARIOUS FITS

RUNNING FITS

Diameter, Inches	For Shafts with Speeds Under 600 R P M. Ordinary Working Conditions Allowances, Inches	For Shafts with Speeds Over 600 R P.M Heavy Pressure—Severe Working Conditions Allowances, Inches
Up to ½	−0.0005 to −0.001	−0.0005 to −0.001
½ to 1	−0.00075 to −0.0015	−0.001 to −0.002
1 to 2	−0.0015 to −0.0025	−0.002 to −0.003
2 to 3½	−0.002 to −0.003	−0.003 to −0.004
3½ to 6	−0.0025 to −0.004	−0.004 to −0.005

STANDARD FITS

Diameter, Inches	For Light Service where Part is Keyed to Shaft and Clamped Endwise—No Fitting Allowances, Inches	With Play Eliminated—Part Should Assemble Readily—Some Fitting and Selecting May be Required Allowances, Inches
Up to ½	Standard to −0.00025	Standard to +0.00025
½ to 3½	Standard to −0.0005	Standard to +0.0005
3½ to 6	Standard to −0.00075	Standard to +0.00075

SLIDING FITS

Diameter, Inches	For Shafts with Gears, Clutches or Similar Parts which Must be Free to Slide Allowances, Inches
Up to ½	−0.0005 to −0.001
½ to 1	−0.00075 to −0.0015
1 to 2	−0.0015 to −0.0025
2 to 3½	−0.002 to −0.003
3½ to 6	−0.0025 to −0.004

DRIVING FITS

Diameter, Inches	For Permanent Assembly of Parts so Located that Driving Cannot be Done Readily Allowances, Inches	For Permanent Assembly and Severe Duty and where there is Ample Room for Driving Allowances, Inches
Up to ½	Standard to +0.00025	+0.0005 to +0.001
½ to 1	+0.00025 to +0.0005	+0.0005 to +0.001
1 to 2	+0.0005 to +0.00075	+0.0005 to +0.001
2 to 3½	+0.0005 to +0.001	+0.00075 to +0.00125
3½ to 6	+0.0005 to +0.001	+0.001 to +0.0015

FORCED FITS

Diameter, Inches	For Permanent Assembly and very Severe Service—Hydraulic Press Used for Larger Parts Allowances, Inches
Up to ½	+0.00075 to +0.001
½ to 1	+0.001 to +0.002
1 to 2	+0.002 to +0.003
2 to 3½	+0.003 to +0.004
3½ to 6	+0.004 to +0.005

53-556

in the outer part is smaller than the part it fits. Heat the outer part, which causes it to expand, and then place it on the inner part. When the outer part cools, it will shrink and grip the inner part tightly.

g. Expanding. Use an expanding fit when a shrink fit is desired but the nature of the work will not permit the outer member to be heated. Shrink the inner part by cooling it with solidified carbon dioxide (dry ice) or liquid nitrogen. Then place the cooled part, such as a valve seat, in the mating part and allow it to return to its normal temperature. As it warms up it will expand and tighten against the mating part.

14-3. **Reworking Parts.** Parts that are to be assembled must be free of burrs, sharp corners, and high spots. Parts are reworked so that they can be assembled properly and so that they will operate without binding. You can remove high spots by (1) filing, (2) using abrasive cloth or stones, as shown in figure 118, (3) hand scraping, or (4) polishing and buffing. In extreme cases you may even have to set up the work in an appropriate machine and remove additional material by machining. You are already familiar with filing and the use of abrasives. Therefore, we will limit this discussion to polishing and buffing.

14-4. **Polishing and Buffing.** Polishing and buffing are sometimes considered to be identical operations. However, a distinction should be made between them. In polishing the abrasive is glued to the wheel, and in buffing the abrasive is applied to the wheel during the buffing operation. Buffing is a finer process than polishing.

14-5. *Polishing*. Machine polishing is the process of removing a slight amount of material from the work by means of cushion type wheels and flexible belts. The abrasive grains are secured by glue or other cementing materials. Polishing wheels are driven at a relatively high speed. Figure 119 shows a typical polishing and buffing machine. You may be required to do the following types of polishing:

a. Rough polishing on a dry wheel with a coarse abrasive up to 60 grain size.

b. Dry finish polishing on a dry wheel with a fine abrasive 70 to 120 grain size.

c. Oil polishing with a fine abrasive 120 or finer grain size, in which a grease, such as tallow, holds the abrasive. This type is nearly the same as buffing and is sometimes substituted for a buffing operation.

14-6. Polishing wheels are made from a wide variety of materials. The type of wheel you use for

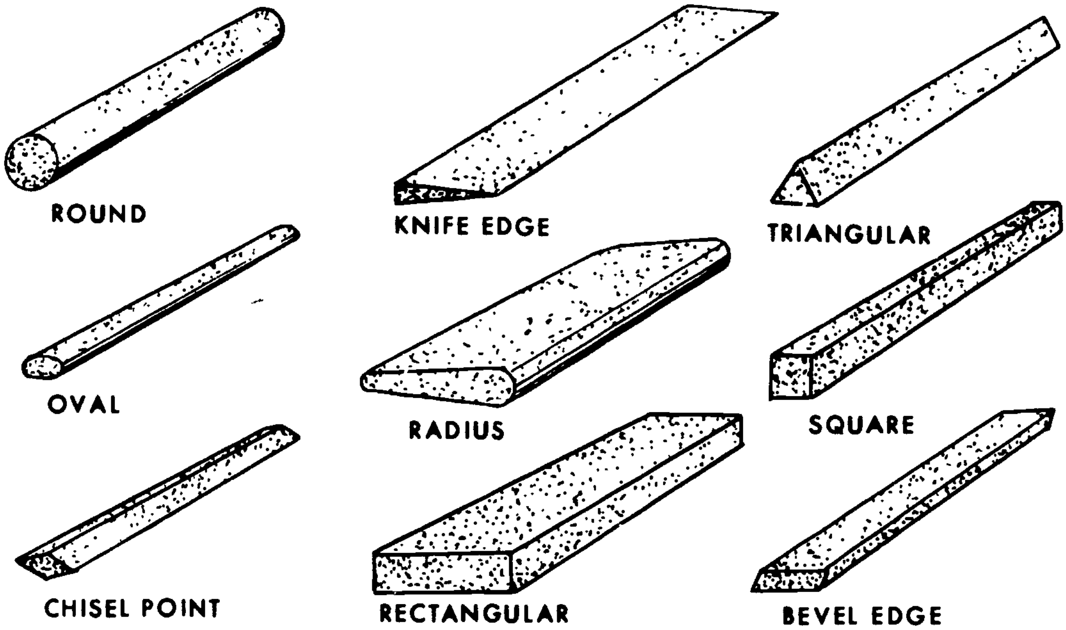


Figure 118. Abrasive stones.

a specific operation will depend upon such factors as the shape of the work, type of metal, abrasive grain size required, and desired finish. To do the job efficiently, you must use a round wheel that is properly balanced. Some of the most common types of wheels are as follows:

a. Sewed canvas—widely used for severe polishing. The grain size varies from 24 to 46. It is made of canvas discs sewed together.

b. Disc canvas—also made of canvas discs, but they are bonded together with glue. Use this wheel for heavy polishing.

c. Compress—especially adapted for polishing irregularly shaped work. It is generally made of canvas or leather and comes in hard, medium, or soft grades.

d. Felt—used to obtain a high luster on steel, iron, brass, or aluminum castings. The grain size is 150 or finer.

e. Sewed buff—a soft, flexible wheel used for rough and fine polishing of brass, aluminum, and steel sheets.

f. Sheepskin leather—also a soft, flexible wheel. It is widely used to polish brass, aluminum, and corrosion-resistant steel sheets.

g. Steel wire—used mainly to remove paint, corrosion, scale, etc., and for rough polishing. Al-

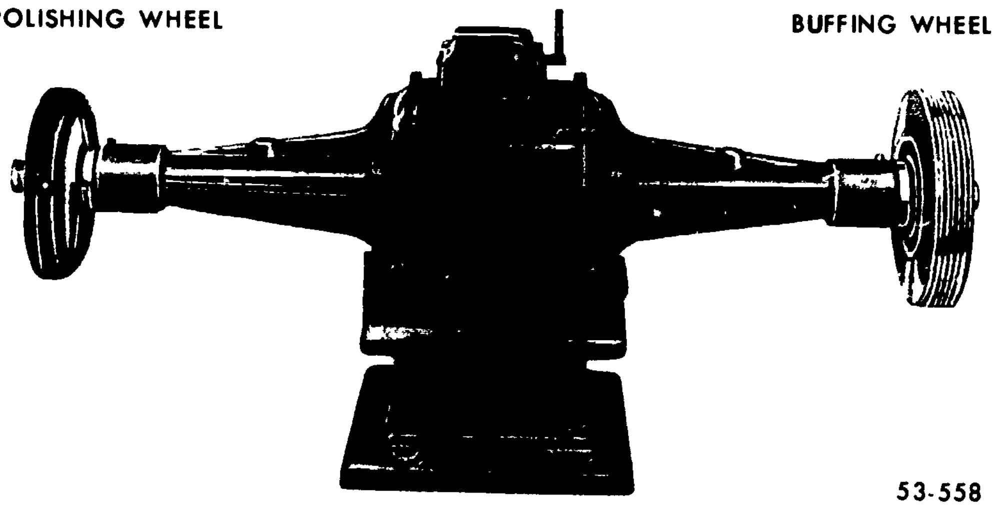


Figure 119. Bench polisher.

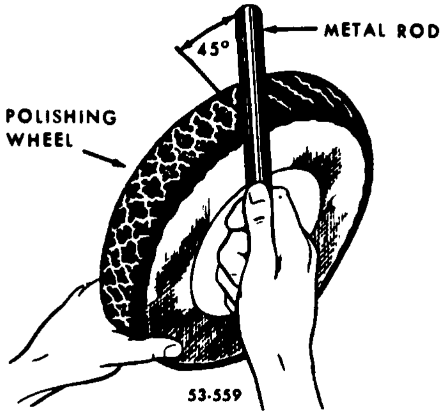


Figure 120. Breaking a polishing wheel.

though an abrasive is not used, it is still classed as a polishing wheel.

14-7. Before using a brand-new nonmetallic polishing wheel, which is usually too stiff, you should "break" it along both sides of the entire circumference with a metal rod, as shown in figure 120. This will give the wheel enough flexibility for proper use.

14-8. *Buffing.* Buffing is a smoothing operation that is achieved more by plastic flow of the metal than by abrasion. The primary purpose of buffing is to produce a high luster of color, not to obtain accurate dimensions. There are two major buffing operations:

a. Cutdown. Cutdown buffing produces a rapid smoothing action by means of fast cutting compounds (usually tripoli or silica) and relatively hard buffs. You buff with a high pressure and a high speed, allowing both abrasion and plastic flow to occur. The grain size is usually 120 or finer.

b. Color. The purpose for color buffing is to produce a high luster or surface finish on the work. Use a soft abrasive, such as crocus (iron oxide) or lime. The grain size should be from 400 to 600.

14-9. In buffing you apply the abrasive to the buffing discs prior to or during the buffing operation. The buffing compound consists of a grease binder and an abrasive. The binder is a mixture of tallow and stearic acid with other materials added. You hold the buffing compound firmly against the cloth on the wheel as it rotates until the cloth is saturated. You may apply more compound from time to time to improve the buffing action of the wheel. The heat generated by the friction softens the tallow, which spreads out and distributes the abrasive grains evenly over the face of the wheel. Tripoli, a form of natural silicon oxide, is the best abrasive for cutdown buffing. When you want sharp, even buffing, use emery paste or emery cake. Use aluminum-oxide and chromium-oxide compounds to color chromium or corrosion-resistant steels; crocus to color nickel and brass; and a lime composition to produce a high luster on nickel, brass, and bronze.

15. Assembly of Machined Parts

15-1. The fitting and assembly of machined parts may involve the use of standard parts, the use of fixtures, precision measuring, and gaging processes. Often you will have to locate and drill holes for bolts, screws, and pins used in assembly.

15-2. **Standard Parts.** You should, whenever possible, use standard fastening devices and bearings in the assembly of machined parts. Standard parts are usually readily available. Therefore, the number of parts that must be manufactured is reduced. This reduces the cost. The following information will help you to select the proper types of fastening devices and bearings.

15-3. *Fastening devices.* The most common devices for fastening parts together are bolts, screws, nuts, and pins.

a. Bolts. A bolt is an externally threaded fastener whose threaded and unthreaded portions are the same nominal diameter. The unthreaded portion is known as the grip. A bolt head can be square, hexagonal, domed, or flat circular countersunk shaped, as shown in figure 121. Bolts must be assembled with a nut in order to serve as a fastener. A stud can be considered a type of bolt when it is installed in the parent metal.

b. Screws. A screw is an externally threaded fastener whose shank is generally threaded along its entire length. Most screws, except those manufactured for use in aircraft and missiles, have a lower material strength and a looser thread fit than a bolt of the same size. Screws differ from bolts mainly in that they are torqued by their heads into

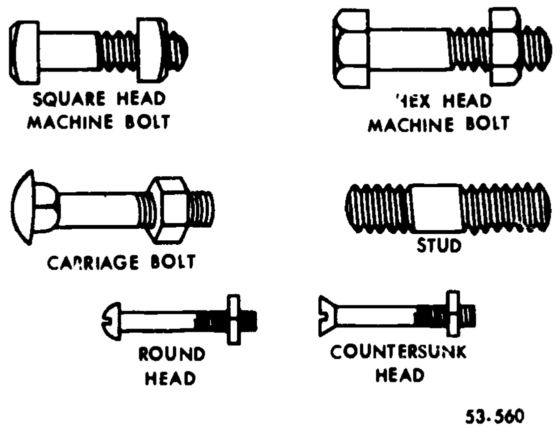


Figure 121. Bolts.

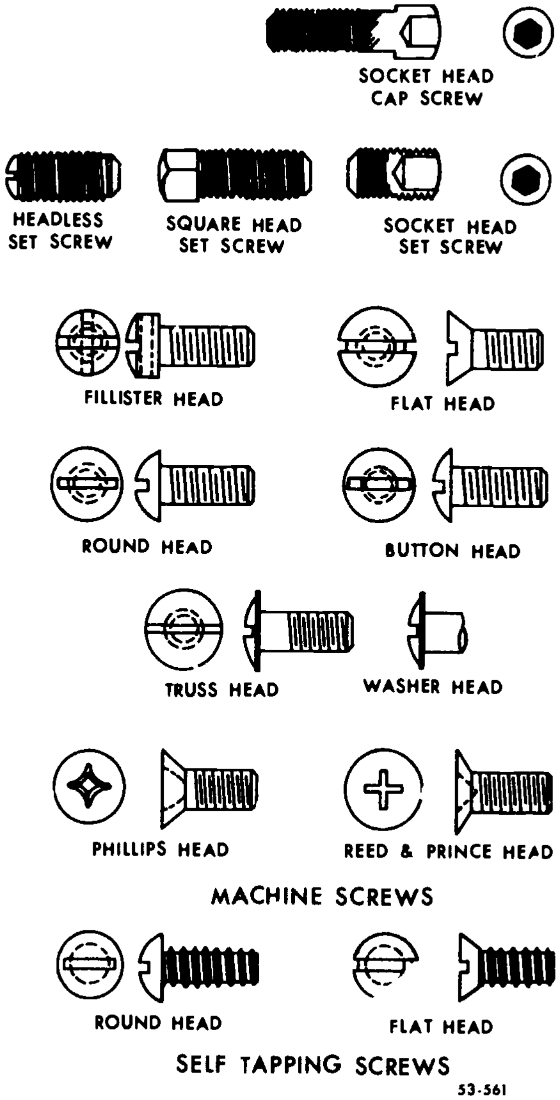


Figure 122. Screws.

a tapped hole. The three main groups of screws are machine, structural, and setscrews. There is one other group, known as self-tapping, that forms its own mating threads in the part as its head is torqued. All four of these groups of screws can have the same style heads as bolts. These heads can have either a common screwdriver slot or a cross-point recess in them.

NOTE: There are two types of cross-point recesses: Phillips, and Reed & Prince. The screwdriver for one type should never be used in a screw of the other type, because the recess will be damaged.

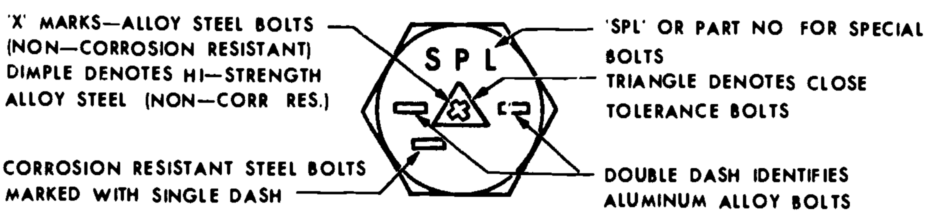
Screws are usually described according to the shape of the heads. See figure 122 for illustrations of the more common types of screws. Because of the possibility of someone accidentally replacing a high-strength bolt with a weaker one, the Defense Department and aircraft industry devised a symbol system to mark the heads of bolts and screws. Some of these symbols, as shown in figure 123, indicate the type of metal of which the fastener is constructed, whether it is corrosion resistant, or if it is close tolerance. For further information, you should consult Technical Order 1–1A–8, *Structural Hardware.*

c. Nuts. A nut is a square or hexagonal-shaped piece of metal with a threaded hole through the center so that it can be mated to a bolt or stud. The threads of a nut must correspond to the threads on the bolt or they will both be damaged. There are many types of nuts. The most common ones are illustrated in figure 124. The castle, shear, and slotted engine nuts all require the use of a cotter pin to prevent them from backing off the bolt after installation. Self-locking nuts do not require any assistance to hold their position on a bolt.

NOTE: A fiber insert self-locking nut, as shown in figure 125, is used on an aircraft or missile only one time. Once it is removed from the bolt, it is no longer self-locking.

Nuts are made of almost all metals, but the most common ones are made from steel, aluminum, and brass.

d. Pins. Pins are used to retain parts in a fixed position and to maintain alinement. The most common types are the flathead, plain and threaded taper, cotter, straight, and spring pins, as shown in figure 126.



53—563

Figure 123. Bolt head markings.

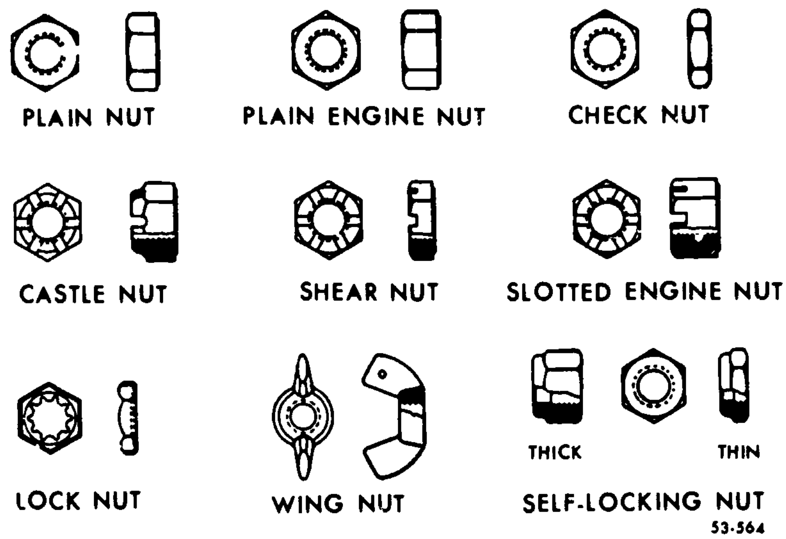


Figure 124. Nuts.

(1) Flathead pins, sometimes called clevis pins, are used on secondary connections that do not operate continuously. This type of pin should be secured with a cotter pin and installed with the head up, if possible, as shown in figure 127,B.

(2) Taper pins are used in joints that carry shear loads where the absence of clearance is essential. They should also be used for parts that have to be taken apart frequently, since their removal does not damage the hole. When parts must be kept in absolute alinement, use taper pins. There are two styles of taper pins; one without threads that is secured with safety wire and one with threads that is secured with a nut.

(3) Cotter pins are used to secure bolts, screws, nuts and other pins, as shown in figure 127,A, and should not be used more than once.

(4) Straight pins are used to keep parts in alinement or in a fixed position. They are solid, with a small chamfer on one or both ends, which enables the pin to slide into the hole easily. These pins are sometimes hardened to increase their strength.

(5) Spring pins are slotted or coiled, chamfered, heat treated, and manufactured with a slightly greater diameter than the holes in which they are to be used. They are stronger than mild carbon steel straight and taper pins. The main advantages of spring pins are that they do not need securing and will not loosen by vibration. This is because of the continuous spring pressure exerted against the sides of the hole by the tendency of the pin to spring apart. Spring pins can also be used one inside another to increase shear strength

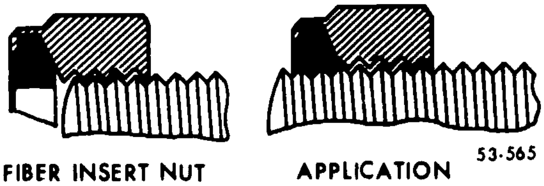


Figure 125. Fiber insert nut.

15-4. *Bearings*. Bearings support or guide various moving parts of machinery, such as shafts. There are two types of bearings: plain and antifriction.

a. Plain. Plain bearings are generally made from material such as bronze, which is hard enough to wear well, but softer than the material that it contacts. When a plain bearing is pressed into a hole, it is known as a bushing. Bushings can be plain or flanged, as shown in figure 128, and are replaced when they become worn. Another type of plain bearing is the thrust bearing, which is designed to resist movement of a shaft toward either end. Plain bearings depend upon lubricants to reduce friction and wear. There are also special

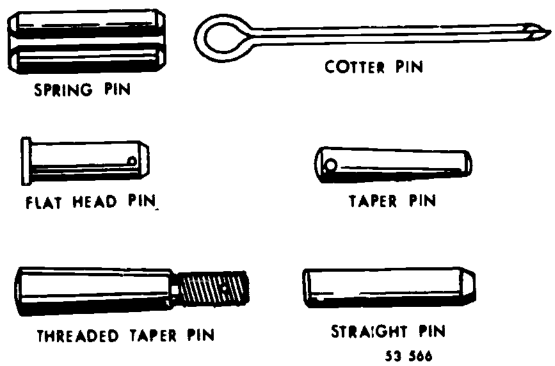


Figure 126. Pins.

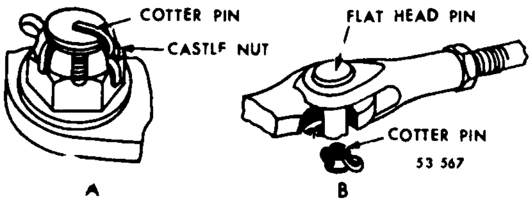


Figure 127. Application of cotter pin, castle nut, and flathead pin.

bearing materials that have a lubricant impregnated into the pores of the metal. Bronze that has been impregnated with oil is known as "oilite" and is very useful for bearings that cannot be lubricated regularly.

b. Antifriction. Ball and roller bearings are the most common types of antifriction bearings. Very little friction is present in these bearings because of the small area of contact between the balls or rollers and the bearing races and because of the rolling, rather than sliding, action.

(1) Ball bearings of the type shown in figure 129 consist of an outer and an inner race, which are separated by freely moving balls; they are primarily intended to resist radial thrust. Ball bearings of the type shown in figure 130 are used to resist end thrust.

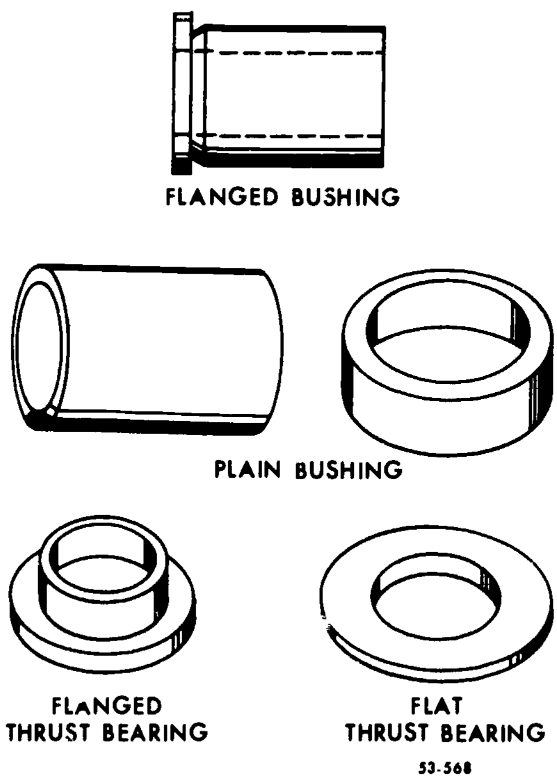
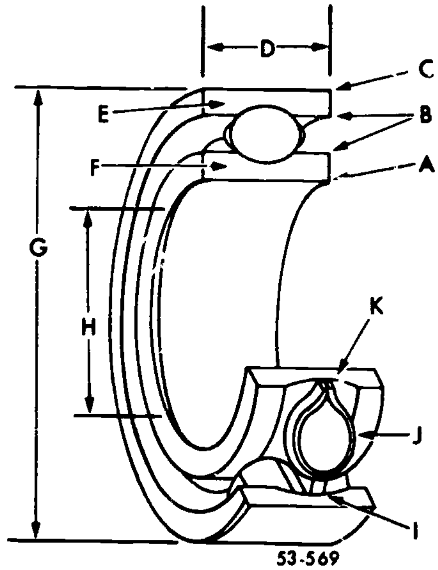


Figure 128. Bushings and bearings.

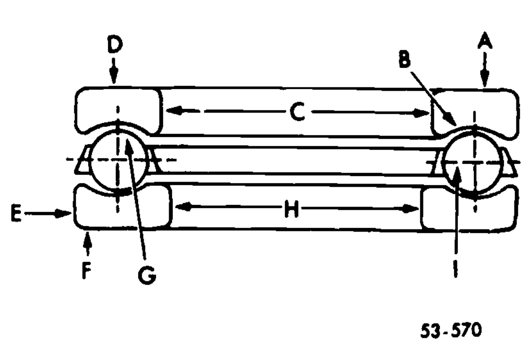


A. CORNER RADIUS
B. SHOULDERS
C. CORNER RADIUS
D. WIDTH
E. OUTER RING
F. INNER RING
G. OUTSIDE DIAMETER
H. BORE
I. OUTER RING BALL RACE
J. SEPARATOR
K INNER RING BALL RACE

Figure 129. Ball bearings.

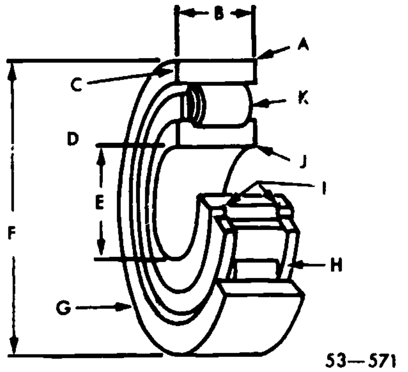
(2) Roller bearings, shown in figures 131 and 132, are similar to ball bearings except that rollers are used instead of balls. The rollers may be cylindrical, tapered, or barrel shaped. Bearings with rollers that are relatively long compared to the roller diameter are called needle bearings. Tapered roller bearings are usually used to combat extremely high end thrust. Additional information pertaining to fasteners and bearings can be found in publications such as the *Machinery's Handbook*, and TO 1–1A–8, *Structural Hardware*.

15-5. **Jigs and Fixtures.** Jigs and fixtures are very similar workholding devices. The principal



A. FACE
B. RACEWAY DEPTH
C. LARGE BORE
D. LARGE BORE WASHER
E. SMALL BORE WASHER
F. FACE
G. GROOVED RACEWAY
H. SMALL BORE
I. BALL

Figure 130. Ball type thrust bearing.



STRAIGHT
ROLLER BEARING

A. CORNER RADIUS
B. WIDTH
C. OUTER RING
D. INNER RING
E. BORE
F. OUTSIDE DIAMETER
G. FACE
H. SEPARATOR
I. SHOULDERS
J. CORNER RADIUS
K. ROLLER

Figure 131. Straight roller bearing.

difference is that a jig not only can hold the work, but it guides the cutting tool. Using jigs and fixtures can save time when many like items require the same machining operations. Their use eliminates having to lay out each piece individually. Once the jig or fixture has been fabricated, it can be used by less skilled personnel to produce accurate work.

15-6. *Jigs.* The two most common types of jigs are the drill jig and the boring jig. Drill jigs are not only used for drilling, but can also be used for reaming, tapping, counterboring, and countersinking. You will use most drill jigs in a drill press, but some will be designed to be used directly on the equipment to be modified. Boring jigs are used on milling machines for boring large holes that would not be practical to drill. They are also used when extreme accuracy or hole alinement is required.

15-7. *Fixtures.* The class of a fixture is determined by the machine on which the fixture is to be used. The type is determined by the construction and features of the fixture. Some of the more common types of fixtures include the angle plate and the jaws of vises and chucks. Other types of fixtures include the gang fixture, which holds many identical items for machining in the same pass of the cutter. The duplex fixture is usually mounted on a milling machine and designed so that while one part is being machined another part is being loaded or unloaded. In this way, there is a minimum of time lost waiting for the machine to finish the cut before inserting a new piece. You can find additional information on jigs and fixtures in the *Machinery's Handbook* and other technical publications.

15-8. **Precision Measuring and Gaging.** A gage is a device which is used to determine whether or not the dimensions of a manufactured part are within specified limits. Fixed gages are used more than any other type. Fixed gages may be subdivided into these general classes: ring, receiving, plug, pin, snap, length, flushpin, thread, form, and gage block.

a. Ring. A ring gage has a circular hole that is ground accurately to a specified size. Ring gages are frequently used in pairs. The difference in the hole sizes of the two gages is equal to the tolerance of the parts being machined. A pair of ring gages is known as go-no-go gages. If a part fits into the larger gage but does not fit into the smaller gage, it is within tolerance. A part that fits into the smaller gage is too small and is not acceptable. A part that does not enter the larger ring gage is too large and must be remachined.

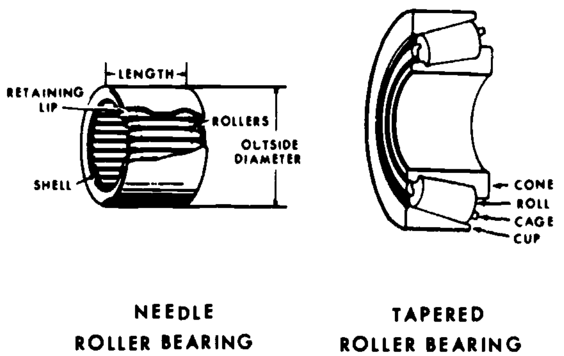
b. Receiving. Receiving gages are similar to ring gages. They are used to check the size and contour of noncircular parts. They are used quite extensively to check splined shafts.

c. Plug. A plug gage has an outside gaging surface that is shaped to fit a hole. It may be round, tapered, or irregular in shape. It may have either an integral or a replaceable handle. Like ring gages, they may be used in pairs as go-no-go gages.

d. Pin. Pin gages are used for measuring large holes when a plug gage may be too heavy. The pin gage is placed lengthwise across the hole and the measurement is made in a manner similar to measuring with an inside micrometer. Pin gages may also be used to measure the width of slots and grooves.

e. Snap. Snap gages are gages with inside measuring surfaces for checking diameters, lengths, thicknesses, and other similar dimensions.

f. Length. A length gage is a special device that is designed to replace a machinist rule. This type



NEEDLE
ROLLER BEARING

TAPERED
ROLLER BEARING

Figure 132. Needle and tapered roller bearings.

of gage is used when a large number of parts is to be made, since the workpieces can be checked quickly.

g. Flushpin. Flushpin gages are used for gaging special shapes that may be difficult to check by conventional methods. They are primarily used to gage the depth of slots.

h. Thread. Thread gages are made in the form of ring or plug gages. This gage has definite limitations for precise measurement. Single full-form plug or ring screw gages cannot be used to check all the thread elements. At least three sets of plug and ring gages are required to check a screw thread completely. One set is used to check the pitch diameter, the second to check the major diameter, and the third to check the minor diameter.

i. Form. As a rule, the ring and plug gages are used for checking ordinary work. When greater accuracy is required, a method such as contour projection is employed. Form gages are specially designed to check the form or contour of a workpiece. An example of a form gage is a gage that is used to check the angles and shape of a lathe tool.

j. Gage block. Gage blocks are used to check measuring instruments, to set indicating gages, and to locate work. The gage blocks commonly known to all machinists are the Johansson gage blocks. They are made of special steel to resist wear and temperature changes. Gage block sets vary in their accuracy. Some sets are accurate within 0.0001 inch and others may be accurate within 0.000002 inch. The accuracy (and the price) of a set of blocks is determined by the precision with which they are made. These blocks have such a fine, smooth finish that they will adhere to each other after being firmly pressed together. Gage blocks are used extensively to set up sine bars.

15-9. **Nondestructive Inspection Methods.** The visual inspection of material, parts, and complete units is no longer the most important method of determining their condition. Various nondestructive inspections are now used to detect variations in structure, changes in surface finish, and the presence of such physical discontinuities as cracks. Although Nondestructive Inspection, AFSC 536X0, is now a separate ladder of the metalworking career field, you should have a basic understanding of nondestructive inspection methods.

15-10. *Penetrant inspection.* Penetrant inspection is used to inspect nonporous materials for defects open to the surface. Surface defects are all types of cracks in connection with welding, grinding, fatigue, forging, seam laps, or poor bonding between two metals. Penetrant methods are restricted to the location of defects open to the surface. The main types of penetrant inspection are dye and fluorescent.

a. Dye penetrant method. Advantages of this method are that it provides a fast, on-the-spot inspection during overhaul or shutdown periods, and the initial cost of the test is relatively low. A perfectly white or blank surface indicates freedom from cracks or other defects that are open to the surface. Disadvantages are that it is not practical on rough surfaces and the color contrast may be limited on some surfaces.

b. Fluorescent penetrant method. Advantages of this method are that the test is positive even on rough surfaces, the procedure is easy, and the location and size of the defects are marked for visual inspection. Disadvantages are that an ultraviolet light is needed and only clean defects open to the surface can be detected.

15-11. *Magetic particle inspection.* Certain materials have the property of attracting iron and steel. This property of attraction is called magnetism. Some metals which have this property, besides iron and steel, are nickel and cobalt and some of their alloys. Magnetic particle inspection reveals surface or near surface defects. Magnetic particles tend to adhere to the surface of a magnetized object only at points where discontinuities (such as cracks) are located. This method is not suited for very small deep-seated defects. The deeper the defect is below the surface, the larger it must be to show up. With magnetic particle inspection, the surface to be inspected must be available to the inspector. This means that shafts or other equipment cannot be inspected without removing pressed wheels, pulleys, or bearing housings. This method has several advantages. It can be used on any magnetic type material, and it is a positive method of finding cracks at the surface.

15-12. *Eddy current inspection.* Eddy current inspection is used to detect surface, or near surface, defects in most metals, to identify metals, and to detect fire damaged areas. It can be applied to airframe parts or assemblies where the defective area is accessible to contact by the eddy current probe. This type of inspection is highly sensitive.

15-13. *Ultrasonic inspection.* Ultrasonic inspection is used to detect surface and subsurface defects, such as cracks, lack of bond, laminations, and porosity. It can also be used for gaging thickness, detecting corrosion and detecting air leaks in pressurized systems. One big advantage of this method is that it can be used to inspect both metallic and hard nonmetallic materials.

15-14. *Radiography.* Radiography includes X-ray and gamma ray inspection. X and gamma radiations, because of their unique ability to penetrate material and disclose defects, have been applied to the radiographic inspection of castings,

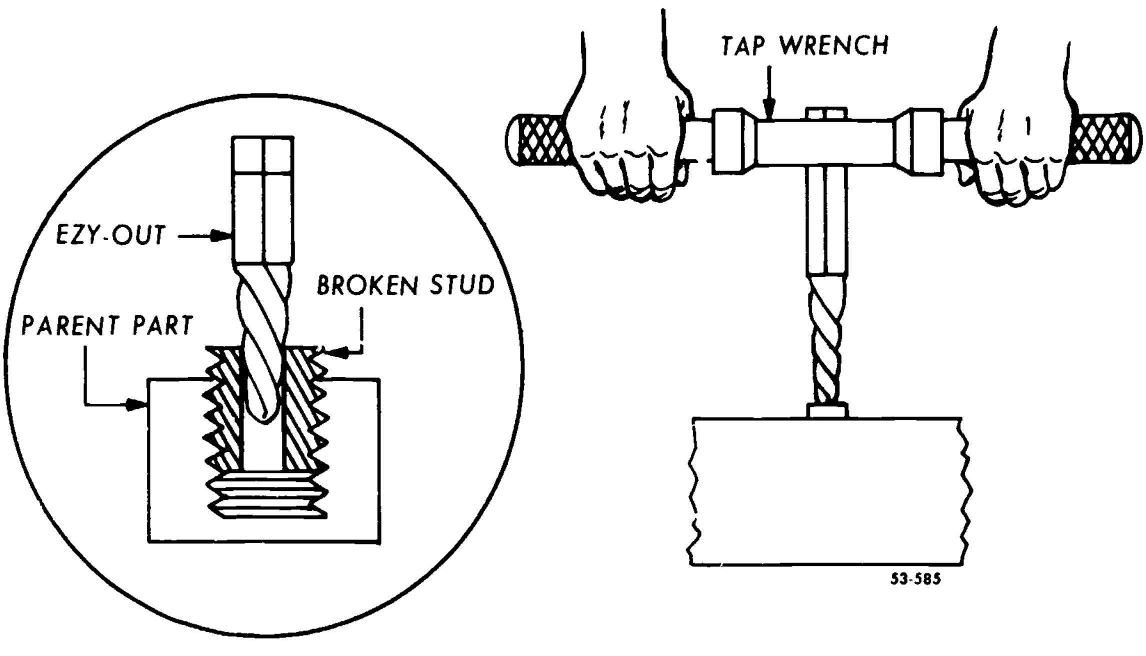


Figure 133. Commercial type Ezy-Out extractor.

welds, metal fabrications, and nonmetallic products. Radiographic inspection is superior to many other methods because it provides a permanent visual representation of the interior of the test objects. This method reveals the nature of a material without alteration, damage, or destruction to the material; discloses errors in manufacturing procedures; and discloses structural unsoundness, assembly errors, and mechanical malfunctions.

16. Removal and Replacement of Studs, Plugs, Screws, and Inserts

16-1. You will frequently have to remove and replace such threaded fasteners as studs, plugs, screws, and inserts. Certain types of aircraft studs require frequent removal and replacement, especially those that are subjected to stress caused by intermittent loading. When you must replace a stud that has been damaged, you should always inspect the threads of the parent metal from which the stud has been removed. A careful examination will often reveal thread weakness. If you can correct this weakness, you can prevent the threads from tearing out when the replacement stud is screwed into place.

16-2. Stud failure may be due to several causes other than stress and intermittent loading. For instance, vibration and exposure to heat on aircraft engines may cause stud failure. Disregard of technical orders and safety factors on the part of the aircraft mechanic, usually involving excessive torque, and the abuse of parts during mounting or dismounting are causes of damage. Regardless of the cause of the failure, it will be your job as a machinist to remove and replace the stud, plug, screw, or insert without damaging the parent part.

16-3. **Removal.** No specific methods or procedures can be recommended for the removal of studs, plugs, or inserts. Regardless of the method you may have to use, your first concern must be to protect the threads of the parent part from damage. If the threads in the parent part are either damaged or destroyed, it may be necessary to manufacture and install an oversize stud or insert. In some cases, the parent part will have to be replaced with a new part. When an aircraft or engine part is involved, this can be very expensive.

16-4. Removing a stud can be very simple or extremely difficult You can sometimes file a square on a stud that protrudes above the parent metal and, with a wrench and some penetrating oil, remove the stud. Occasionally, a stud that has broken off flush with the parent metal can be jarred loose with a punch and hammer and twisted out by hand. If there are remaining threads on the stud, you can lock two nuts on the stud and remove it with a wrench. When a stud breaks off in an area in which there are two adjacent studs, you can use a drill jig as an aid in drilling out the broken stud. The jig is so constructed that it is held by the two adjacent studs, and a guide hole lines up over the broken stud. A drill jig makes the drilling of a stud easy and accurate. In most cases you will not be able to determine what caused the stud to fail and you will not know how tightly the stud is held in the parent metal.

EZY-OUT DRILL SIZES

EXTRACTOR NUMBER	DIAMETER AT SMALL END (INCHES)	DIAMETER AT LARGE END (INCHES)	LENGTH OF FLUTES (INCHES)	LENGTH OVERALL (INCHES)	SIZE DRILL TO USE (INCHES)
1	0.054	0.117	1/2	2	5/64
2	0.080	0.174	3/4	2 3/8	7/64
3	1/8	1/4	1	2 11/16	5/32
4	3/16	21/64	1 1/8	2 7/8	1/4
5	1/4	7/16	1 1/2	3 3/8	17/64
6	3/8	19/32	1 3/4	3 3/4	13/32
7	1/2	3/4	2	4 1/8	17/32
8	3/4	1	2	4 3/8	13/16
9	1	1 9/32	2 1/4	4 5/8	1 1/16
10	1 1/4	1 9/16	2 1/2	5	1 5/16
11	1 1/2	1 7/8	3	5 5/8	1 9/16
12	1 7/8	2 5/16	3 1/2	3 1/4	1 15/16

33-586

16-5. Your ingenuity as a machinist will be a great help in the removal of broken studs. A certain stud may be removed easily on one occasion and the same type of stud on another occasion may not come out at all. Ideally, a stud should be removed in the shop. However most studs, plugs, and inserts must be removed from the aircraft on the flight line. There are probably as many methods of stud removal as there are studs, and you will have to use ingenuity and common sense to select the method that is applicable to the job. We will discuss some of the most common methods.

16-6. *Penetrating oil.* You can sometimes expedite the removal of a stud by applying penetrating oil to the stud and the parent part. The oil helps break up any corrosion between the threads. This will sometimes loosen the stud enough so that you can twist it out by using pliers, vise grips, or a commerical stud driving and removal kit. Penetrating oil has a disadvantage. You must be careful that it does not soak electrical wiring and damage the insulation.

16-7. *Extractors.* You can use extractors, such as the Ezy-Out, shown in figure 133, to advantage to remove studs, plugs, screws or inserts without damaging the parent metal. If the stud to be removed is not corroded too badly and Ezy-Outs are available, it is best to first try to remove the stud with an Ezy-Out. An Ezy-Out has a square tapered body with a left-hand twist, giving it the appearance of a left-hand thread. When the Ezy-Out is inserted in the drilled hole in the stud, it tends to screw in and lock in position.

NOTE: A hole drilled in a stud should be as close to the center of the stud as possible.

You can normally remove a stud with an Ezy-Out by using the proper amount of pressure to turn it out. However the Ezy-Out will not always remove the stud, and it is easily broken by too much twisting pressure. For ordinary conditions the makers of Ezy-Out recommend certain drill sizes, such as the sizes shown in chart 11. Unusual conditions may require the use of larger drill sizes, depending

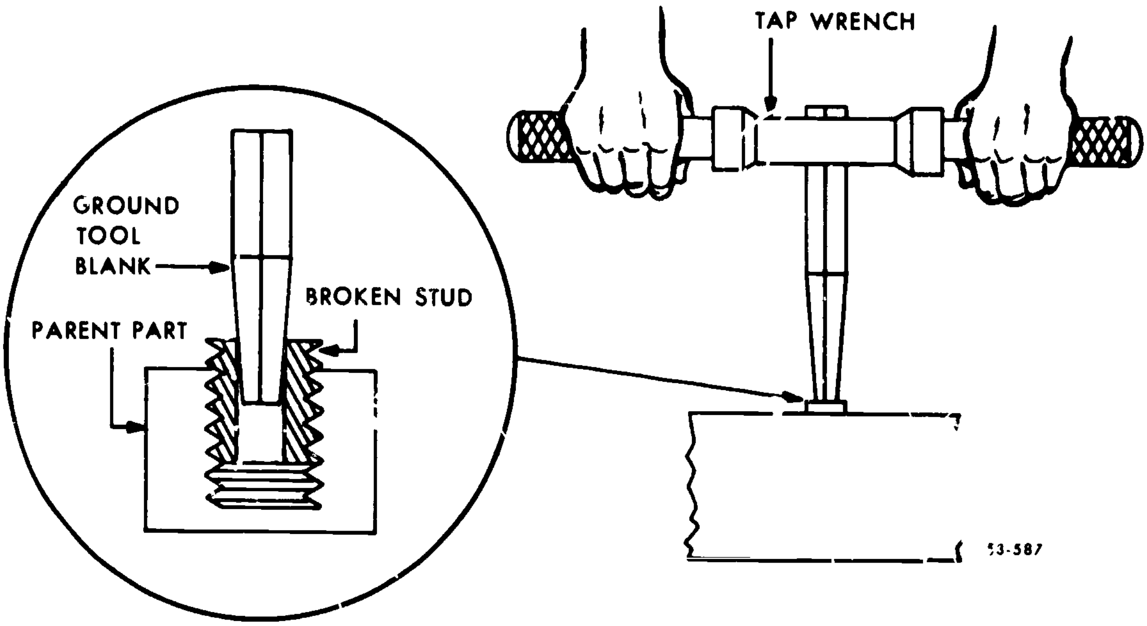


Figure 134. Ground tool type extractor.

upon the length of the broken part or its depth in the parent part. If commercial extractors are not available, you can make a good stud extractor by grinding a high-speed lathe tool with a square tapered end, as shown in figure 134. Drive the square tapered end lightly into the hole drilled in the . The square taper cuts into the wall of the st ving the tool grooves to hold in when you twist it with a wrench. You should turn the Ezy-Out or the ground tool with *even* pressure. The best way to do this is by using either a tap wrench, as shown in figures 133 and 134, or two wrenches placed opposite one another on the extractor shank. Pressure is applied equally to both. This is especially important in the initial loosening of the stud. Usually, once a stud is loosened in this manner, it will screw out of the hole without further difficulty. But if the stud is corroded too badly, the

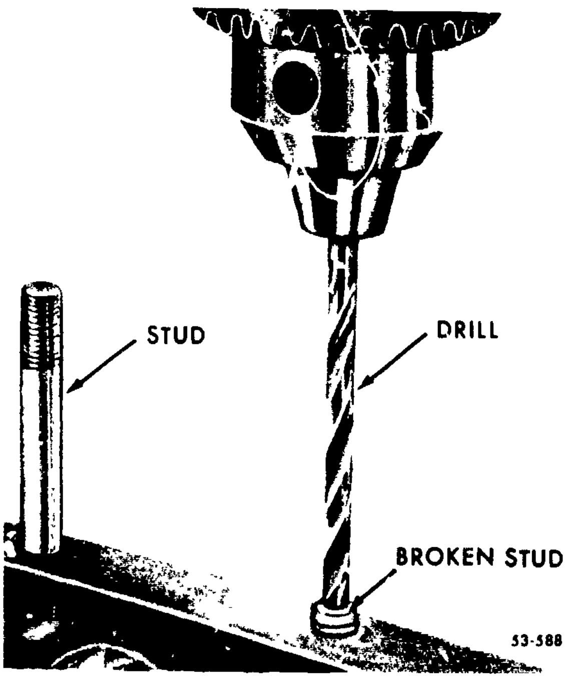


Figure 135. Drilling a broken stud.

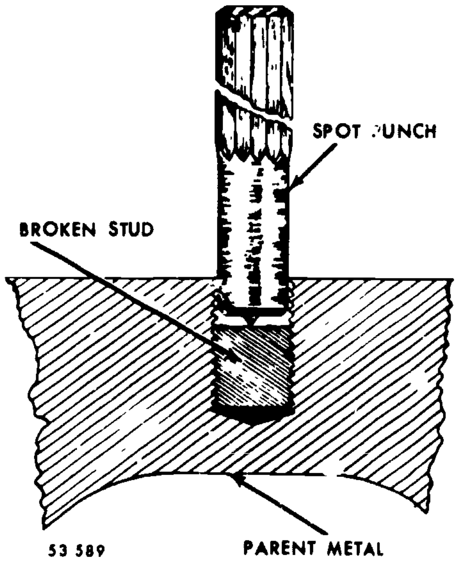


Figure 136. Spot punching a broken stud.

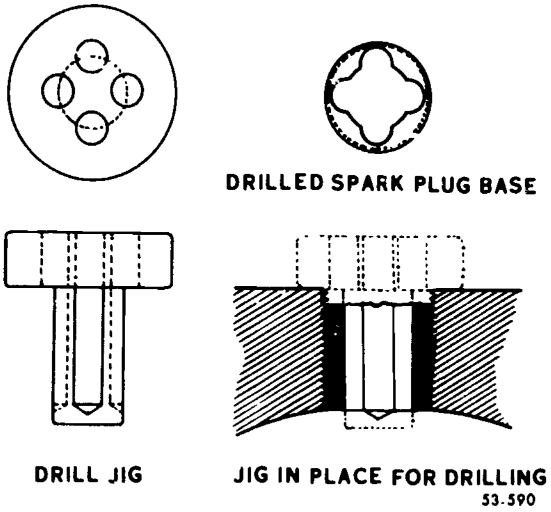


Figure 137. Spark plug drill jig.

extractor will slip out or break when you apply the twisting force.

NOTE: Before using either a commercial Easy-Out or a ground tool to remove a broken stud, you should drill the stud through its *entire length*. Then, if the Ezy-Out or ground tool should break off inside the stud while you are attempting to remove it, the broken off piece can be driven down into the space between the end of the stud and the bottom of the hole. The stud can then be drilled out without damaging the drill or the hardened piece of the extractor.

16-8. *Drilling*. Broken studs may be removed by drilling out the portion left in the parent part, as shown in figure 135. There are several methods of drilling out broken studs. Probably the most accurate method of drilling is to use a drill jig. However, due to close working quarters, you will sometimes have to drill without a jig. Usually, you should try to spot punch the center of the broken

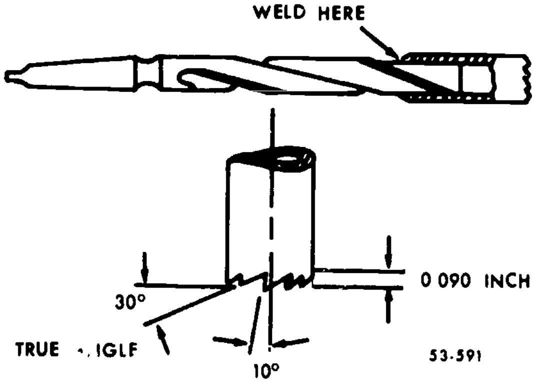


Figure 138. Trepanning tool.

stud, as shown in figure 136, and then drill for an extractor. The accuracy of the drilled hole will depend entirely upon your skill, and if the extractor fails, you will have to drill the stud completely out. The object of drilling is to drill out the exact center of the stud and not touch the threads in the parent part. If you leave a thin enough wall, you can sometimes remove the remainder of the stud with the point of a scribe or a similar sharp-pointed tool.

16-9. *Drill jigs*. You will use drill jigs quite frequently in stud removal. There are so many types, styles, and configurations of drill jigs that we could not possibly explain all of them. Drill jigs are made for specific recurring jobs. A typical drill jig,

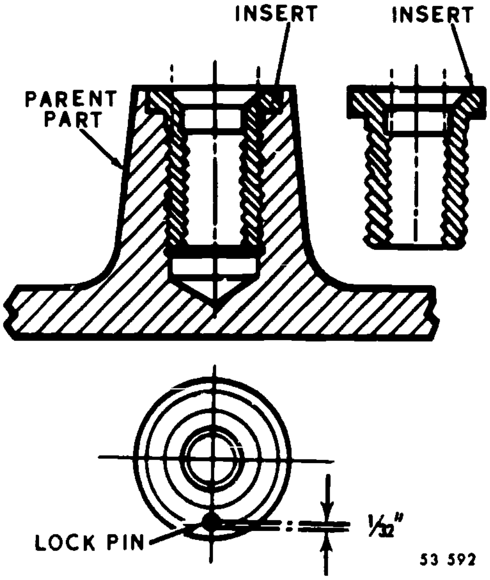


Figure 139. Solid insert.

such as the one shown in figure 137, is used to drill the base of a broken spark plug. The four holes in the face of the jig guide a drill into the wall of the plug base. After the drilling is completed, you can remove the jig and tap the sections between the holes inward toward the center with a chisel to loosen the thread from the mating part. Sometimes, drilling the plug base loosens it enough so that you can unscrew it by using an extractor of the proper size.

16-10. *Welding*. It is possible to remove a worn or damaged stud by welding a nut to the protruding portion of the stud. It is also possible to build up a stud that is broken off below the surface of the parent part and then weld a nut on the stud. A wrench is then used to turn out the stud. The welding should be done by a welder. Great care must be taken to protect the parent part from damage from welding splatter. Welding equipment

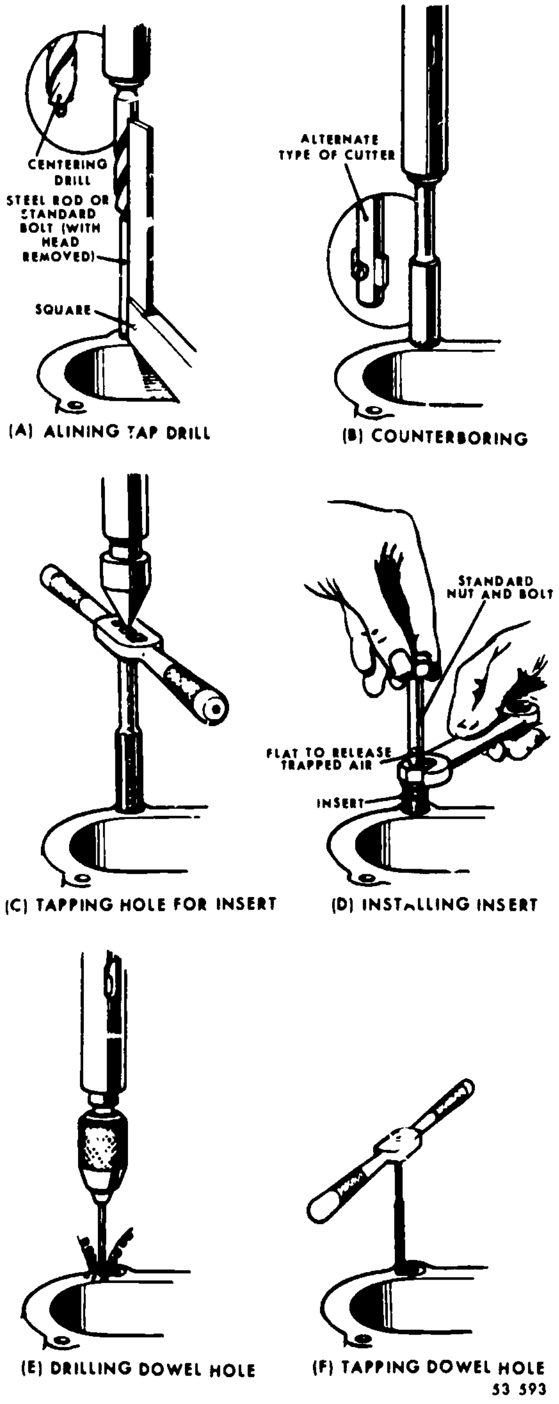


Figure 140. Installation of solid insert.

should not be used around aircraft. The part must be removed and brought to the shop.

16-11. *Trepanning.* You can also remove studs and inserts by the trepanning method. Trepanning is generally used when the parent metal around the stud has become so damaged that it is impossible to save the threads. You should use trepanning only as the last resort, since it involves making a replacement insert or installing a stud with oversize threads. Since commercial trepanning tools are not available, you will have to manufacture the cutter. Trepanning tools (see fig. 138) can be made to fit nearly any size of stud. The tool works like a hollow mill. The disadvantage of trepanning is that the original threads in the parent part are destroyed and oversize threads must then be tapped into the parent part. You should use this method only when there is absolutely no possibility of replacing the damaged stud with a standard sized stud.

16-12. **Replacement.** Successful stud or insert replacement usually depends upon the removal of the faulty part. In most cases, if you encounter no trouble in the removal and the threads are not damaged in the parent part, it will be necessary only to clean out the mating threads with the correct size tap, and make the replacement with standard parts. If you do encounter trouble during the removal and the thread in the parent part is damaged, it will be necessary to tap the parent part with an oversize tap and install an oversize part. There are occasions, however, when even if the part has been removed and there is very little damage to the parent part, you should use an oversize replacement part. This is especially true on aircraft engine work where safety depends upon perfect threads in every part. You will have to use your judgment on equipment other than aircraft as to how well the part must fit. Technical orders will often specify the use of oversize studs.

16-13. *Studs.* Studs are generally made with fine threads on one end and coarse threads on the opposite end, which is the end that fits into the parent metal. Coarse threads are used because they are deeper and stronger. Many times when studs on aircraft engines need replacing, the technical order will require that an oversize stud be used as the replacement. Oversize studs are identified by numbers or symbols on one end of the stud which indicate an increased pitch diameter on the coarse threaded end. Certain oversized studs require oversized tapping of the hole before insertion. Oversize taps are marked as such on the tap shank and usually come in sets. Oversize studs are removed and installed in the same manner as standard size studs.

16-14. *Solid inserts.* You sometimes use a solid insert, shown in figure 139, to return a hole to its original size in the parent part. In some cases a bronze or brass insert is satisfactory. But, if heavy or intermittent loading is evident, it may be desirable to use a stainless steel or nickel steel insert. The pitch diameter of the insert threads should be 0.0015 inch to 0.0025 inch larger than the pitch diameter of the threads in the hole in which the insert is to be installed. The oversize produces a

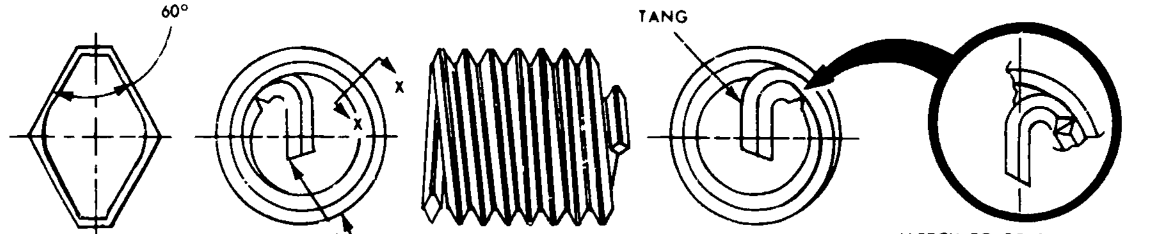
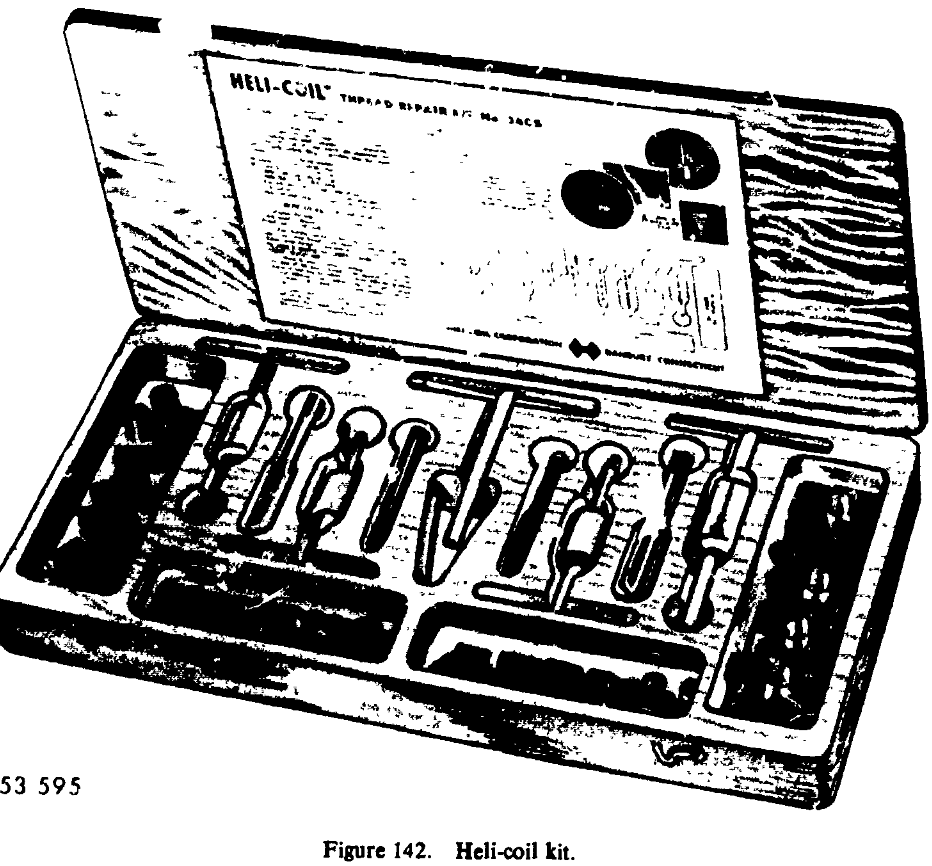


Figure 141. Diagram of a heli-coil insert.

tight fit and keeps the insert from working loose. If there is sufficient area around the stud hole, the insert flange should be wide enough to allow a straight or threaded pin to be used for a lock, as shown in figure 140. If the flange is quite wide, you should use one large pin. If the flange is narrow, you should use two pins. The pins are usually made from stainless steel rod about $\frac{1}{16}$ inch to $\frac{3}{32}$ inch in diameter, or even larger in the case of very large inserts.

16-15. *Heli-coil inserts.* As shown in figure 141, these are spring-shaped thread inserts that are made of 18–8 stainless steel or phosphor bronze wire. A cross section of the wire in one of these inserts appears to have the shape of a diamond. Heli-coil inserts provide new threads that resist wear, corrosion, stripping, galling, and cross threading. On new equipment, Heli-coil inserts help prevent thread failure before it starts. They reduce maintenance costs and their great strength permits the use of smaller and fewer screws and bolts, smaller bases, and thinner flanges. They provide an inexpensive way to restore damaged or stripped threads to their original size. One advantage of Heli-coil inserts is the ease with which they may be removed and replaced. Another advantage



53 595

Figure 142. Heli-coil kit.

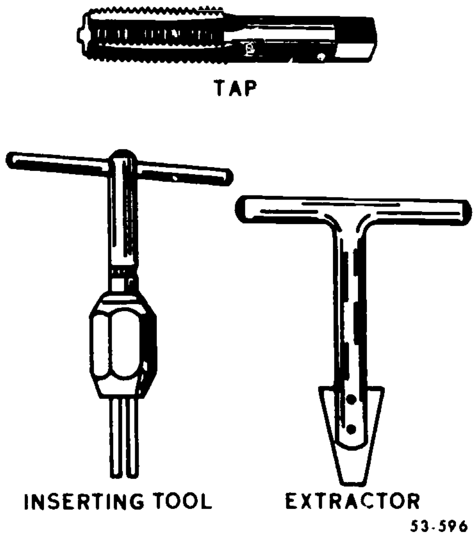


Figure 143. Heli-coil handtool set.

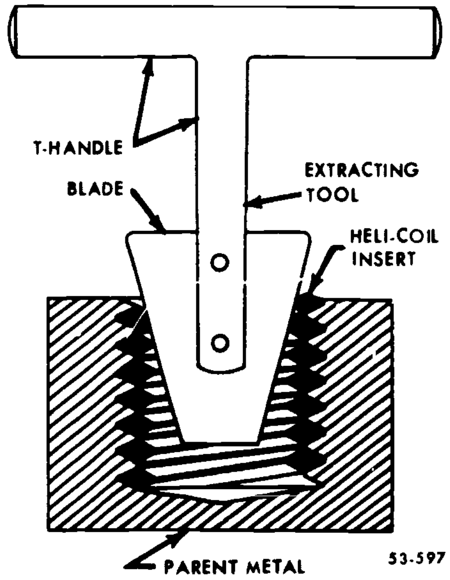


Figure 144. Heli-coil extracting tool.

is that the material they are made of is harder than most screws and bolts. Therefore, if damage does occur, it is the screw or bolt that is damaged, rather than the insert. The smooth finish of the insert makes it desirable for aircraft repairs because there is less danger of seizure of the mating screw threads. The smooth finish reduces thread strain and increases the tightness of the mating threads. Heli-coil inserts are especially useful in soft materials, such as aluminum and magnesium.

NOTE: Stainless steel screws or bolts should never be used in Heli-coil inserts because of the strong possibility that the threads will seize.

Aircraft manufacturers are using the Heli-coil insert more and more in the manufacture of airframes and engines, especially in the later models of jet aircraft.

16-16. Heli-coil kits, similar to the one shown in figure 142, are made up to cover a specific range of standard thread sizes. These kits usually have a chart inside their covers with tap drill sizes and installation data on them. If the kit is not available, you can remove and replace almost any Heli-coil insert with three basic tools, consisting of an extractor, tap, and inserting tool, as shown in figure 143. Every size of Heli-coil insert has its own special tap and inserting tool and you use the extractor for any insert within its range, as shown in figure 144.

NOTE: You can also use a ground lathe tool stud extractor as a Heli-coil extractor.

16-17. There are several important things to remember in installing Heli-coils. You should always use the correct tap drill size for the size of Heli-coil you are installing. Some Heli-coil taps come in two sizes, roughing and finishing, and are marked as such on the tap shank. If you rough tap the hole and install the Heli-coil, the screw or bolt will not screw into it. So be sure that you use a finishing tap before installing the Heli-coil. After installing the Heli-coil, its drive tang must be removed. There is a special tool called a tang breakoff tool that you can use. If a tang breakoff tool is not available, you can use a drive pin punch and hammer. The drive pin punch should be slightly smaller than the minor diameter of the Heli-coil. If the minor diameter of the Heli-coil permits, a pair of long-nosed pliers can be used to break the tank off. You should never use the inserting tool to break off the tang, because the tool could break or the bottom thread of the Heli-coil could be damaged. If the Heli-coil or its broken off tang is accidentally dropped into an inaccessible area, you can use a small magnet to retrieve it. Additional information can be found in Technical Order 44H1-1-117, *General Installation of Heli-Coil Inserts,* and Technical Order 1-1A-8, *Structural Hardware.*

Machine Tool Maintenance

THE AMOUNT of machine tool maintenance that a machinist in an Air Force machine shop is required to do varies with different bases. Every machinist is responsible for a certain amount of routine maintenance involving installation, cleaning, painting, lubrication, minor adjustments, and troubleshooting. The machinist is not responsible for major repairs. When facilities are available, major repairs are made locally. Otherwise, they are made by contract maintenance.

2. Even the most highly skilled machinist cannot do accurate work on a machine that has been improperly installed or needs maintenance. The machinist must do a certain amount of routine maintenance to insure that his work will be safe and accurate and to prevent damage to the machine.

3. In this chapter we will discuss the installation, servicing and adjustment of machine tools. Let's consider first the installation of machine tools.

17. Installation

17-1. Machine shop personnel are often required to move and install machine tools. Installation includes locating, painting, leveling, and securing the machine.

17-2. Let's assume that you are getting a new machine in your shop. You should understand the importance of locating it properly with respect to the available workspace, leveling the machine, and attaching it securely to the floor to prevent vibration.

17-3. **Location.** The first problem concerning the installation of any machine tool is that of locating the machine in the shop. What is the amount of available workspace in the shop? How much workspace is occupied by other machines? What are the dimensions of the machine to be installed? Can the machine be located so that it will not interfere with the operation of other machines? The manufacturer furnishes a plan dimension with each new machine. Some technical orders may also have these dimensions. Allow ample space over and above the dimensions on the plan drawing. Allowance must be made, for instance, for the table movement of a shaper or milling machine. Leave plenty of room for the operator to work around the machine. The floor should be solid and fairly level. In some instances it may be necessary to reinforce the floor to prevent the machine from vibrating because vibration could cause the machine to get out of level.

17-4. Some machines are provided with holes that extend horizontally through the base. Sections of pipe or bar stock of convenient length and size are inserted through these holes to provide a means for lifting the machine. Other machines have lifting lugs; their location on the surface of the base insures that the lift will be exerted at the balance points of the machine. There are many ways to lift or move a machine. The most common way is to use a hoist. When you use a hoist you should use a heavy fiber rope sling on the machine. Using a chain is not practical since the chain will not follow the form of the machine and it may damage some part of the machine. Heavy fiber rope will follow the form very readily and the strain will be equalized throughout its full length.

17-5. **Painting.** Machines occasionally require repainting in accordance with TO 34–1–3, *Repair and Maintenance—Machinery and Shop Equipment.* It is Air Force policy to highlight the working area of the machine and other critical parts by painting the background surfaces with ivory-colored enamel. This color reflects the light and causes the work areas of the machine and tools to stand out against the ivory background. Highlighting increases visibility and reduces eyestrain. Other surfaces that require painting should be painted with a glossy grey enamel. TO 34–1–3 contains illustrations of typical machines and the areas that should be highlighted.

17-6. The operation of some machines may involve extremely hazardous conditions, such as open flywheels, gears, or other moving parts that cannot be guarded. These parts should be painted orange. However, overuse of the orange color will

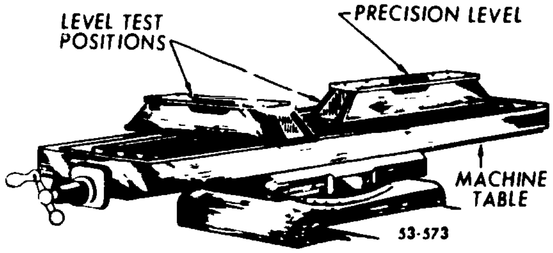


Figure 145. Placement of levels on milling machine.

defeat the intended purpose. Machined parts, such as faceplates, chucks, spindles, etc., should not be painted. For instructions on surface preparation prior to painting, consult TO 34–1–3.

17-7. **Leveling.** After the machine is in place on the floor, it must be leveled. You can use hardwood wedges, shingles, or shimstock to level it. They are placed under the base of the machine, usually in the area of the mounting bolts. Use a precision level to check the levelness of the machine. An ordinary carpenter's level or a combination square level is not sensitive enough for this job. Figure 145 shows the level test positions for leveling a milling machine. When you level a lathe you should position the precision level crosswise and lengthwise on the ways, as shown in figure 146

17-8. **Securing.** Most machines must be fastened to the floor. Some rest on a base or on legs. Figure 147 shows various methods of fastening a machine to a floor. You can use lag screws in concrete or wood. Drill holes in the concrete, and place metal shields or pour molten lead in the holes. Then tighten the lag screws in the holes. If the floor is wood, screw the lag screw directly into the wood. After securing the machine to the floor, check for levelness again, and correct as required.

18. Servicing

18-1. Servicing of machine tools includes lubrication, cleaning, and storage. Servicing also includes the changing of coolants and cutting oils. Occasionally, the hydraulic fluid must be replenished or changed.

18-2. **Lubrication.** All machine tools have definite lubrication requirements for spindle bearings, work heads, ways, lead screws, gearing, and various bearings. All these require lubricants with the proper chemical stability, proper viscosity of oil or consistency of grease, and a high film strength. Because there are so many oil companies refining lubricating oils and greases under so many different trade names, it would be difficult to make any recommendations here on the kind of oil or grease to use on the operating parts of machine tools. For this reason, it is always best to consult the operator's manual, technical order, or machine data plate for the lubrication needs of the machine. This is important, especially on some of the precision grinders, since a regular medium grade of oil will usually cause the spindle to heat excessively and seize. Most grinder spindles require a very fine grade of oil. Usually, the lubricant recommendations in the operator's manual are general, whereas each oil company recommends using specific types of lubricant. It is up to the machinist to convert the general oil specification from the operator's manual to the specification recommended by the manufacturer. Oils and greases are also made by oil companies to meet Government specifications. Always use the grade of oil recommended in technical orders, in operator handbooks, or on the data plate of the machine. During the "breaking in" period of a new machine, be sure that all bearings are oiled and that none of them run hot. If the machine has an oil reservoir, the oil must be changed periodically. Remove the plug to drain the oil and flush the reservoir with an approved

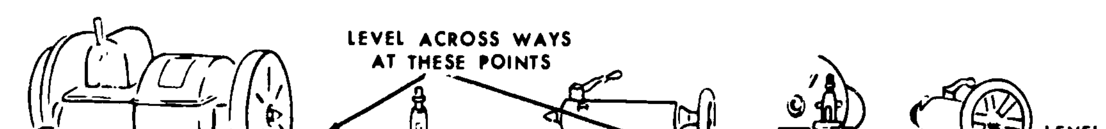


Figure 146. Placement of levels on lathe.

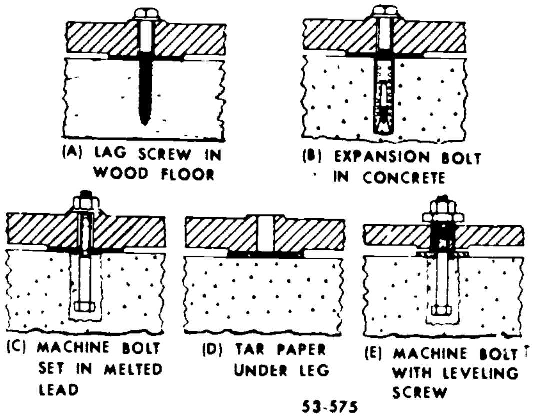


Figure 147. Fastening devices for securing machines.

solvent before you refill it. The machine should be running during the flushing process.

18-3. **Cleaning.** Working parts that are exposed to dust, water base coolant, and chips should be cleaned and oiled frequently. Chips should not be allowed to accumulate on the surface of the table, on the ways, or on any flat bearing surface. Corrosion or ships on a bearing surface can affect the alinement of the cutting tool and cause premature wear of the bearing surfaces.

18-4. **Storage.** The operation of a machine shop involves the use of a great variety of materials, such as shop equipment, handtools, cutting tools, chucks, arbors, measuring devices, attachments, etc. You may use many of these items daily—others only occasionally. Those which are used frequently in the operation of a particular machine should be stored in a location convenient to the machine. Many machine tools, such as contour

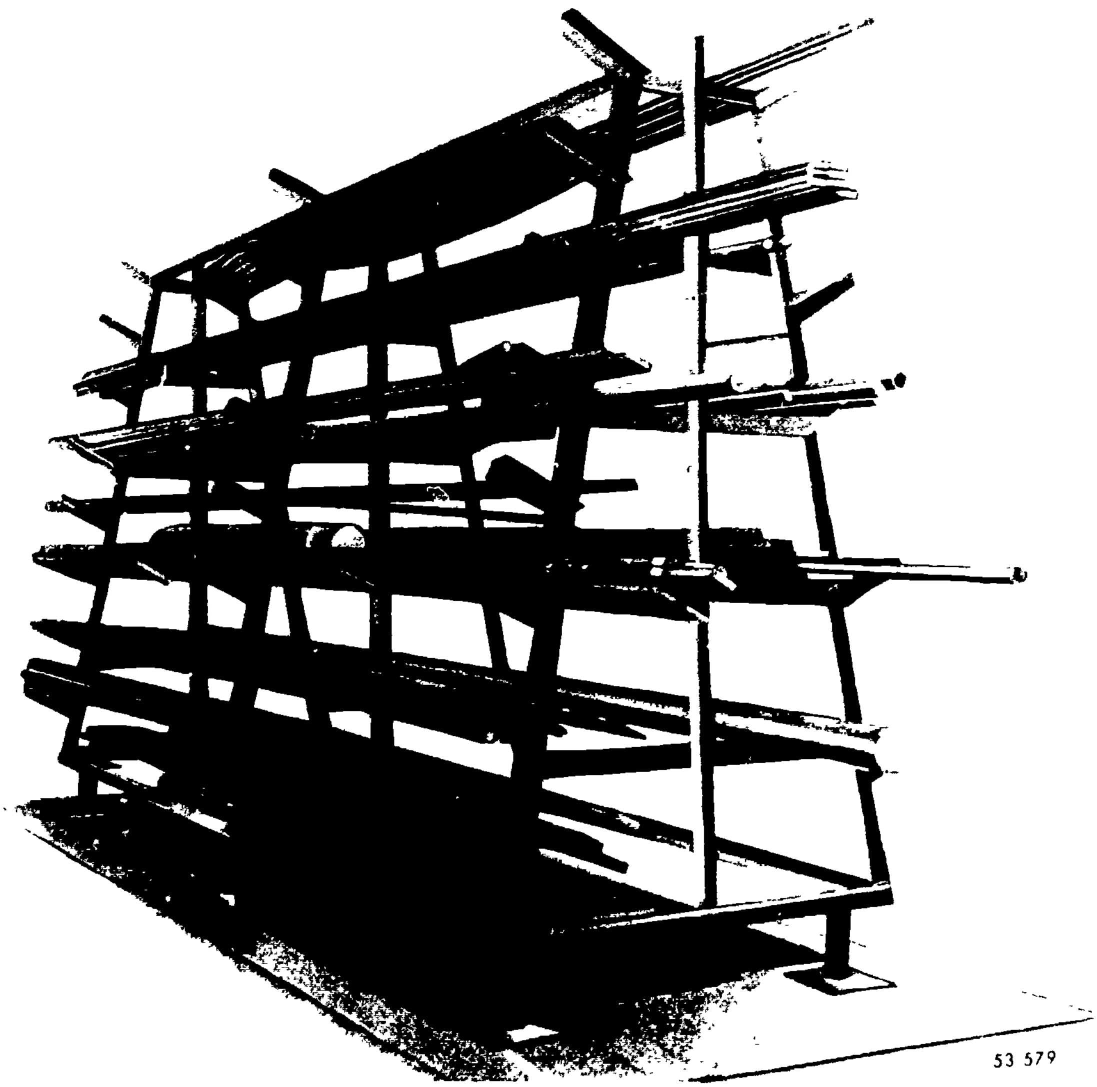


Figure 148. Metal storage rack.

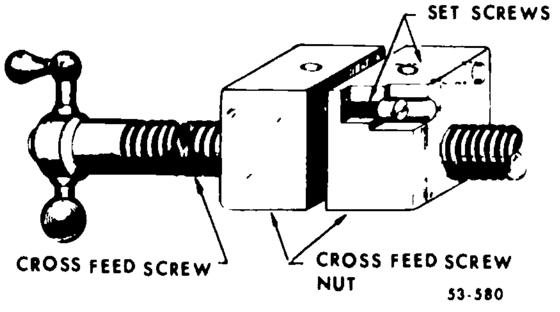


Figure 149. Crossfeed screw wear adjustment.

machines, have cabinets or tables that are specifically designed for the storing of attachments and tools used on that particular machine. The polished and finished surfaces of machines, such as the ways of a lathe, or the table of a milling or contour machine, should have a light coating of oil applied when the machine is not in use. Oil should also be applied to chucks, faceplates, arbors, milling cutters, parallels, etc., when they are not being used. Steel cutting tools or holding devices should get this treatment also, especially at bases where the humidity is high. The reason is that your fingerprints can leave traces of corrosive acids on any metal parts you may touch.

18-5. Materials, other than items used as a part of a machine, are stored in various ways. Paints, oils, and flammable materials should be stored separately from one another and 50 feet from the immediate shop area in a well-ventilated storage building or shed. Other materials frequently used, such as the metal storage rack shown in figure 148, should be located so that they will be easily accessible to the work area.

19. Adjustments

19-1. You are not required to be a specialist in the repair of machine tools. However, there are

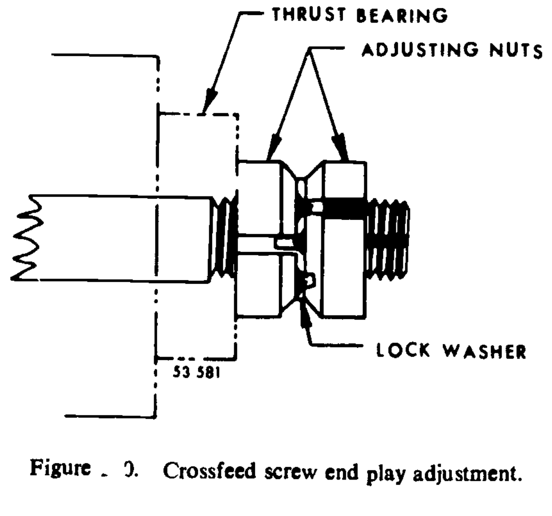


Figure 150. Crossfeed screw end play adjustment.

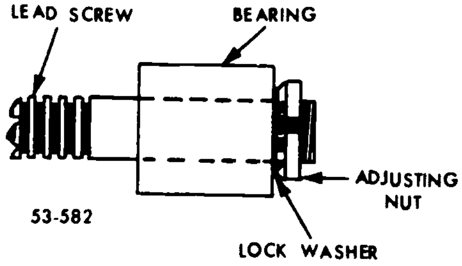


Figure 151. Lead screw end play adjustment.

many minor repairs and adjustments that you can make to correct a variety of troubles. You know when your car is not operating properly and you can often make some minor adjustment that will correct the trouble. When the trouble is too complex or of a major nature, you can take your car to a garage where mechanics who specialize in mechanical and electrical troubles will repair it. In this respect, the maintenance of the machine tools is very similar to the maintenance of your car. You will know when the machine is not operating properly by its sound and by its response to the controls. There are many relatively simple adjustments that you can make to correct various troubles. If you are able to make minor adjustments, you will be able to do more accurate machine operations and to reduce the downtime on the machine. Obviously, if repairs or adjustments are needed which you or your shop foreman cannot make, you will not be able to send the machine to a garage. In this case, the machine is sent to contract maintenance, the manufacturer, or an Air Force depot. However, most machine tool troubles can be corrected by simply adjusting the feed screws, bearings, belts, clutches, gibs, and gearings.

19-2. **Feed Screws.** Feed screws are probably the most important part of any machine tool. Feed screws position the cutting tool in relation to the work and they regulate the rate of feed for a cut. Feed screws that are loose, worn, or incorrectly adjusted decrease the accuracy of the machine. The adjustment of feed screws is practically the same on all machine tools. Adjustments of the feed screws of a lathe are given as typical examples. You can adjust the crossfeed screw of a lathe for wear by tightening the setscrews in the end of the crossfeed split nut, as shown in figure 149. You

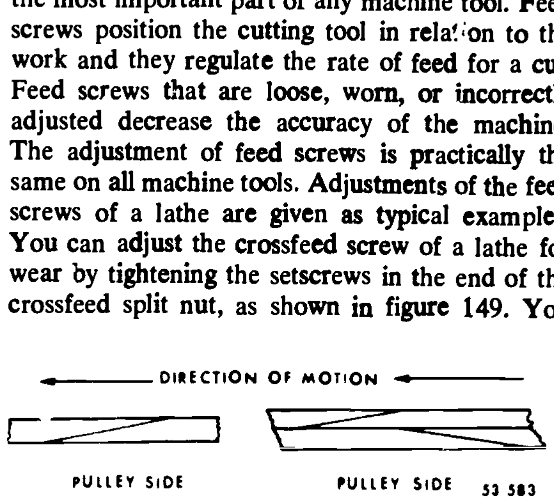


Figure 152. Flat belt lap joint.

can adjust for end play of the lathe crossfeed screw by tightening the nut on the end of the screw, as shown in figure 150. You can adjust the lathe lead screw for end play by removing the cap from the end and tightening the adjusting nut, as shown in figure 151.

19-3. **Bearings.** Spindle bearings on lathes, milling machines, and grinding machines are subject to wear because of high rotational speeds and the pressure of the work against the cutting tool. Spindle bearings on the newer types of lathes and milling machines are the antifriction or taper roller bearing types. They seldom require adjustment. On machines with antifriction bearings, the spindle is mounted on preloaded precision bearings. The end thrust is taken up in both directions by two preloaded ball bearings. These bearings have sealed lubrication and never require adjusting or oiling. When it is necessary to adjust the end play of lathe or milling machine spindle bearings, first place the spindle gearing in neutral. This will permit you to rotate the spindle by hand. You should be able to turn the spindle easily. Depending upon the make of the machine, the thrust bearing may be located behind the rear bearing or the front bearing. In either event, loosen the locking setscrew or nut and tighten the thrust nut just enough to remove the slack. Chuck a rod on the spindle and use it as a lever to move the spindle back and forth along its axis. You can then use a dial indicator to measure the end play. It should not exceed 0.003 inch.

19-4. Grinding machine spindle bearings are even more critical than lathe or milling machine spindle bearings. The grinding machine spindle turns at very high speeds and there is great danger of overheating. Spindle bearings on most grinding machines are either the plain or the antifriction ball bearing types. The surfaces of plain bearings are hardened, ground, and lapped. Some bearings have bronze boxes with spring shoes, which automatically compensate for wear. Others are equipped with setscrews for adjusting the end play. Follow the machine technical order or the operator's manual instructions carefully in making any adjustment because bearings seize easily if they are not correctly adjusted. Plain bearings require a special thin spindle oil because of the close tolerances to which they are built. Do not use an excessive amount of oil because it may overflow and get on the drive belt and cause the belt to slip.

19-5. **Belts.** The power to operate a machine may be transmitted by gears, belts, or a combination of gears and belts. At some point between the motor and the cutting tool, you will usually find a belt. Belting is generally used between the motor and a clutch and gearing is used between the clutch and the spindle. When a belt is used on a

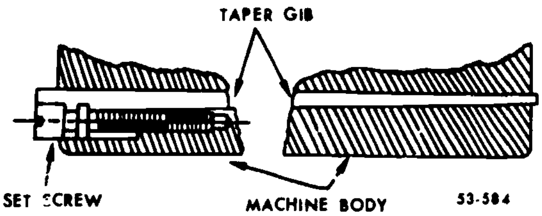


Figure 153. Headless gib.

machine, there is some arrangement for adjusting the tension. Insufficient tension results in a loss of power. Too much tension leads to premature belt wear and excessive side loads on the motor and spindle bearings. Consult the technical order or the operator's manual for the machine for instructions on adjusting belt tension.

19-6. Endless belts are used when it is possible. Sometimes the design of a machine will make it impossible to use an endless belt. Never use a laced or hook joint for belting a grinding wheel spindle, because this will set up vibration and cause inaccurate work. The ends of the belt used on a grinder should be joined with a cemented lap, as shown in figure 152.

19-7. **Clutches.** We will base this discussion of the adjustment of clutches on the milling machine, since all machine tool clutches are similar. The most common clutch is the plate-disc type. There are single-plate and multiplate types. The clutch may operate dry or it may operate in oil. The operation of all types is similar in that the drive depends upon friction. If the plates or discs do not come together tightly enough when the clutch is engaged, slippage will occur and cause excessive wear on the plates. To adjust the clutch, pull out the plunger lock and turn it in the direction to tighten—usually clockwise. The clutch is properly adjusted when you can fully engage the starting lever and when the clutch cone contacts the fingers and causes them to compress the drive plates. If considerable force is needed to fully engage the starting lever, the clutch is adjusted too tightly; the pressure places strain on the clutch fingers and may cause them to break.

19-8. **Gibs.** Gibs are used on machine tools, such as lathes and milling machines, for two reasons: to compensate for wear between a machine's mating surfaces and to hold these mating surfaces in such a way that neither the ability to slide freely nor alinement is impaired. The gibs are installed at the factory for the express purpose of providing a means of adjustment between these mating, sliding surfaces. They are usually constructed from a material softer than that used for the ways of the machine. After a machine has been in use for a time, enough wear will occur to affect the accuracy of machining operations. When this wear becomes excessive, the gibs are adjusted to take up the slack. Gibs may have either tapered or parallel sides, but are usually of the headless taper type with one or more adjusting screws. There may be one screw in the small end of the gib, which is used to secure the gib after adjustment, and one in the large end. In some instances there may be only one adjusting screw in the large end of the gib, as shown in figure 153. To adjust gibs with two screws, such as the table, saddle, or knee gibs of a milling machine, loosen the screw in the small end of the gib and tighten or loosen the gib with the adjusting screw in the large end. Then tighten the screw on the small end again to keep the gib from slipping out of adjustment. You would adjust the gib illustrated in figure 153 by turning the setscrew. This gib is designed so that turning the setscrews tightens or loosens the gib. When you are adjusting either type of gib, the adjustment is correct when the mating surfaces fit snugly together. If the adjustment is too tight, binding will occur and mating surfaces may be scored. Loose gibs cause chatter and vibration as well as rapid wear of the mating surfaces.

19-9. **Gearing.** Spindle transmission drive gearing on machine tools seldom needs adjusting or replacement. Machines such as lathes and some milling machines may be provided with change gears, or replacement gears. These change gears change the ratio of spindle speeds to feeds, as for thread cutting on a lathe. Change gears change the ratio between the index head and table feed screw on a milling machine. Change gears are used on milling machines in cutting helices and for indexing and spacing divisions. The end gearing on most modern lathes transmits the motion from the headstock spindle to the feeding and threading mechanism. This end gearing assembly is usually mounted on an adjustable quadrant. The gears can be swung in and out of mesh. Due to the shock or repeated starting and stopping of the machine, the gearing may slip out of adjustment. Adjustments on change gearing and end gearing should provide a slight clearance between mating teeth. Proper adjustment is achieved when the rotating gears will pass a piece of ordinary bond or tablet paper through the gears without tearing the paper. Proper adjusted gears have a smooth, quiet action. Remember that no adjustment is complete until all guards have been properly replaced.

Bibliography

Books

Handbook of Grinding. Niagara Falls, New York: The Carborundum Comp., 1967.

Operator's Instruction Book. *No. 2 Tool and Cutter Grinder*. Cincinnati, Ohio: The Cincinnati Milling Machine Company, 1943.

OBERG, Erik and Franklin D. Jones. *Machinery's Handbook*, 18th Edition. New York: The Industrial Press, 1968.

LEGRAND, Rupert (Editor). *The New American Machinist Handbook*. New York: McGraw-Hill Book Co., Inc., 1955.

Department of the Air Force Publications

AFM 127–101, *Industrial Safety and Accident Prevention Handbook*, 26 June 1970.

TO 32–1–151, *Hand, Measuring, and Power Tools*, 19 April 1965.

TO 34–1–3, *Repair and Maintenance—Machinery and Shop Equipment*, 3 May 1962.

TO 34–1–10, *Fundamentals of Ordnance Corps Machine Tools*, 2 February 1959.

TO 44H1–1–117, *General Installation of Heli-Coil Inserts*, 1 December 1965.

TO 1–1A–8, *Structural Hardware*, 1 January 1959; Change 27, 10 March 1970.

Resident Course

Training Literature From Block IV, ABR53130, Machinist.

Training Literature From Block V, ABR53130, Machinist.

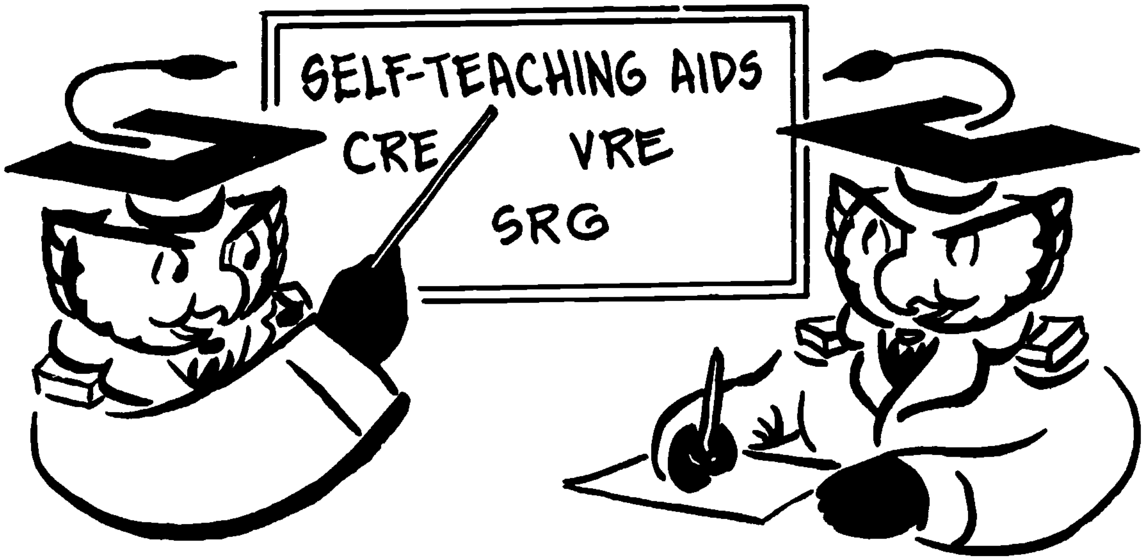
Training Literature From Block VIII ABR53130, Machinist.

NOTE: None of the items listed in the bibliography above are available through ECI. If you cannot borrow them from local sources, such as your base library or local library, you may request one item at a time on a loan basis from the AU Library, Maxwell AFB, Alabama, ATTN: ECI Bibliographic Assistant. However, the AU Library generally lends only *books* and a limited number of *AFMs*. TOs, classified publications, and other types of publications are *not* available. (For complete procedures and restrictions on borrowing materials from the AU Library, see the latest edition of the *ECI Catalog*.)

AU GAFS AL(812787)500

53130 04 21
WORKBOOK

MILLING AND GRINDING MACHINE WORK



This workbook places the materials you need *where* you need them while you are studying. In it, you will find the Study Reference Guide, the Chapter Review Exercises and their answers, and the Volume Review Exercise. You can easily compare textual references with chapter exercise items without flipping pages back and forth in your text. You will not misplace any one of these essential study materials. You will have a single reference pamphlet in the proper sequence for learning.

These devices in your workbook are autoinstructional aids. They take the place of the teacher who would be directing your progress if you were in a classroom. The workbook puts these self-teachers into one booklet. If you will follow the study plan given in "Your Key to Career Development," which is in your course packet, you will be leading yourself by easily learned steps to mastery of your text.

If you have any questions which you cannot answer by referring to "Your Key to Career Development" or your course material, use ECI Form 17, "Student Request for Assistance," identify yourself and your inquiry fully and send it to ECI.

Keep the rest of this workbook in your files. Do not return any other part of it to ECI.

EXTENSION COURSE INSTITUTE

Air University

TABLE OF CONTENTS

Study Reference Guide

Chapter Review Exercises

Answers For Chapter Review Exercises

Volume Review Exercise

ECI Form No. 17

STUDY REFERENCE GUIDE

1. *Use this Guide as a Study Aid.* It emphasizes all important study areas of this volume.

2. *Use the Guide as you complete the Volume Review Exercise and for Review after Feedback on the Results.* After each item number on your VRE is a three digit number in parenthesis. That number corresponds to the Guide Number in this Study Reference Guide which shows you where the answer to that VRE item can be found in the text. When answering the items in your VRE, refer to the areas in the text indicated by these Guide Numbers. The VRE results will be sent to you on a postcard which will list the *actual VRE items you missed.* Go to your VRE booklet and locate the Guide Number for each item missed. List these Guide Numbers. Then go back to your textbook and carefully review the areas covered by these Guide Numbers. Review the entire VRE again before you take the closed-book Course Examination.

3. *Use the Guide for Follow-up after you complete the Course Examination.* The CE results will be sent to you on a postcard, which will indicate "Satisfactory" or "Unsatisfactory" completion. The card will list *Guide Numbers* relating to the questions missed. Locate these numbers in the Guide and draw a line under the Guide Number, topic, and reference. Review these areas to insure your mastery of the course.

Guide Number	*Guide Numbers 400 through 417*
400	Introduction; Construction and Operation; pages 1-8
401	Cutters and Arbors; Cutters; pages 8-13
402	Cutters and Arbors; Arbors; pages 13-18
403	Indexing; pages 18-24
404	Introduction; Plain, Face, and Angular Milling; pages 25-27
405	Milling Flat Surfaces on Cylindrical Work; pages 27-33
406	Slotting, Parting, and Milling Keyseats and Flutes; pages 33-40
407	Drilling, Reaming, and Boring; Milling Machine Attachments; pages 40-43
408	Gearing and Gear Cutting Practices; pages 44-50
409	Introduction; Basic Grinding Information; pages 51-59
410	Surface Grinding Machine Operations; pages 59-63
411	Cylindrical Grinding Machine Operations; pages 63-71
412	Tool and Cutter Grinding Machine Operations; pages 71-74
413	Introduction; Machine Fits; pages 75-78
414	Assembly of Machined Parts; pages 78-84
415	Removal and Replacement of Studs, Plugs, Screws, and Inserts; pages 84-90
416	Introduction; Installation; Servicing; pages 91-94
417	Adjustments; pages 94-96

439

CHAPTER REVIEW EXERCISES

The following exercises are study aids. Write your answers in pencil in the space provided after each exercise. Immediately after completing each set of exercises, check your responses against the answers for that set. Do not submit your answers to ECI for grading.

CHAPTER 1

Objectives: To show an adequate understanding of the construction and operation of milling machines. To be able to identify the various types of cutters and arbors and to explain their uses. To show an adequate knowledge of the various types of indexing and to understand when to use and how to perform the various types of indexing.

1. What is the main difference between a plain milling machine and a universal milling machine? (1-2)

2. What are the two features of a ram type milling machine that make it so versatile? (1-3)

3. What is the most common type of milling machine in AF shops? (1-5)

4. On what type of milling machine is the cutting done horizontally? (1-6)

5. What is the longitudinal table travel of a number 4HD milling machine? (1-7)

6. Where are the coolant reservoir and pump located? (1-10)

7. How is the knee raised and lowered? (1-11)

8. Which feeds can be powered? (1-12)

9. How is the rate of feed temporarily increased in positioning the work to the cutter? (1-12)

440

10. By what two feed methods can the milling machine table be moved? (1-13)

11. How is the arbor secured in the spindle? (1-14)

12. How far should the overarm be extended? (1-15)

13. What are the two types of arbor supports, and how are they different? (1-16)

14. What precaution should you take in loosening or tightening the arbor nut? (1-16)

15. In using attachments that change the ration between the cutter rpm and the spindle rpm, which speed should be determined first? (1-18)

16. What spindle speed would you use to finish mill a piece of aluminum with a 4-inch-diameter cutter? (1-19)

17. When can climb milling be used? (1-21)

18. What is "feed per tooth"? (1-22)

19. What are the two methods of classifying milling cutters? (2-3)

20. How is clearance provided on the cutting edge of a profile cutter? (2-4)

21. Which type of milling cutters are use most often? (2-4)

22. How are formed cutters sharpened? (2-4)

23. What hand is a cutter whose helix angle twists in a counterclockwise direction? (2-6)

24. In what way are the teeth of a 1-inch-wide plain cutter different from the teeth of a 1/2-inch-wide plain cutter of the same diameter? (2-8)

25. Why are cutting edges nicked? (2-8)

26. How is a cutter size designated? (2-9)

27. What is straddle milling? (2-9)

28. How is the width of slots cut with interlocking side milling cutters regulated? (2-11)

29. What are metal slitting saws used for? (2-12)

30. How would you mount a cutter in order to mill dovetail ways? (2-14)

31. What are the types of angle cutters? (2-15)

32 What are end mills used for? (2-17)

33. What operations are required to mill a T-slot? (2-19)

34. What are concave and convex cutters used for? (2-22)

35. How would you describe a gear hob milling cutter? (2-25)

36. What type of cutter would be the best to use for producing a nonstandard shape on a limited number of pieces? (2-26)

37. What are some factors that have to be considered in the selection of a cutter? (2-28)

38. Why should a coarse-tooth cutter be used on material that produces a curled, continuous chip? (2-29)

39. What is the largest diameter of a number 30 taper, and what is its taper per inch? (2-33)

40. You have a milling machine arbor with a code on its flange that reads 62B-20-5. What do these numbers and letters mean? (2-34)

41. How is a screw slotting cutter prevented from flexing during use? (2-38)

42. How is the cutting tool secured in a cutter adapter? (2-41)

43. How can you prevent damaging the threads of the drawbolt or the arbor shank? (2-44)

44. Besides indexing, what other jobs can an index head be used for? (3-3)

45. In what positions can the swivel block be set? (3-7)

46. How are workholding devices mounted on the index head spindle? (3-8)

47. How is the index crank pin positioned in the desired hole circle? (3-9)

48. In alining the swivel block with a dial indicator and test bar, why should you check alinement after reclamping? (3-11)

49. What are the three most common types of indexing? (3-12)

50. What type of indexing would be the fastest for producing a 12-tooth gear, and why? (3-13)

51. How many turns and holes of the index crank and which circle would be required to divide a workpiece into 7 divisions if the only index plate available has hole circles of 27, 28, 30, 31, and 33? (3-14)

52. What are sector arms used for? (3-15)

53. How many turns and holes of the index crank and which circle would be required to divide a workpiece into divisions 14° apart if the only index plate available has hole circles of 29, 31, 32, 36, and 38? (3-16, 17)

CHAPTER 2

Objectives: To be able to explain how plain, face, and angular milling are performed. To show an adequate understanding of milling flat surfaces on cylindrical work. To show an adequate knowledge of slotting, parting, milling keyseats, and milling flutes. To explain how drilling, reaming, and boring are performed. To show an adequate understanding of milling machine attachments and their uses. To show an adequate knowledge of gearing and gear cutting practices.

1. What are the basic milling operations? (4-1)

2. In plain milling, which is set up first, the cutter or the work, and why? (4-2)

3. Why should the work be mounted as close to the column as possible? (4-2)

4. What should you do immediately prior to inserting the arbor into the spindle? (4-3)

5. When the work is returned to its starting position after a cut has been completed, should the cutter be rotating? (4-5)

6. How is backlash removed from the vertical feed screw? (4-6)

7. In what way is face milling different from plain milling? (4-8)

8. Which is mounted first in face milling, the cutter or the work? (4-9)

9. How is angular milling performed when you use side, plain, or face milling cutters? (4-14)

10. Why are square and hexagonal shapes milled on tools? (5-2)

11. Under what conditions are side and end mills used to machine squares and hexagons? (5-3)

12. What is the maximum size square that can be milled on a piece of round material that is 1¼ inches in diameter? (5-6)

13. What is the minimum diameter material you can use to manufacture a hexagonal nut that measures 7/8 inch across the flats? (5-7)

14. What position should the index spindle be in when the feed is horizontal? (5-9)

15. In what sequence are the sides of a square machined on work held in an index head? (5-11)

16. Why is a flat machined on a shaft? (5-14)

17. What is the principle of the slotting attachment? (6-2)

18. What should be done to the work prior to machining a slotted portion that does not extend through the work? (6-2)

19. What are some of the advantages of parting off material with a slitting saw? (6-4)

20. What type of milling is used to saw thick aluminum? Thin steel? (6-5)

21. Which milling cutters can be used to machine straight external keyseats? (6-7)

22. How are the rpm and feed of an end mill computed? (6-9)

23. Why should milling operations performed with an end mill use a lot of coolant? (6-9)

24. How can the accuracy of a keyseat be checked? (6-12)

25. How is the cutter selected for a Woodruff keyseat? (6-15)

26. What is the most accurate method of checking the depth of a Woodruff keyseat? (6-17)

27. Why are flutes machined in cutting tools? (6-18)

28. What are the advantages of flutes milled with form cutters over those milled with angular cutters? (6-23)

29. What feed and depth of cut are used in fly cutting? (6-27)

30. In what way does the actual drilling and reaming of a hole on a milling machine differ from that on a lathe? (7-1)

31. What is the main advantage of eliminating the boring bar holder from the boring adapter? (7-3)

32. What speeds and feeds should be used in boring on a milling machine? (7-5)

33. In what ways can you perform vertical milling on a horizontal milling machine? (8-1, 2)

34. What attachment would be used to hold and space internal gear teeth that are on the inside of an arc? (8-3)

35. How can the swing capacity of an index head be increased? (8-6)

36. How can compound angles be milled on a plain milling machine? (8-7)

37. What is a spur gear? (9-2)

38. What is the difference between pitch diameter and diametral pitch? (9-3)

39. How is a standard gear cutter shaped? (9-5)

40. You have a piece of a broken gear for which you have determined that the number of teeth is 25 and the diametral pitch is 8. What is the outside diameter, pitch diameter, addendum, working depth tooth thickness, and whole depth of tooth? (9-6)

41. After you have nicked the entire gear blank, what is counted, the nicks or the spaces? (9-11)

CHAPTER 3

Objectives: To be able to explain the essential features of grinding wheels and their maintenance and use. To show an adequate understanding of surface grinding machine operations. To show an adequate knowledge of cylindrical grinding machine operations. To show an adequate understanding of tool and cutter grinding machine operations.

1. How many combinations of bond, grade, grain size and abrasive are possible for a grinding wheel? (10-1)

2. What are some of the factors that have to be considered in selecting a grinding wheel? (10-2)

3. What are the most important qualities of an abrasive? (10-3)

4. What are the two main abrasives? (10-3)

5. What abrasive should be used for hard, brittle materials? (10-3)

448

6. What controls the hardness of a grinding wheel? (10-4)

7. How is the grain size for the grinding wheel you want to use determined? (10-5)

8. How are standard wheel shapes and faces identified? (10-7)

9. What do the markings on a grinding wheel represent? (10-8)

10. During your inspecting of a new grinding wheel, you tap it with a wooden hammer handle, and it emits a dull, thudding sound. Why? (10-10)

11. How should grinding wheels be stored? (10-11)

12. Why should you not let coolant continue to flow on a stationary wheel? (10-12)

13. How much of the wheel should the safety guard cover? (10-13)

14. Besides the ratio of their diameter to the wheel diameter, what are the other important requirements of flanges? (10-14)

15. What is the difference in dressing and truing? (10-16)

16. When should the Huntingdon wheel dresser be used? (10-17)

17. How is the abrasive type of wheel dresser driven? (10-19)

18. What type of dresser is the best for truing wheels to be used for precision grinding when high finish and accuracy are required? (10-20)

19. How can diamond dressers "harden" a wheel? (10-20)

20. If a grinding operation requires using coolant, how is the wheel dressed? (10-21)

21. How fast should you feed the dresser across the face of the wheel? (10-22)

22. What speeds have to be considered in grinding work? (10-23)

23. What is the approximate rpm you should use with a 7-inch-diameter wheel and an SFS of 5500? (10-26)

24. How can you vary wheel action when you do not have the correct wheel for a specific grinding job? (10-28)

25. What are the two types of surface grinders? (11-2)

26. What is the maximum crossfeed rate for using a surface grinder? (11-6)

27. How can you hold rectangular work rigidly on edge during surface grinding? (11-8, 9)

28. Why should you wipe a magnetic chuck with your hand before placing the work on it? (11-10)

29. In what direction should the diamond dresser tilt in dressing the wheel? (11-11)

30. Which is more important when coolant is used, pressure or volume? (11-12)

31. What is meant by the term "sparking out"? (11-12)

32. Machining material with a cylindrical grinder is similar to machining material on what other machine? (12-1)

33. Why are both ends of the spindle tapered? (12-4)

34. What is the minimum graduation on the small crossfeed handwheel? (12-5)

35. Is the automatic table travel dependent on whether or not the spindle is turning? (12-8)

36. Why should center holes be lapped before work is cylindrically ground? (12-11)

37. What is cylindrical grinding? (12-14)

38 How is a shear cutting action produced in cylindrical grinding? (12-15)

39. Why should the spindle drive motor not be turned off until after the grinding operation is finished? (12-18)

40. In grinding reamers, how far should the grinding wheel be allowed to run off the shank or blades into the undercut between them? (12-19)

41. What action has to be taken after the table has been swiveled toward the wheelhead? (12-20)

42. What are some examples of conical grinding? (12-23)

43. In setting up to perform co-ical taper grinding, when can the wheel be dressed and trued? (12-25)

44. What are the most common methods of checking tapers? (12-28)

45. In grinding a straight shouler with the side of a wheel, what type of wheel should be used, and why? (12-29)

46. What is more important during plunge grinding, the rate of stock removal or the wheel's ability to hold its form? (12-31)

47. What type of feed should be used in grinding a shoulder? (12-32)

48. What attachment is used to grind a mandrel to size on a tool and cutter grinder? (13-5)

49. You want to modify a plain milling cutter to cut fillets as a slot is milled. What attachment would you use to grind the corners? (13-7)

50. In what directions do the wheel and the work rotate in grinding clearance on a reamer blade? (13-9)

51. Why do have to be careful in sharpening a milling cutter when the wheel rotation is away from the cutting edge? (13-10)

52. What do you know about the clearance if the teeth on a milling cutter drag or chatter occurs when the machine is run at the correct spindle rpm? (13-12)

53. Where is the tooth rest fastened when a helical tooth cutter is to be sharpened? (13-13)

54. What would happen if you sharpened a form cutter with the same method you used on a profile cutter? (13-14)

55. How wide should the wheel be if a convex radius is to be put on its face? (13-15)

CHAPTER 4

Objectives: To show an adequate understanding of the various types of machine fits. To be able to explain how machined parts are assembled. To explain how studs, plugs, screws, and inserts are removed and replaced under various circumstances.

1. What are some of the factors that have to be considered in selecting a fit? (14-1)

2. What is an example of a sliding fit? (14-2*c*)

3. How can a drive fit be assembled? (14-2*d*)

4. What types of fits take advantage of the fact that metal expands and contracts as it is heated and cooled? (14-2*f*, *g*)

5. What are the differences between polishing and buffing? (14-4)

6. What size or grain is used in rough polishing with a dry wheel? (14-5)

7. What should be done to a brand-new polishing wheel before you use it? (14-7)

8. What is the principal purpose of a buffing operation? (14-8)

9. What type of abrasive should you use for sharp, even buffing? (14-9)

10. Why should standard parts be used during fitting and assembly, if at all possible? (15-2)

11. What are the two types of cross-point recesses used in the heads of screws? (15-3. *b*)

12. Why are the heads of bolts and screws marked with symbols, such as dashes, triangles, etc? (15-3, *b*)

13. How many times is a fiber self-locking nut allowed to be used on a bolt installed in an aircraft? (15-3, *c*)

14. How are flathead pints installed? (15-3, *d*(1))

15. Why are ball and roller bearings more efficient than plain bearings? (15-4, *b*)

16. What are the most common types of jigs? (15-6)

17. What factor determines the class of a fixture? (15-7)

18. What type of gage is the most frequently used? (15-8)

454

19. What are go-no-go gages? (15-8, *a, b, c*)

20. What type of gage is used in checking large holes that would require a heavy plug gage? (15-8, *d*)

21. How many sets of ring and plug gages are required to completely check a screw thread, and what does each set check? (15-8, *h*)

22. What is used to set up a sine bar? (15-8, *j*)

23. What are the two types of penetrant inspection methods? (15-10)

24. What type of nondestructive inspection requires that pulleys be removed from shafts, or that bearings be removed from their housings? (15-11)

25. Which nondestructive inspection methods can be used on nonmetallic materials? (15-13, 14)

26. What are some reasons for stud failure? (16-2)

27. What is the advantage of using a drill jig to drill a broken stud? (16-4)

28. How should an Ezy-Out or ground lathe tool extractor be turned? (16-7)

29. If a drill jig cannot be used, what determines the accuracy of a hole drilled in a stud? (16-8)

30. When are oversize studs needed? (16-12)

31. How are solid inserts prevented from working loose? (16-14)

32. What are some advantages of Heli-coil inserts? (16-15)

33. Where is information on correct tap drill sizes for Heli-coil taps found? (16-16, 17)

34. How should the tang be broken off of a newly installed Heli-coil? (16-17)

CHAPTER 5

Objectives: To be able to explain how to properly install various types of machine tools. To show an adequate knowledge of the servicing of various types of machine tools. To show an adequate knowledge of the adjustments which are necessary for the various types of machine tools.

1. How can you determine the required dimensions of a machine you are going to install? (17-3)

2. Why shouldn't a chain be used to lift a machine? (17-4)

3. What technical order covers painting of Air Force machine tools? (17-5)

4. Why should orange paint not be overused? (17-6)

5. How are machines leveled? (17-7)

6. Where can lubrication information on a specific machine be found? (18-2)

7. Why should a thin film of oil be applied to finished surfaces on machines and accessories? (18-4)

8. How should paint and oil be stored? (18-5)

9. Why are feed screw adjustments so important on machine tools? (19-2)

10. How much end play is permissible on spindle bearings? (19-3)

11. Why do grinding machines require a lightweight oil? (19-4)

12. Where are belt drives used on machine tools? (19-5)

13. How can you tell when a milling machine clutch is adjusted too tightly? (19-7)

14. What is a gib and what is it used for? (19-8)

15. What is the purpose of change gearing on machine tools? (19-9)

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 1

1. A universal milling machine has a table which can swivel in a horizontal plane, while a plain table cannot swivel at all.

2. The spindle can be positioned at any angle, and the cutterhead ram can be extended or retracted to fit the requirements of the job.

3. Column and knee.

4. Vertical column and knee.

5. 42 inches.

6. In the base.

7. By using either hand or power feed.

8. Longitudinal, transverse, and vertical.

9. By engaging the rapid traverse lever.

10. By using hand or power feed.

11. With a drawbolt and a jamnut.

12. Just far enough to enable the arbor support to be located over the arbor bearing.

13. The A type, with a small diameter bearing hole, which supports the arbor on its extreme end only, and the B type, with a large diameter bearing hole, which can be positioned anywhere along the arbor length.

14. The arbor support should be installed.

15. The cutter speed should be determined first.

16. 300 rpm.

17. When the work springs or lifts up when conventional milling is used and if the machine is in good mechanical condition.

18. The thickness of the chip.

19. Relief can be profile or form, and the cutter can have a shank or be arbor mounted.

20. By grinding behind the cutting edge.

21. Profile.

22. By grinding the tooth face parallel to the cutter axis.

23. Left-hand.

24. The teeth of the 1-inch cutter are helical.

25. The nicks act as chip breakers.

26. By the diameter and width of the cutter and the diameter of the hole.

27. Cutters used in pairs to mill parallel sides.

28. By inserting thin washers between the cutters.

29. To mill narrow slots or to cut off work.

30. On a stub or shell end mill arbor.

31. Single and double.

32. To mill slots and tangs, and edges and ends of work.

33. A slot has to be milled first to provide clearance. Then a T-slot cutter is used to mill the T part of the slot.

34. Concave cutters are used to mill convex surfaces, and convex cutters are used to mill concave surfaces.

35. It is a formed cutter whose teeth are cut like screw threads.

36. A fly cutter.

37. Kind of cut, material to be cut, number of pieces to be cut, and type of milling machine.

38. To permit an easire flow of chips and coolant and to eliminate chatter.

39. The largest diameter is $1\frac{1}{4}$ inches, and the taper is $3\frac{1}{2}$ inches per foot.

40. A number 60 taper, a 2-inch shaft diameter, a B style arbor, 20 inches usable shaft length, using a $2\frac{3}{4}$-inch-diameter bearing sleeve.

41. By the flanges mounted on both sides of the cutter when it is mounted on the arbor.

42. With a setscrew.

43. By making sure that the drawbolt is extended far enough into the arbor shank so that the drawbolt doesn't pull out of the arbor shank when the jamnut is tightened.

44. In mounting work between centers and in rotating the work during helical milling.

45. From just below center to a point beyond vertical. This varies between manufacturers.

46. Taper-shanked devices can be inserted into the tapered hole in the spindle while chucks can be screwed onto or bolted to a flange on the spindle.

47 Loosen the nut on the center shaft, slide the crank past the shaft to the desired location, and tighten the nut.

48. Because the retightening may cause shifting of the block.

49. Direct, plain, and degree.

50. Direct, because 12 will divide into 24 evenly.

51. $40/7 = 5^5/_7$; $5/7 \times 4/4 = 20/28$. So crank 5 full turns and 20 holes in a 28 hole circle.

52. To keep from having to count the holes for division, thus saving time.

53 $14/9 \times 2/2 = 28/18 = 1^{10}/_{18} = 1^5/_9$, $5/9 \times 4/4 = 20/36$. You need 1 complete turn and 20 holes of a 36 hole circle.

CHAPTER 2

1. Plain and face milling.

2. Set up the work first to prevent accidentally striking the cutter with your hand or arm.

3. To insure a more rigid setup.

4. Wipe off the tapered shank of the arbor and the tapered hole in the spindle.

5. It should not be rotating.

6. By dropping the knee past the original setting for the roughing depth of cut, and then moving the knee back up to the original setting for the roughing cut.

7. Surfaces to be milled that are perpendicular to the cutter axis are face milled, while in plain milling, the surface is parallel to the cutter axis.

8. The cutter.

9. By positioning the work at the required angle.

10. In order to provide a positive drive.

11. End mills can be used for work held in a chuck or between centers, using light cuts. Side mills can also be used for work held in a chuck, using heavier cuts.

12. .884.

13. 1.010

14. Vertical.

15. After the first flat is milled, then do the flat opposite Next, do the flat between the first two, and then the flat opposite the third flat.

16 To provide a seat for a setscrew.

17. The slotting attachment changes the rotary motion of the spindle to a reciprocating motion.

18. Machine a recess to provide clearance for runout of the tool.

19. The work cut off is reasonably square and there is less waste of material.

20. Conventional milling should be used for the aluminum, and climb milling for thin steel.

21. Plain milling and Woodruff cutters, two-lipped end mills, and slitting and slotting saws.

22. With the same formulas as those used for conventional milling.

23. Because of the higher speed and feeds involved.

24. By using rules, outside and depth micrometers, vernier calipers, and go-no-go gages.

25. The cutter has the same thickness and diameter as the key that fits into the keyseat.

26. Insert the correct size key into the keyseat and measure over the work and key with an outside micrometer.

27. Flutes provide cutting edges for the tools and provide a means of letting coolant in and the chips out.

28. A formed flute produces a stronger tooth, lets the chips be removed easier, and prevents cracking during heat treatment.

29. A slow feed and shallow depth of cut.

30. On a lathe, the work rotates around the drill or reamer, while on a milling machine, the drill or reamer rotates into the work.

31. This gives a more rigid operation.

32. The same that you would use for boring a similar sized hole in a lathe or drill press.

33. By using such attachments as the vertical spindle, universal, and high-speed milling attachments.

34. Circular milling attachment.

35. By placing the index head on a set of raising blocks.

36. By attaching a rotatable toolmaker's knee to the table.

37. A gear whose teeth are parallel to the gear blank axis.

38. Pitch diameter is the pitch circle diameter, while diametral pitch is the number of teeth per inch of pitch diameter.

39. To accurately produce the lowest number of teeth in the range.

40. OD = 3.375, D = 3.125; a = .125; Wd = .250; T = .196; W = .269.

41. The spaces.

CHAPTER 3

1. Approximately 12,000.

2. Kind of material to be ground, wheel and work speed, finish and accuracy required, area of contact between the work and wheel, nature of the operation, and type and condition of the wheel.

3. Toughness, hardness, and the nature of the fracture.

4. Aluminum oxide and silicon carbide.

5. Silicon carbide.

6. The amount and strength of the bond

7. By the amount of material to be removed, the desired finish, and the material's physical properties.

8. The shape is identified by a number, while the face is identified by a letter.

9. Kind of abrasive, number of grain size, grade, structure, bond, and modification number.

10. Because it is probably cracked.

462

11. Store straight wheels larger than 6 inches in diameter on edge. Those smaller than 6 inches should go in racks. All thin wheels should be laid on flat surfaces. Flaring cup and dish wheels should have material placed between them and they are stacked flat.

12. Because the wheel will absorb coolant and be out of balance.

13. One-half to three-quarters of the wheel diameter.

14. They should be equal in diameter and relieved in the center.

15. Dressing improves the wheel's cutting action, while truing restores the wheel face concentricity.

16. To dress coarse-grit wheels and wheels used in hand grinding.

17. By contact with the wheel it is dressing.

18. A diamond dresser.

19. If it is dull, it will press the wheel cuttings into the bond pores and load the wheel face, thus making the wheel act harder.

20. The wheel should be dressed with coolant applied in same amount as during the actual grinding operation.

21. As fast as the grain and grade of the wheel and the finish desired on the work permit.

22. Spindle, work, and machine.

23. 3000 rpm.

24. Vary the work speed.

25. The reciprocating table and rotary table types.

26. One-half the wheel face width, per revolution of the wheel.

27 By clamping the work between two parallels.

28. To detect any possible burrs which could impair the seating of the work.

29. In same direction as wheel rotation.

30. Volume.

31. This occurs when the work is left at a setting and permitted to continue moving under the wheel until no more sparks are visible.

32. A lathe

33. To maintain true running wheels and ease of changing wheels.

34. .0001 inch.

35. No, it is driven independently.

36. In order to remove scale and distortion, and to correct inaccurate or rough drilled holes.

37. It is the grinding of cylindrical surfaces to remove heat treating warpage, to reduce the work to exact size, and to improve the finish.

38. By having the wheel and work revolve in opposite directions at the area of contact.

39. Leaving the motor on keeps the spindle bearings at their operating temperature and prevents unbalancing the wheel by absorption of the coolant.

40. Not more than half the wheel width.

41. Back the wheel away from the work and pick up the cut.

42. Shank and point of lathe center and tapered plug gage.

43. Either before or after swiveling the table.

44. With a vernier-equipped micrometer, or a plug or ring gage.

45. A soft grade, recessed wheel, to give a free cutting action.

46. The ability of the wheel to hold its form.

47. A fine hand feed.

48. The cylindrical grinding attachment.

49. A radial attachment.

50. In the same direction at the area of contact.

51. The cutter should not be permitted to move off the tooth rest, as the wheel will remove too much metal.

52. The dragging is caused by not having enough clearance, while the chatter is probably caused because of too much clearance.

53. On the wheelhead.

54. The shape of the formed teeth would change.

55 Twice the desired radius.

CHAPTER 4

1. Temperature, lubrication, materials, bearing load, speed, and length of engagement.

2. A lathe tailstock spindle.

3. By tapping with a hammer or inserting between the jaws of a vise and tightening the screw.

4. The shrink and expanding fits.

5. Polishing is done by using a wheel with abrasives glued to it, while buffing, which is a finer process, uses abrasives that have been applied during the buffing operation.

6. 60 grain

7. It should be "broken" along its entire circumference with a metal rod.

8. To produce a high luster.

9. Emery, in a paste or cake form.

10. Standard parts are readily available, and thus save time, effort, and money.

11. Reed and Prince, and Phillips.

12. To let the user know just what kind of a fastener it is, and to prevent substituting the wrong type of fastener for the right one.

13. Just once.

14. With the head up, and secured with a cotter pin.

15. Because the smaller amount of contact between the balls and rollers with their races produces very little friction during the rolling action.

16. Boring and drilling.

17. The machine on which it is to be used.

18. The fixed gage.

19. These are gages with set limits that are accurately machined to the maximum and minimum size of the part being checked.

20. Pin gages

21. It takes three sets: one for the pitch diameter, one for major diameter, and one for the minor diameter.

22. Gage blocks.

23. The dye and fluorescent methods.

24. Magnetic particle.

25. Ultrasonic and radiographic.

26. Stress, intermittent loading, vibration, exposure to heat, excessive torque, and abuse.

27. This method is more accurate.

28. Preferably by using two wrenches or a tap wrench so as to apply even pressure.

29. The skill of the person doing the drilling.

30. When the threads in the parent part become damaged during removal of the broken stud, or when the technical order specifies it.

31. By having the pitch diameter a few thousandths larger than the mating hole's pitch diameter, thus producing a tight fit, or by using straight pins through the insert flange.

32. They can be removed and replaced easily and cheaply, and because of their hardness, it is usually the bolt or screw that gets damaged.

33. On a chart in a Heli-coil kit and in TO 44H1-1-117.

34. By using a tang breakoff tool, a drive pin punch, or long-nose pliers.

CHAPTER 5

1. By consulting the manufacturer's handbook or the technical order.

2. Because a chain doesn't follow the machine contours and can cause damage to the machine. Also, any strain is concentrated on only a few links, instead of its full length.

3. TO 34-1-3.

4. Because too much orange can defeat the purpose of using orange paint, which is to identify unguarded hazardous conditions.

5. By inserting hardwood wedges, shingles, or shimstock under the machine base while precision levels are laid on the ways or table.

6. In the operator's manual, technical order, or data plate on the machine

7. To prevent corrosion.

466

8. Separately and no closer than 50 feet from a hangar or shop.

9. They position cutting tools in relation to the work and regulate the feed rate.

10. Not more than .003 inch.

11. Because of the close tolerances to which they are built and their very high operating speeds.

12. Us .lly between the motor and the clutch.

13. When it takes excessive force to engage the starting lever.

14. It is a piece of metal fitted between sliding, mating surfaces used to compensate for wear and to hold these mating surfaces in alinement with one another. Gibs are usually made from softer material than the machine they are used on.

15. To change the ratio between the machine spindle and its feeding mechanism.

STO! -

1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER.

2. USE NUMBER 2 PENCIL ONLY.

53130 04 21

EXTENSION COURSE INSTITUTE
VOLUME REVIEW EXERCISE

Carefully read the following:

DO'S:

1. Check the "course," "volume," and "form" numbers from the answer sheet address tab against the "VRE answer sheet identification number" in the righthand column of the shipping list. If numbers do not match, take action to return the answer sheet and the shipping list to ECI immediately with a note of explanation.
2. Note that item numbers on answer sheet are sequential in each column.
3. Use a medium sharp #2 black lead pencil for marking answer sheet.
4. Write the correct answer in the margin at the left of the item. (When you review for the course examination, you can cover your answers with a strip of paper and then check your review answers against your original choices.) After you are sure of your answers, transfer them to the answer sheet. If you *have* to change an answer on the answer sheet, be sure that the erasure is complete. Use a clean eraser. But try to avoid any erasure on the answer sheet if at all possible.
5. Take action to return entire answer sheet to ECI.
6. Keep Volume Review Exercise booklet for review and reference.
7. If *mandatorily* enrolled student, process questions or comments through your unit trainer or OJT supervisor.
   If *voluntarily* enrolled student, send questions or comments to ECI on ECI Form 17.

DONT'S:

1. Don't use answer sheets other than one furnished specifically for each review exercise.
2. Don't mark on the answer sheet except to fill in marking blocks. Double marks or excessive markings which overflow marking blocks will register as errors.
3. Don't fold, spindle, staple, tape, or mutilate the answer sheet.
4. Don't use ink or any marking other than a #2 black lead pencil.

NOTE: TEXT PAGE REFERENCES ARE USED ON THE VOLUME REVIEW EXERCISE. In parenthesis after each item number on the VRE is the *Text Page Number* where the answer to that item can be located. When answering the items on the VRE, refer to the *Text Pages* indicated by these *Numbers.* The VRE results will be sent to you on a postcard which will list the *actual items you missed.* Go to the VRE booklet and locate the *Text Page Numbers* for the items missed. Go to the text and carefully review the areas covered by these references. Review the entire VRE again before you take the closed-book Course Examination.

Multiple Choice

Chapter 1

1. (400) The type milling machine that requires a minimum of work setups is the

   a. column and knee.
   b. ram.
   c. universal.
   d. plain.

2. (400) The main advantage of a vertical column and knee milling machine over a horizontal column and knee milling machine is that the

   a. cutting tool can be mounted easier.
   b. work can be observed more easily.
   c. work can be mounted quickly
   d. end and face mill cuts can be better observed.

3. (400) Milling machine size is determined by the amount of

   a. horsepower required for operation.
   b. cross feed travel.
   c. longitudinal table travel.
   d. vertical feed travel.

4. (400) The primary difference in the function of a lathe and a milling machine spindle is

   a. that the milling spindle drives cutters rather than the work.
   b. in their position in relation to the work.
   c. in the method in which they are driven.
   d. that the milling spindle is used to feed the work.

5. (400) The chip thickness each cutter tooth removes is used to determine the

   a. spindle RPM.
   b. feed.
   c. speed.
   d. depth of cut.

6. (400) The term "climb milling" describes a cut during which the feed moves in a direction that is

   a. vertical to the cutter axis.
   b. opposite to the cutter rotation.
   c. horizontal to the cutter axis.
   d. the same as cutter rotation.

7. (401) Milling cutters are classified according to their

   a. mounting method and type of cutting edge relief.
   b. diameter and width
   c. name and use.
   d. shape and size.

8. (401) The class of milling cutter that is used the most is the

   a. formed cutter.
   b. helical teeth cutter.
   c. profile cutter.
   d. staggered tooth cutter.

469

9. (401) The type of cutter used to mill flat surfaces parallel to the cutter axis is the

a. face milling cutter.
b. side milling cutter.
c. parallel milling cutter.
d. plain milling cutter.

10. (401) The characteristic that plain and side milling cutters share is their

a. helical teeth.
b. nicked cutting edges.
c. size designations.
d. inserted teeth.

11. (401) If a plain milling cutter has a face width of 3/16 inch or less, it is then known as a

a. side milling cutter.
b. slitting saw.
c. slot cutter.
d. cutoff saw.

12. (401) The form tooth cutter used to mill reamer flutes is a variation of

a. a double angle cutter.
b. an oblique angle cutter.
c. a single angle cutter.
d. a combination angle cutter.

13. (401) The best type of end mill to use in milling a slot without first drilling a starting hole is the

a. center cutout end mill.
b. shell end mill.
c. taper end mill.
d. two-lipped end mill.

14 (401) Which of the following sizes of Woodruff keyseat cutters does *not* have a shank?

a. 1 inch in diameter.
b. 1¼ inches in diameter.
c. $1^7/_{16}$ inches in diameter.
d. $1^9/_{16}$ inches in diameter.

15. (401) The cheapest method of cutting a limited number of special forms on a milling machine is to use a

a. form cutter.
b. fly cutter.
c. gear hob.
d. special cutter.

16. (402) Which designation would be correct for an arbor that has a pilot bearing, a 16-inch usable shaft length, a number 50 taper, and shaft diameter of 1 3/8 inches?

a. 501 3/8A-16.
b. 51.375B-16.
c. 51 3/8A-16.
d. 50B-16-1 3/8.

17 (402) A milling cutter is prevented from slipping on a standard arbor by

a. threading the cutter on the end of an arbor shaft.
b. tightening the arbor nut against the spacing collar.
c. a friction fit between the cutter and the arbor.
d. keying the cutter to the arbor shaft.

18. (402) The primary difference between a taper adapter and a cutter adapter is the

a. method of holding the cutter.
b. internal taper.
c. external taper.
d. method of mounting the adapter.

19. (402) The primary purpose for cleaning the spindle hole and arbor shank prior to mounting an arbor is to

a. prevent damage to the drawbolt.
b. insure that the arbor and spindle are alined.
c. insure alinement of the arbor support
d. prevent damage to the overarm.

20. (403) The process of spacing divisions along the circumference of work is called

a. dividing.
b. spacing.
c. splitting.
d. indexing.

21. (403) The index head is alined with the longitudinal axis of the milling machine table by

a. screws on the swivel block.
b. T-slot bolts.
c. keys on the base plate.
d. angle plate guide pins.

22. (403) The purpose of notches on the edge of some index plates is to

a. provide a means of hole-and-pin alinement.
b. hold the index plate on the dividing head.
c. keep the index plate from slipping.
d. serve as reference marks for the next division.

23. (403) The simplest and quickest method of indexing 15° divisions is the

a. direct method.
b. plain method.
c. degree method.
d. wide range method.

24. (403) How many turns and holes of the index crank would be required to divide a workpiece into 9 divisions if a 36-hole circle is used and the index head has the standard ratio?

a. 4 turns, 0 holes.
b. 4 turns, 4 holes.
c. 4 turns, 10 holes.
d. 4 turns, 16 holes.

25. (403) One turn of the crank for a standard ratio index head equals how many degrees?

a. 4°.
b. 9°.
c. 10°.
d. 12°.

Chapter 2

26. (404) In plain milling, the width of the cutter in relation to the width of the surface to be milled should be

a. one-half as wide.
b. the same.
c. slightly wider.
d. twice as great.

27. (404) Prior to setting a new depth of cut if the trial cut is too deep, you should

a. snug down the knee lock bolts.
b. remove the vertical feed backlash.
c. lower the speed and feed.
d. check the work and vise alinement.

28. (404) Setting up to perform face milling differs from setting up for plain milling in that

a. the cutter is mounted prior to the work.
b. the cutter width is greater than the work width.
c. the work is mounted pric to the cutter.
d. climb milling is used.

29. (404) The direction the work should approach the cutter during *face* milling is from the

a. left to the right.
b. right to the left.
c. side which places the cutter thrust down.
d. side which places the cutter thrust up.

30. (404) An important step to remember after setting the depth of cut in face milling is to

a. set the graduated dial at ZERO.
b. adjust the coolant flow.
c. hand-feed the work into the cutter.
d. lock the saddle.

31. (405) The best way to prevent arbor interference in machining squares or hexagons on work ends is to

a. use a large enough cutter.
b. use the smallest diameter arbor spacers
c. feed the work horizontally.
d. feed the work vertically.

32. (405) In order to mill squares and hexagons by using *conventional* milling, the cutter thrust direction on mounted work should be

a. up on vertical and horizontal.
b. down on vertical and up on horizontal.
c. down on vertical and horizontal.
d. up on vertical and down on horizontal.

33. (405) In milling work held in a screw-on type chuck on the index head, be sure that the

a. depths of cut aren't too deep.
b. tail stock helps support the work.
c. cutter thrust tends to tighten the chuck.
d. index spindle is alined.

34. (405) What is the depth of cut for each side which is required to obtain the largest square from a piece of 2-inch round material? (Distance across flats of a square = diameter of st' times .707.)

a. 0.293 inch.
b. 0.586 inch.
c. 0.707 inch.
d. 1.414 inches.

35. (405) The diagonal of a hexagon is equal to the

a. length of one side.
b. distance across the flats
c. diameter of a circle passing through its six corners.
d. diameter of a circle tangent to its six sides.

36. (405) Milling a square or hexagon on work held by a vertically positioned index head means that the work will feed

a. vertically as the cutter performs climb milling.
b. horizontally as the cutter performs up milling
c. vertically as the cutter performs up milling.
d. horizontally as the cutter performs climb milling.

37. (405) If possible, the end mill to use in milling flats on work held between centers should have

a. two cutting edges per 1/2 inch of work diameter.
b. two cutting edges in contact with work at all times.
c. a diameter equal to the length of the flat.
d. a diameter slightly greater than the flat length.

38. (405) In cutting a square or hexagon, accuracy should be checked for the first time after the

a. first roughing cut.
b. first finishing cut.
c. second roughing cut.
d. fourth finishing cut.

39. (405) You have to mill two flats in one plane on opposite ends of a workpiece, and have already scribed the horizontal lines on each end. After inserting the work into the chuck, these lines are alined with the previously set surface gage point by

a. inserting shimstock under the index head.
'. rotating the work inside the chuck.
c. raising or lowering the knee.
d. rotating the index ad spindle.

40. (406) The primary reason that the work and slotting attachment should be positioned vertically is to allow the

a. coolant to drain off the work better.
b. cutting action to be observed more easily.
c. chips o keep clear of the work.
d. work o be removed and replaced easily.

41. (406) The slitting saw you would use to saw thin material should na

a. coarse teeth.
b. stagge.ed teeth.
c. fine teeth.
d. inserted teeth.

42. (406) In milling an external keyseat, the distance the work is moved to cline it under the cutter is equal to the

a. work diameter plus one-half cutter width.
b. radius of the work plus cutter width.
c. work diameter plus cutter width.
d. radius of the work plus one-half cutter width.

473

43. (406) The width of a keyseat should be checked

a. after the first roughing cut.
b. by inserting the key that is to be used.
c. after the finish cut.
d. by nicking the end of the work with the cutter.

44. (406) A Woodruff key can best be described as

a. an oval piece of metal.
b. a half-disc of metal.
c. an oblong piece of metal.
d. a square piece of metal.

45. (406) Cutting a Woodruff keyseat differs primarily from cutting other types of keyseats in the

a. speeds and feeds used.
b. manner of holding the work.
c. use of hand-feed for entire depth of cut.
d. rotation of cutter.

46. (406) The width of a convex cutter used to cut flutes in a tap is equal to

a. one-sixth of the tap diameter.
b. one-quarter of the tap diameter.
c. one-third of the tap diameter.
d. one-half of the tap diameter.

47. (406) The depth of a flute in a tap is equal to

a. one-sixth of the tap diameter.
b. one-quarter of the tap diameter.
c. one-third of the tap diameter.
d. one-half of the tap diameter.

48. (406) In milling tap flutes, you should use

a. up milling, with the thrust toward the index head
b. down milling, with the thrust toward the index head.
c. up milling, with the thrust toward the tailstock.
d. down milling, with the thrust toward the tailstock.

49. (406) Straight reamer flutes are unequally spaced in order to

a. provide more teeth.
b. help reduce chatter.
c. simplify indexing.
d. strengthen the teeth.

50 (406) The approximate depth of a reamer flute is determined by the

a. length of the flute.
b. number of flutes.
c. diameter of the reamer.
d. rake of the teeth.

51. (406) The cutting tool that can be used to cut flutes in taps and reamers, as well as keyways in shafts, is

a. an arbor cutter.
b. a fluting cutter.
c. a fly cutter.
d. a keyway cutter.

52. (407) The main advantage of drilling and reaming on a milling machine is that the drills and reamers can be

a. used horizontally and vertically.
b. accurately located in relation to the work.
c. held securely with a more positive drive.
d. fed by hand or power feed.

53. (407) You want to use a 1/4-inch diameter end mill in the vertical position on a horizontal spindle milling machine. The *best* attachment to use would be a

a. vertical spindle attachment.
b. universal spindle attachment.
c. high speed universal spindle attachment.
d. circular milling attachment.

54. (408) In calculating the dimensions used to manufacture a spur gear, the most important one is the

a. gearing ratio.
b. circular pitch.
c. outside diameter.
d. pitch circle.

55. (408) In order to select the proper cutter for a spur gear, you have to know the gear's

a. pitch diameter and number of teeth.
b. number of teeth and diametral pitch.
c. outside diameter and diametral pitch.
d. pitch circle and circular pitch.

56. (408) The reference point for taking gear teeth measurements is located at the area

a. midway between highest and lowest runout.
b. of minimum runout.
c. of maximum runout.
d originally closest to the cutter.

Chapter 3

57. (409) The abrasive that should be used to sharpen high-speed steel tools for use in a lathe is

a. aluminum carbide.
b. silicon carbide
c. aluminum oxide.
d. silicon oxide.

58. (409) If the amount of bond is increased, the grinding wheel will

a. become harder.
b. not be affected.
c. become softer.
d. absorb less coolant.

59. (409) When the contact area between a grinding wheel and the work is small and you don't want the wheel to wear too rapidly, you should use a

a. fine-grain, widely spaced, medium hard wheel.
b. medium-grain, closely spaced, hard wheel.
c. fine-grain, closely spaced, medium hard wheel.
d. medium-grain, medium spaced, medium hard wheel.

60. (409) The grade of a grinding wheel is indicated by

a. letters, with A the softest and Z the hardest.
b. numbers, with 10 the softest and 600 the hardest
c. letters, with A the hardest and Z the softest.
d. a word—soft, medium, or hard.

61. (409) In flange mounting a grinding wheel, the flange diameter should be approximately

a. one-quarter the wheel diameter.
b. one-third the wheel diameter.
c. half the wheel diameter.
d. three-quarters the wheel diameter.

62. (409) The screws used to hold the flange of a collet-type wheel holder should be tightened

a. consecutively around the rim, one by one.
b. consecutively around the rim, but every other one.
c. as tightly as possible.
d. indirectly in opposite pairs.

63. (409) A grinding wheel is dressed for all of the following reasons *except* to

a. improve the wheel's cutting action.
b. clean out the clogged pores.
c. balance the wheel.
d. alter the wheel's cutting characteristics.

64. (409) To prevent chatter and gouging while using a diamond dresser, its point should be positioned

a. 30° to the wheel's plane and slanted 3° to 15° in the direction of rotation.
b. 30° to the wheel's axis and slanted 3° to 15° in the direction of rotation.
c. 45° to the wheel's plane and slanted 13° to 15° to the wheel's axis.
d. 60° to the wheel's axis and not slanted at all.

65. (409) In order to increase the surface foot speed on a machine with a fixed spindle rpm, you will have to

a. select a wider faced wheel.
b. decrease the wheel diameter.
c. select a narrower faced wheel.
d. increase the wheel diameter.

66. (410) Surfaces of heat-treated workpieces that are to be finished are rough ground in order to

a. relieve internal stresses.
b. remove heat treatment scale.
c. maintain parallelism.
d. eliminate shimming the work.

67. (410) Depths of cut for *finish* grinding should *not* exceed

a. .0002 inch.
b. .0005 inch.
c. .002 inch.
d. .005 inch.

68. (411) The maximum swivel range of the universal tool and cutter grinder swivel table is

a. 45°.
b. 60°.
c. 90°.
d. 135°.

69. (411) A cylindrical grinder footstock differs from a lathe footstock primarily because it has

a. provisions to maintain constant spring pressure on the center.
b. a different way of clamping to the table.
c. no method of lateral adjustment of the center.
d. a different angle on the point of the center.

70. (411) In order to provide a shear type of cutting action in cylindrical grinding, the direction of the wheel and the work travel should be

a. opposite each other at the area of contact.
b. at the same revolutions per minute.
c. in the same direction at the area of contact.
d. at different revolutions per minute.

71. (411) Clearance can be ground onto the teeth of a reamer by cylindrically grinding it in such a manner that the

a. cutting edge strikes the wheel first.
b. wheel never travels completely off the tool.
c. first quarter of the tooth length is tapered
d. tooth heel strikes the wheel first.

72. (411) Grinding a true conical taper requires the grinding wheel axis to be

a. above the work axis.
b. below the work axis.
c. at the same height as the work axis.
d. at an angle to the work axis.

73. (411) One difference between picking up the cut on a conical taper and a straight cylindrical surface is that the

a. work speed is slower.
b. table traverse is in motion.
c. wheel speed is slower.
d. footstock end of the work is picked up first.

74. (411) A groove or undercut can be ground in hardened steel by

a. face grinding.
b. cylindrical grinding.
c. angular wheel grinding.
d. plunge grinding.

75. (411) Universal tool and cutter grinders differ from plain tool and cutter grinders primarily because they have

a. more accessories.
b. greater table swivel capability.
c. power wheelhead elevation.
d. a greater range of spindle speeds.

76. (411) You have a job requiring the grinding of a compound angle and the only machine available is a tool and cutter grinder. The angles could be ground by using a

a. radial grinding attachment.
b. surface grinding attachment.
c. tool post grinder.
d. set of angled parallels.

77. (412) The biggest disadvantage of sharpening a milling cutter by using the wheel rotation to hold the cutter teeth against the tool rest is the

a. raising of a burr on the cutting edge.
b. possible tooth damage if the cutter should turn.
c. drawing out of the cutting edge temper.
d. greater breakdown of the grinding wheel.

78. (412) The main reason that formed cutters are sharpened with a radial rake is to

a. reduce chatter.
b. simplify the setup of the grinder.
c. avoid overheating the cutting edge.
d. retain the shape of the teeth.

Chapter 4

79. (413) The types of fits with the strongest holding power are the

a. expanding and shrink fits.
b. drive and shrink fits.
c. drive and force fits.
d. expanding and force fits.

80. (413) Producing a high luster on a work piece with soft abrasives is known as

a. smooth buffing.
b. cut-down buffing.
c. color buffing.
d. corrosion buffing.

81. (414) The best type of pin to use on parts that have to maintain accurate alinement after being repeatedly taken apart for cleaning is the

a. taper pin.
b. straight pin.
c. flathead pin.
d. spring pin.

82. (414) The best type of pin to use on parts subject to vibration, and which is self-securing, is the

a. taper pin.
b. straight pin.
c. flathead pin.
d. spring pin.

83. (414) The type of antifriction bearing most commonly used to resist very high end thrust is the

a. needle bearings.
b. tapered roller bearings.
c. ball bearings.
d. plain bearings.

478

84. (414) The main difference between a jig and a fixture is that the

a. jig guides the cutting tool.
b. fixture guides the cutting tool.
c. fixture is used primarily on the drill press.
d. jigs hold work while fixtures do not.

85. (414) Receiving gages differ from ring gages in that they are used primarily to check

a. circular parts.
b. noncircular parts.
c. diameters of shafts.
d. depths of holes.

86. (414) In order to determine if an assembly error has been made, the method of inspection that should be used is the

a. fluorescent penetrant method.
b. eddy current method.
c. radiography method.
d. ultrasonic method.

87. (415) The most important fact to remember in removing any damaged threaded fastener is

a. not to damage the parent part.
b. to remove it in one piece.
c. to find out why it got damaged.
d. to eplace it with a stronger fastener.

88. (415) The least preferred method of stud removal is

a. welding a nut on the stud.
b. trepanning.
c. drilling.
d. using a punch and hammer.

89. (415) When a stud that is threaded into a Heli-coil insert is subjected to high stresses, any damage that may occur will be to the threads of the

a. stud that mates with the Heli-coil.
b. Heli-coil that mates with the stud.
c. Heli-coil that mates with the parent part.
d parent part that mates with the Heli-coil.

Chapter 5

90. (416) A new milling machine vibrates at high speeds. The *most probable* cause is

a. a bent spindle.
b. a floor that hasn't been reinforced.
c. an excessive amount of backlash.
d. a floor that is too hard.

91. (416) The best way to hoist a machine that has no lifting lugs is by using a sling made of

a wire rope.
b. chain.
c. fiber rope.
d. leather.

92. (416) In leveling a machine, the best type of level to use is a

a. combination square level.
b. masonry level.
c. carpenter's level.
d. precision level.

93. (416) The machine tool part that would most probably seize if the lubricant were too heavy would be the spindle bearings on

a. a toolroom lathe.
b. a precision grinder.
c. an all-angle milling machine.
d. a radial drill press.

94. (416) A repair action that would *not* be considered as operator maintenance is the

a. resurfacing of damaged lathe ways.
b. replacement of a drive belt.
c. adjustment of gibs.
d. removal of feed screw backlash.

95. (417) The machine tool whose bearings are the most critical is the

a. lathe.
b. milling machine.
c. drill press.
d. grinder.

96. (417) The best type of belt to use in turning a grinding machine spindle is a

a. laced joint type.
b. leather belt.
c. cemented lap type.
d. hooked joint type.

97. (417) A newly sharpened milling cutter develops chatter. After determining that the speeds, feeds, and work setup are correct, the probable cause is that the

a. drive belt tension is too high.
b. clutch needs adjustment.
c. antifriction bearings need lubrication.
d. gibs are worn or loose.

98. (417) The gears on lathe and milling machines that often require adjusting are

a. end and change gears.
b. drive-spindle and feed gears.
c. spindle transmission gears.
d. drive-spindle and index head gears.

180

STUDENT REQUEST FOR ASSISTANCE

PRIVACY ACT STATEMENT

AUTHORITY: 44 USC 3101. PRINCIPAL PURPOSE(S): To provide student assistance as requested by individual students. ROUTINE USES: This form is shipped with every ECI course package. It is utilized by the student, as needed, to place an inquiry with ECI. DISCLOSURE: Voluntary. The information requested on this form is needed for expeditious handling of the student's need. Failure to provide all information would result in slower action or inability to provide assistance.

SECTION I: CORRECTED OR LATEST ENROLLMENT DATA: MAIL TO ECI, GUNTER AFS, ALA 36118

1. THIS REQUEST CONCERNS COURSE
2. TODAY'S DATE
3. ENROLLMENT DATE
4. PREVIOUS SERIAL NUMBER
5. SOCIAL SECURITY NUMBER
6. GRADE/RANK
7. INITIALS
LAST NAME
8. OTHER ECI COURSES NOW ENROLLED IN
9. ADDRESS: (OJT ENROLLEES - ADDRESS OF UNIT TRAINING OFFICE/ALL OTHERS - CURRENT MAILING ADDRESS)
ZIP CODE
10. NAME OF BASE OR INSTALLATION IF NOT SHOWN ABOVE:
11. AUTOVON NUMBER
12. TEST CONTROL OFFICE ZIP CODE/SHRED

SECTION II: Old or INCORRECT ENROLLMENT DATA

1. NAME:
2. GRADE/RANK:
3. SSAN:
4. ADDRESS:
5. TEST OFFICE ZIP/SHRED:

SECTION III: REQUEST FOR MATERIALS, RECORDS, OR SERVICE

(Place an "X" through number in box to left of service requested)

ADDITIONAL FORMS 17 available from trainers, OJT and Education Offices, and ECI. The latest course workbooks have a Form 17 printed on the last page.

1. EXTEND COURSE COMPLETION DATE. (Justify in Remarks)
2. SEND VRE ANSWER SHEETS FOR VOL(s): 1 2 3 4 5 6 7 8 9 - ORIGINALS WERE: NOT RECEIVED, LOST, MISUSED
3. SEND COURSE MATERIALS (Specify in remarks) - ORIGINALS WERE: NOT RECEIVED, LOST, DAMAGED.
4. COURSE EXAM NOT YET RECEIVED. FINAL VRE SUBMITTED FOR GRADING ON (Date):
5. RESULTS FOR VRE VOL(s): 1 2 3 4 5 6 7 8 9 NOT YET RECEIVED. ANSWER SHEET(s) SUBMITTED ON (Date):
6. RESULTS FOR CE NOT YET RECEIVED. ANSWER SHEET SUBMITTED TO ECI ON (Date):
7. PREVIOUS INQUIRY (ECI FORM 17, LTR, MSG) SENT TO ECI ON:
8. GIVE INSTRUCTIONAL ASSISTANCE AS REQUESTED ON REVERSE:
9. OTHER (Explain fully in remarks)

REMARKS: (Continue on Reverse)

OJT STUDENTS must have their OJT Administrator certify this request.

ALL OTHER STUDENTS may certify their own requests.

I certify that the information on this form is accurate and that this request cannot be answered at this station. (Signature)

ECI FORM 17 JUN 77 PREVIOUS EDITIONS MAY BE USED

SECTION IV: REQUEST FOR INSTRUCTOR ASSISTANCE

NOTE: Questions or comments relating to the accuracy or currency of textual material should be forwarded directly to preparing agency. Name of agency can be found at the bottom of the inside cover of each text. All other inquiries concerning the course should be forwarded to ECI.

VRE ITEM QUESTIONED:

Course No. ___________

Volume No. ___________

VRE Form No. _________

VRE Item No. _________

Answer You Chose (Letter) __________

Has VRE Answer Sheet been submitted for grading?

☐ YES ☐ NO

REFERENCE
(Textual support for the answer I chose can be found as shown below)

In Volume No: ________

On Page No: __________

In______ (Left)_____(Right) Column

Lines_____Through______

MY QUESTION IS:

Remarks:

482

53130 00 S01 7709

CHANGE SUPPLEMENT

CDC 53130

APPRENTICE MACHINIST

(AFSC 42730)

IMPORTANT: Make the corrections indicated in this supplement before beginning study of Volumes 1, 2, 3, and 4. This supplement contains both "pen-and-ink" changes and replacement pages. It is perforated and three-hole-punched so that you tear out the replacement pages and insert them in your volumes.

Prelicimus Alert Intelent

Extension Course Institute

Air University

CHANGES FOR THE TEXT: VOLUME 1

Pen-and-Ink Changes:

Page-Col	*Paragraph*	*Line(s)*	*Correction*
iii (preface)		16	Change "(TSOC)" to " TTGOX "
iii (preface)		21	Change to "Gunter AFS AL 36118 "
5	Chart 2		Change "AF Form 1098" to "AF Form 2095 " (Three entires). Also delete "Specialty Knowledge Test" (three entries)
11L	5-2	3	Change "*Safeguarding Classified Information*" to "*Information Security Program*, which supplements DOD ISPR 5200 1-R "
13R	6-12	12	Change "tetnus" to "tetanus "
15L		11	Change "gound" to "ground "
15L	6-19	17	Delete both commas (,).
15L	6-20,*a*	2	Change "toolbix" to "toolbox."
16L	6-26,*g*	1	Change "spead" to "speed."
24R		3, 4	Change "*The Machinery's Handbook*" to "TO 1-1A-9" and delete other."
29R	10-12	7	Change "536X0" to "427X2 "
30I	11-3	7	Change symbol before "plus" from ÷" to "+ "
43I	Fig 29		The figure is printed upside down
47R		1	Change "1," to "2 "
57L		12 fr bot	Change "figure 71" to "figure 67."
73R	18-13	3	Change "88" to "85 "
73R	18-13,*c*	3	Change "89" to "86."
83R	19-11	2	Change "clerance" to "clearance "
86L	20-5	3	Change "parrallel" to "parallel."
89L	20-18	2	Change "30°" to "60° "
89R	20-19	3	Change "60°" to "90°."
89L	Fig. 120		Change "60°" to "90°" and "30°" to "60° "
91	**Bibliography, Dept. of AF Publications**	2	Change "AFM 127-101, *Accident Prevention Handbook*" to "AFR 127-101, *Ground Accident Prevention Handbook.*"
92	**Resident Course**		Delete "**Resident Course**" and listed materials.

Page Changes: Remove old pages and insert new pages as indicated. If the replacement pages number more than those they replace, those pages in excess of the pages being replaced will carry letter designations For example. 20, 20a, 20b, etc.

Remove Pages	*Insert Pages*
1-4	1-4

General Machine Shop Information

ONE OF the great needs of the Air Force today is for well-trained technicians. Opportunities are almost unlimited for workmen who are skillful with their hands and who are trained to think about their work, to diagnose troubles, and to suggest improvements. And, of course, no one can hope for success in any line of work unless he is willing to study and improve his skills.

2. Most of the skilled machinists and machine shop technicians in the Air Force are the products of Air Force apprentice training programs. This is your opportunity to lay the groundwork to become a highly qualified craftsman and to get started on a fine career The rewards for the skilled machinist can be very gratifying, both in advancement and in pride of workmanship.

3. This volume is designed to give you the background information you need to know to get started in machine shop work. It covers general machine shop information, metals and heat treatment, preparatory shop work, and introductory machine and bench work.

1. The Aircraft Systems Maintenance Career Field

1-1 Everyone in the Air Force, regardless of his rank or grade, has a job to do. You are probably wondering what you will be doing throughout the remainder of your enlistment. It is only natural that you are concerned about what the future holds for you in your career field. The Air Force has a great need for skilled airmen in hundreds of different jobs Your job assignment depends upon Air Force needs and on your ability to learn and do certain kinds of work

1-2 **Air Force Career Field System.** The Air Force has a system for grouping related jobs into common work areas or career fields. Each job grouping or career field requires the same general qualifications and the same sort of ability to learn and perform related jobs. The *Airman Classification Structure Chart* 39-1 is Air Force visual aid chart (AFVA) which shows all career fields and their number designations. This chart also shows the skill levels and the equivalent grade spread for each job specialty. You can usually find this chart posted in the shop area where you work The Aircraft Systems Maintenance Career Field is divided into three subdivisions: Aircraft Accessory Systems; Aircraft Propulsion; and Fabrication. As a result of your background and the job classification interviews and the tests which you took in basic training, you were assigned to the fabrication career field subdivision.

1-3. The fabrication career field subdivision includes fabricating, shaping, cutting, and joining metals and repairing metal parts; aircraft structural repair; and metal heat treating, welding, plating, and machining. The installation, modification, and formation of plastic articles are also a part of this career field. It also includes corrosion control for missile, aircraft, and support systems and the maintenance and repair of fabric and rubber equipment such as parachutes, life preservers, radiation barriers, and protective clothing The fabrication career field subdivision is divided into six ladders: Machinist, Corrosion Control, Nondestructive Inspection, Fabrication and Parachute, Metals Processing, and Airframe Repair. Each career ladder is divided into job specialties; for example, the machine shop ladder is divided into technician and machinist specialties. The apprentice machinist is a part of the machinist specialty.

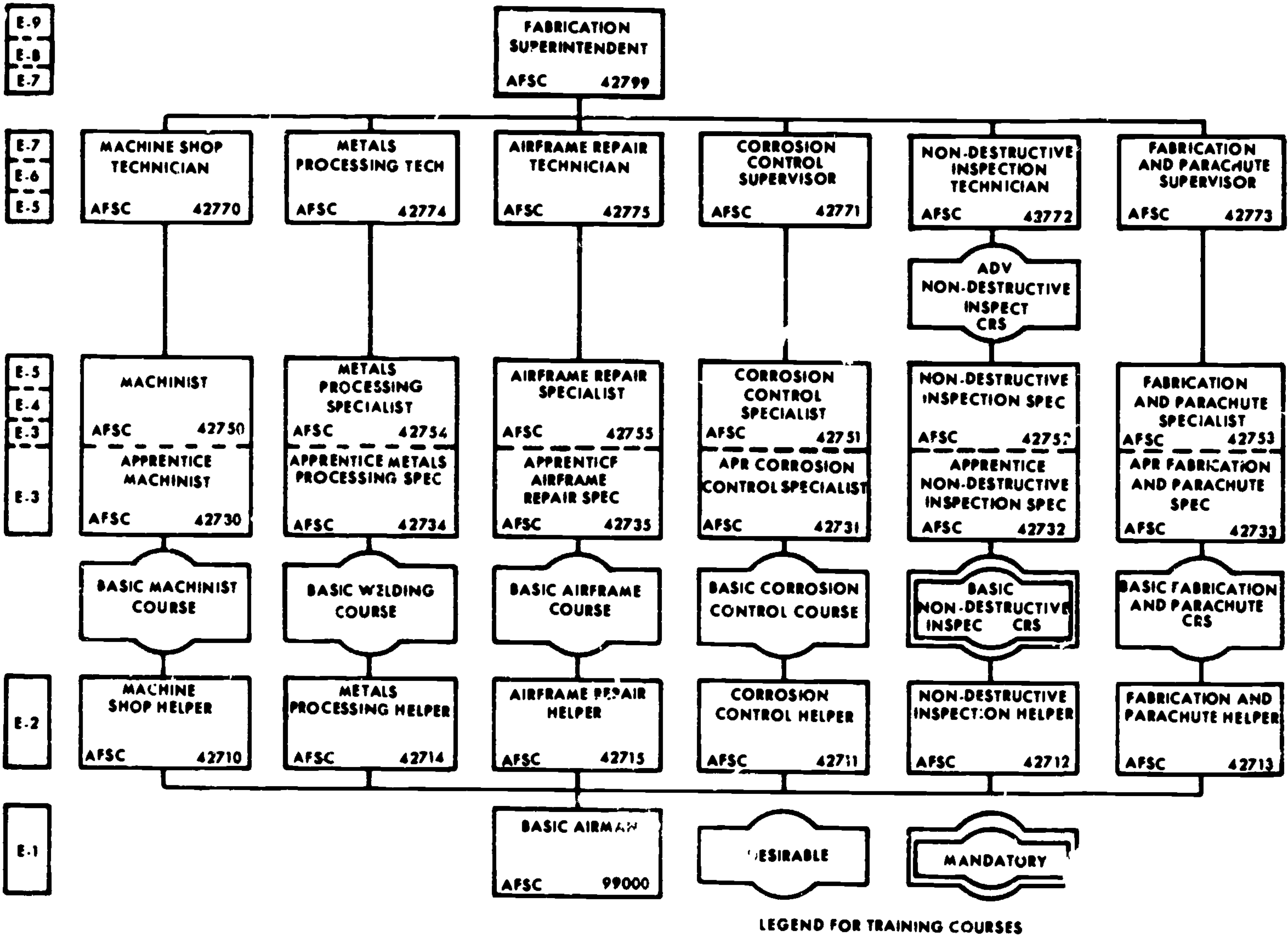
1-4. **Air Force Specialty.** An Air Force specialty (AFS) is identified by title and code. A five-digit code number is used to make up an Air Force Specialty Code (AFSC) which identifies an AFS The first two digits identify the career field. The third digit identifies the career field subdivision. The fourth digit shows the skill level. The fifth digit identifies the career ladder within the career field subdivision. All together, the five digits identify the specific AFS. Since you are in training to become an apprentice machinist, let's show the breakdown of the apprentice machinist AFSC 42730. The breakdown of this AFSC is as follows:

08

42	Career Field	Aircraft Systems Maintenance
7	Subdivision	Fabrication
3	Skill Level	Semiskilled
0	Specific AFS	Machinist
42730	Complete AFSC of the Apprentice Machinist	

CHART 1

FABRICATION CAREER FIELD SUBDIVISION



1-5. Study the fabrication career subdivision chart shown in chart 1. Start at the bottom of the chart. You will note that, as a basic airman, you had the AFSC 99000. This was your AFSC during basic training because you had not yet been assigned to a career field. Upon completion of basic training, you were assigned to the fabrication career field subdivision as a machine shop helper and awarded the AFSC 42710. This AFSC meant that you were qualified to enter into training in the machinist ladder in either of two ways. You could have received your training by taking the basic machinist course conducted by the Army at Aberdeen Proving Grounds, Maryland. However, you were given a direct duty assignment of AFSC 42710 and you entered on-the-job training to become an apprentice machinist.

1-6. When you have satisfactorily completed your training you will be awarded the primary AFSC 42730, apprentice machinist. Your primary AFSC is the specialty in which you are most highly qualified. Your duty AFSC is the AFSC to which you are assigned. During training your duty AFSC is normally one skill level higher than your primary AFSC. For example, your primary AFSC is 42710; but since you are in training, your duty AFSC is 42730. By studying the chart, you will note that there are five skill levels in each ladder of the fabrication career field subdivision. You will also note in the far left column of the chart, the grade spread for each skill level. The top level, or fabrication superintendent AFSC 42799, can be an input from any of the ladders. You progress up the ladder from one skill level to the next, usually, through on-the-job training. We will discuss this in the following section.

1-7. **AFM 39-1, Airman Classification Manual.** Air Force Manual 39-1 contains the authorized Air Force Specialties and Air Force Specialty Codes. These AFSs and AFSCs are used in the classification of airmen positions and personnel. AFSs provide job standards for procurement, training, education, utilization, and development of airmen. AFSCs give us a systematic means for identifying training and position requirements.

1-8. Air Force Specialty descriptions are composed of the following parts:

a. Heading. The heading consists of the specialty code, specialty title, and effective date.

b. Summary. The summary is a concise statement of the content of the AFSC.

c. Duties and responsibilities. This part describes the scope of the job specialty in terms of duties and responsibilities.

d. Qualifications. This part gives the job qualification standards for adequate performance in the AFS. Standards are either mandatory or desirable. They are stated in five parts: knowledge, education, experience, training, and other—such as physical requirements, security clearance, certification, etc.

e. Specialty data. This part establishes the grade spread for the AFS. The grade spread is used for authorizing manning positions. This also designates jobs closely related to the AFS—for use in your initial selection or in your return to civilian life.

f. Specialty shredouts. This part designates authorized shredouts to be used with the AFS and the letter suffix identifier for use with the AFSC. It also gives the portion of the specialty to which it is related.

1-9. **Specialty Descriptions (AFSC 42730/50/70/99).** Air Force specialty descriptions in AFM 39-1 describe the duties and responsibilities of the job specialty. You should be especially interested in the duties of your AFSC and in those of other specialties in your career ladder. We will not attempt to cover all the duties and responsibilities of each AFSC in your career ladder. You can read the complete descriptions in AFM 39-1. Therefore we will discuss only the major duties and responsibilities of each AFSC in your ladder.

1-10. *Machinist (AFSC 42730/50).* The specialty summary for a machinist states that he "operates metalworking machines in fabrication, rework, and repair of metal parts" You will note that this specialty description serves both AFSC 42730 and AFSC 42750, since both the 3- and 5-skill-level machinist perform essentially the same duties. The 5-skill-level machinist has more knowledge of his work and is more skillful than is the 3-level machinist. Also, the 5-level machinist has some duties in advanced machine shop work and supervision and training that the 3-level machinist does not have. The major duties and responsibilities of the machinist are as follows:

a. Manufactures and reworks machined parts.

b. Assembles and fits and machines parts.

c. Maintains hand and machine tools.

d. Supervises machine shop personnel. (At this time you are not concerned with this area of work. It will be discussed in CDC 53150.)

1-11. *Machine shop technician (AFSC 42770).* The specialty summary of the machine shop technician states that he "designs and machines precision tools, parts, and assemblies; inspects machine work; and supervises machine shop activities." The major duties and responsibilities of the machine shop technician are as follows:

a. Troubleshoots difficult metal machining, design, and production problems.

b. Inspects in-progress and completed machine work for quality of workmanship and serviceability.

c. Instructs in metals machining techniques and in maintenance of machinery and equipment.

1-12. *Fabrication superintendent (AFSC 42799).* The specialty summary of the fabrication superintendent states that he superintends activities engaged in testing, fabricating, repairing, and machining metals and metal products; repairing parachutes and rubberized survival equipment; corrosion control in missile, aircraft, and support system equipment; and nondestructive inspection of aerospace material parts, components, and pressurized systems. The major duties and responsibilities of the fabrication superintendent are as follows:

a. Plans and organizes fabrication activities

b. Directs fabrication activities

c. Establishes and conducts on-the-job training for fabrication personnel.

d. Inspects and evaluates fabrication activities

e. Performs technical fabrication activities.

1-13. **Coordinating with Other Shops.** A field maintenance organization is made up of people working in various career fields. The machine shop and the other fabrication shops are usually housed in the same building or central area. This arrangement permits close coordination of work with other maintenance activities. Since a large part of all machine shop work is in support of other maintenance activities, the machine shop must coordinate its work with other maintenance shops. As an example, an engine mechanic may have a broken stud which you are required to remove. The hydraulic or instrument mechanic may need a special tool or a part which you are required to make Materials and supplies have to be obtained through the supply section Close coordination and cooperation between maintenance activities are needed if you are to get the job done

2. On-the-Job Training

2-1. What is on-the-job training (OJT), what is its purpose, and how does it work? Since the Air Force has to train thousands of airmen in various career fields and AFSs, it is neither practical nor economical to send all airmen to formal training schools for their 3-skill-level AFSC. In many specialties the Air Force can benefit from the work the students do while they are learning. When they attend a formal training school, some productive maintenance work is lost while they are in training Both formal schooling and OJT have certain advantages, depending upon the type of job.

2-2 **Dual Channel Concept of OJT.** The dual channel concept of OJT consists of two parts: career development and job proficiency development The first part consists of studying a Career Development Course (CDC) By studying the CDC the trainee learns the information he needs if he is to do the various duties and tasks of his AFSC In the second part he develops his skill by using equipment and by doing the jobs required in his AFSC He does both parts at the same time He must satisfactorily complete both parts before he is to be upgraded to the AFSC for which he is training

2-3. *Career development* CDCs are correspondence courses based upon the specialty job description in AFM 39 1 and the related Specialty Training Standard (STS) They include general Air Force subjects, specialty theory, and fundamentals and knowledge requirements for the airman's career progression in the AFSC of his assignment. The subject matter content of CDCs is prepared by Air Training Command The CDCs are published and administered by the Extension Course Institute (ECI) under the direction of the Air University.

2-4. *Job proficiency development.* Job Proficiency Guides (JPGs) are a means by which airmen can attain proficiency by performing tasks of their specific assignment This training uses the principle of "learning by doing," under the guidance of a qualified person A JPG is used to develop each trainee's job proficiency and is required for training in his job The JPG identifies specific tasks or duties to be performed and the degree of skill to be attained. The JPG indicates a supervisor's acknowledgment of a student's satisfactory achievement of required tasks and duties It also contains necessary study reference materials. STSs are readily adjusted for use as JPGs. When STSs are used they should be so identified. A separate continuation sheet is prepared, if required Supervisors must certify job proficiency as a prerequisite for upgrading actions

2-5 **Use of CDCs for Upgrade Training.** AFR 50 23 makes it mandatory for you to enroll in this CDC for upgrade training in your AFSC Your supervisor will see that your training is conducted in accordance with regulations and that proper procedures are followed. It is up to you to make the most of your training. No matter how good the training program is, only you can do the learning and develop your skills. Only you can satisfactorily pass the course tests required for upgrading and only you can satisfactorily pass the Skill Knowledge Test (SKT) required for promotion.

2-6. You will be continually enrolled in OJT throughout your career progression. As you satisfactorily complete your training in one AFSC of

CHANGES FOR THE TEXT: VOLUME 2

Pen-and-Ink Changes:

Page-Col	*Paragraph*	*Line(s)*	*Correction*
Cover		4	Change "(AFSC 53150)" to "(AFSC 42730)."
iii	Preface	11	Change "(TTOC)" to "(TTGOX." Change zip code for Chanute to "61868."
iii	Preface	16	Change "Gunter AFB, AL 36114" to "Gunter AFS AL 36118."
2R		6	Change "clu'ces" to "clutches."
7	Fig. 8		Change the legend as follows: A. Side Rake Angle B. Back Rake Angle C. Outside Diameter D. Finished Surface E. Center Line of Work F. Effective End Relief G. Wedge Angle H. Side Relief Angle
25L	3-36*f*	2	Change "frictition" to "friction."
28R	4-14	5	Change "0 05" to "0.0005."
47R		4	Delete "shown in table 3."
48R	8-30	last	Change "took" to "tool."
72l		9	Change "0.0005" to "0.0001."
73	**Bibliography, Dept AF Publications**	1	Change "AFM 127-101, *Accident Prevention Handbook*" to "AFR 127 101, *Ground Accident Prevention Handbook*."
73	**Resident Course**		Delete "**Resident Course**" and listed materials.

CHANGES FOR THE TEXT: VOLUME 3

Pen-and-Ink Changes:

Page-Col	*Paragraph*	*Line(s)*	*Correction*
Cover			Under "Apprentice Machinist" add "(AFSC 42730)."
iii	Preface	13	Change "(TTOC)" to " TTGOX "
8R	1-28	7	Change "produed" to "produced "
8R	1-28	15	Change "is" to "in "
9I	2-4	4	Change "convesient" to "convenient "
18I	4-7	25	Change "wtih" to "with "
28R	7-3,g	1	Change "amoust" to "amount "
34R	7-18	2	Change "sheel" to "wheel."
34R	7-19	last	Change "is" to "in."
42R	Fig. 85		Change "7 16" R" to "1 7 16" R" and "1/2" R" to "2½" R."
43R	9-1	3	Change "Fling" to "Filing "
49	**Bibliography**		Delete "**USAFI Courses**" and listed material
49	**Bibliography, Dept AF Publications**	1	Change "AFM 127 101, *Accident Prevention Handbook*" to "AFR 127 101, *Ground Accident Prevention Handbook* "
49	**Resident Course**		Delete "**Resident Course**" and listed material

CHANGES FOR THE TEXT: VOLUME 4

Pen-and-Ink Changes:

Page-Col	*Paragraph*	*Line(s)*	*Correction*
Cover		4	Change "(AFSC 53150)" to "(AFSC 42730)."
24R		8 fr bot	Change "$1\,^{6}/_{8}$" to "$1\,^{6}/_{9}$."
44l	9-1	2	Change "grant" to "large."
59l	10-25, *Example*	3	Change "grindng" to "grinding."
65l	12-10	5	Change "wpring" to "spring" and "expension" to "expansion."
73	Fig. 116 (chart above illustration)		Change all "Clearance angle B (degrees)" entries from "30" to "3 to 5."
83L	15-9	9	Change "536X0" to "427X2."
90R	16-17	17	Change "tank" to "tang."
93R	18-3	6	Change "ships" to "chips."
97	**Bibliography, Dept AF Publications**	1	Change "AFM 127-101, *Industrial Safety and Accident Prevention Handbook*" to "AFR 127-101, *Ground Accident Prevention Handbook*."
97	**Bibliography**		Delete "**Resident Course**" and listed materials.

AU GAFS, AL (803989) 500